

Numerical Investigation of Fluid Flow and Heat Transfer Characteristics in Parallel Flow Single Layer Microchannels

O. Asgari¹ and M.H. Saidi^{1,*}

Heat generation from Very Large-Scale Integrated (VLSI) circuits increases with the Abstract. development of high-density integrated circuit technology. One of the efficient techniques is liquid cooling by using a microchannel heat sink. Numerical simulations on the microchannel heat sink in the literature are mainly two dimensional. The purpose of the present study is to develop a three-dimensional procedure to investigate flow and conjugate heat transfer in the microchannel heat sink for electronic packaging applications. A finite volume numerical code with a multigrid technique, based on an additive correction multigrid (AC-MG) scheme, which is a high-performance solver, is developed to solve the steady incompressible laminar Navier-Stokes (N-S) equations over a colocated Cartesian grid arrangement. The results show that the thermophysical properties of the liquid can essentially influence both the flow and heat transfer in the microchannel heat sink. Comparison of the numerical results with other published numerical results and experimental data, available in the literature for Reynolds numbers less than 200, indicates that the assumption of hydrodynamically fully developed laminar flow is valid. The accuracy of the prediction has been verified by comparing the results obtained here with the numerical and analytical results from the open literature which showed a good agreement. The detailed temperature and heat flux distributions, as well as the average and bulk heat transfer characteristics, are reported and discussed. The analysis provides a unique fundamental insight into the complex heat flow pattern established in the channel due to combined convection-conduction effects in the three-dimensional setting.

Keywords: Microchannel; Finite volume numerical simulation; Multigrid technique; Colocated grid arrangement; Heat transfer and fluid flow; Electronic cooling.

INTRODUCTION

In the current state of fast developing electronic equipment, having high speeds and at the same time high heat generation rates, researchers are trying to find an effective microcooling method. The main problem in the cooling of microdevices is their high heat generation rate in a limited space. Therefore, much research has been conducted on the analysis of different novel microcooling methods, such as microheat pipes, microjet impingements, microcapillary pumped loops, microelectrohydrodynamic coolers and microchannel heat sinks. Among different microcooling devices, the microchannel heat sink has been of special consideration due to its capabilities such as its high capacity of heat removal. In principle, microchannels can be machined at the back of a substrate of electronic chips and, therefore, can reduce the internal (contact) thermal resistance of the heat sink practically to zero. The cooling rates in such microchannel heat exchangers should increase significantly due to a decrease in the convective resistance to heat transport caused by a drastic reduction in the thickness of thermal boundary layers.

The overall excellent potential capacity of such a heat sink for heat dissipation is based on the large heat transfer surface-to-volume ratio of the microchannel heat exchangers. The use of a microchannel heat sink was first introduced by Tuckerman and Pease in 1981 [1]. Their research was based on an experiment where they showed that high heat rates of 790 W/cm² could be removed by microchannel heat sinks.

^{1.} Center of Excellance in Energy Conversion, Department of Mechanical Engineering, Sharif University of Technology, Tehran, Iran.

 $^{*. \} Corresponding \ author. \ E\text{-mail: } saman@sharif.edu$

Received 2 October 2007; received in revised form 7 September 2008; accepted 9 February 2009

Currently, research in the field of microchannels is being undertaken using three aspects: experimental methods [2-4], numerical methods [5-8] and analytical methods [9-12]. It is well known that the crosssectional shape of a channel can have a significant effect on the fluid flow and heat transfer characteristics in microchannels. Peng and Peterson [13] performed experimental investigations of the pressure drop and convective heat transfer for water flowing in rectangular microchannels and found that the cross-sectional aspect ratio had a significant influence on the flow friction and convective heat transfer in both laminar and turbulent flows. Kawano et al. [14] provided experimental data on the friction and heat transfer in rectangular, silicon-based microchannel heat sinks and, more recently, Wu and Cheng [15,16] conducted a series of experiments to measure the friction factor and convective heat transfer in smooth silicon microchannels with a trapezoidal cross-section. The experimental method has its own values and new developments in micromachining techniques help researchers to perform precise experiments. However, because of its high cost, it fails to be commonly applied.

There has been much effort to analytically model the microchannel heat sink. Knight et al. [9] used empirical correlations to evaluate the performance of a microchannel heat sink. Koh and Colony [17] first modeled microstructures as a porous medium using Darcy's law. Later, Tien and Kuo [18] developed a model for heat transfer in microchannels using a modified Darcy equation and the two-equation model. Another method to model the microchannel heat sink analytically is to use the fin and plate theory, in which the solid walls are assumed to be as fins that are connected to the base plate. Applying this assumption, the thermal resistance and other thermal characteristics of the system could be derived using the fin and plate heat transfer relations. Because of limits and less accuracy in the analytical approach in some phenomena, e.g. conjugate heat transfer, the numerical method is being used vastly to simulate the performance of microchannels as in other fields of heat transfer.

Webb and Zhang [19] claim that the accepted single-phase flow correlations adequately predict their experimental data for round and rectangular tubes with hydraulic diameters of the order of 1 mm. Wesberg et al. [8] solved a two-dimensional conjugate heat transfer problem for microchannel heat sinks to obtain detailed spatial distributions of the temperature of the heat sink cross-section along the length of the channels. Fedorov and Viskanta [6] developed a threedimensional model to investigate the conjugate heat transfer in a microchannel heat sink, with the same channel geometry used in the experimental work done by Kawano et al. [14]. This simulation showed that the average channel wall temperature along the flow direction was nearly uniform, except in the region close to the channel inlet where very large temperature gradients were observed. Qu and Mudawar [5] conducted a three-dimensional fluid flow and heat transfer analysis for a rectangular microchannel heat sink with a geometry similar to that of Kawano et al. [14], using a numerical method similar to that proposed by both Kawano et al. [14] and Fedorov and Viskanta [6].

The thermal response of a microchannel heat sink cooling a device, with power generation varying both dimensionally and with time, is studied in [20] in which a full 3-dimensional numerical model is developed and validated in order to analyze the thermal behavior of an entire microprocessor and the microchannel heat sink employed. The effects of implementation of two-way fluid flow wherein neighboring channels in the sink have opposite fluid flow directions are also investigated.

Kim [21] presents three optimization methods by which the thermal resistance of the microchannel heat sink can be minimized: the fin model, the porous medium model and the numerical optimization method. Assumptions used in the fin model are shown to be invalid for large values of the aspect ratio, while the porous medium model is shown to accurately predict the thermal performance of the microchannel heat sink. Due to this defect, the fin model fails to provide the design variables. In addition, the optimized design variables and the corresponding thermal resistances are presented, using optimization methods based on the porous medium model and numerical simulation under the constraint of maximum pumping power.

Cheng [22] studies a stacked two-layer microchannel heat sink with an enhanced mixing passive microstructure. To simulate the conjugated heat transfer among the heat sink and fluid, the three-dimensional conjugated model is used to solve this problem. The important parameters (e.g. the ratio of embedded structure height to microchannel height and fluid property) are investigated.

Wang et al. [23] demonstrated a novel microchannel heat sink with a high local heat transfer efficiency contributed by a complicated microchannel system, which comprises parallel longitudinal microchannels etched in a silicon substrate and transverse microchannels electroplated on a copper heat spreader. The thermal boundary layer is developed in transverse microchannels. Meanwhile, the heat transfer area is increased, compared to the conventional microchannel heat sink, only having parallel longitudinal microchannels. Both benefits yield high local heat transfer efficiency and enhance overall heat transfer, which is attractive for the cooling of high heat flux electronic devices.

Wong and Ghazali [24] discuss the numerical sim-

ulation of a micro-channel heat sink in microelectronics A three-dimensional Computational Fluid cooling. Dynamics (CFD) model was built using the commercial package, FLUENT, to investigate the conjugate fluid flow and heat transfer phenomena in a silicon-based rectangular microchannel heat sink. The model provided detailed temperature and heat flux distributions in the microchannel heat sink. The results indicate a large temperature gradient in the solid region near the heat source. The highest heat flux is found at the side walls of the microchannel, followed by top wall and bottom wall, due to wall interaction effects. Silicon is proven to be a better microchannel heat sink material compared to copper and aluminum and that is indicated by a higher average heat transfer. A higher aspect ratio in a rectangular microchannel gives higher cooling capability, due to a high velocity gradient around the channel when the channel width decreases.

This model considered the hydrodynamic and thermal developing flow along the channel and found that the Reynolds number will influence the length of the developing flow region. It was also found that the highest temperature is typically encountered at the heated base surface of the heat sink immediately adjacent to the channel outlet and that the temperature rise along the flow direction in the solid and fluid regions can both be approximated as linear.

In this study, the SIMPLE pressure correction approach has been chosen. For simplicity, the equations are solved in a segregated manner. An effective multigrid solver is developed for the pressure correction equation. Methods for calculating cell-face velocities are studied. Convection and diffusion terms are modeled using upwind and the central difference stencil, respectively. The variables are located in the cellcentred colocated grid. The general coordinates are described using the pure control volume approach with vector notations. In the method developed herein, the velocity field is first solved and then the mean velocity is derived. The thermophysical properties are chosen at a reference temperature (an estimated averaged liquid bulk temperature) determined from the energy balance and then a new final velocity field and temperature distribution for the microchannel is defined. A numerical model with fully developed flow is presented and used to analyze the heat transfer in a microchannel heat sink for low Re numbers. The numerical model is based upon a three dimensional conjugate heat transfer approach (3D fluid flow and 3D heat transfer). Computations were performed for different total pressure drops in the channel. A system of three-dimensional Navier-Stokes equations for conservation of mass, momentum and energy, based on the continuum flow assumption, is used as a mathematical model of the process.

ANALYSIS

Problem Description

The micro-heat sink modeled in this investigation consists of a 10 mm long silicon substrate with a silicon cover. The rectangular microchannels have a width of 57 μ m and a depth of 180 μ m. The hydraulic diameter of the microchannel under development is about 86.58 μ m and is expected to decrease to 10 μ m. This yields a typical Knudsen number (Kn = λ/L) for water, as a coolant, to be between 3.5×10^{-5} and 3.5×10^{-4} , which lies in the continuous flow regime (Kn < 10^{-3}) [25]. Therefore, the conservation equations based on the continuum model (Navier-Stokes equations of motion) can still be used to describe the transport processes.

A schematic representation of the rectangular microchannel heat sink is shown in Figure 1, where a unit of cell consisting of one channel was selected because of the symmetry of the structure. The unit cell of the heat sink can be represented by an asymmetric rectangular channel with cross-sectional dimensions, as shown in Table 1. The channel geometry is similar to that employed in the experimental work of Kawano et al. [14] and in the numerical investigations of Qu and Mudawar [5] and Fedorov and Viskanta [6]. It is assumed that the heat flux input at the bottom of the heat sink is uniform.

Model Equations and Boundary Conditions

Consider a steady 3D flow in a silicon microchannel heat sink with heating from below and with adiabatic conditions at the other boundaries, as shown in Figure 1. There are some assumptions in this numerical simulation, e.g. the transport processes are considered to be steady-state and three-dimensional, the flow is incompressible and laminar, thermal radiation is neglected and thermophysical properties are temperature dependent. The governing equations and related boundary conditions for a fully developed 3D, incompressible, steady state flow and heat transfer are given as [26]:

• Continuity equation:

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

Table 1. Geometric dimensions of the unit cell.

H	L_{x}	L_z	L_y	D_h	W_w	W
(μm)	(mm)	(μm)				
180	10	100	900	86.58	43	57



Figure 1. Schema of a rectangular microchannel heat sink and the unit of cell.

• Momentum equations:

$$\begin{split} \rho \left[\frac{\partial}{\partial x} (u^2) + \frac{\partial}{\partial y} (uv) + \frac{\partial}{\partial z} (uw) \right] &= -\frac{\partial p}{\partial x} \\ + \left[\frac{\partial}{\partial x} \left(\mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mu \frac{\partial u}{\partial z} \right) \right], \\ \rho \left[\frac{\partial}{\partial x} (vu) + \frac{\partial}{\partial y} (v^2) + \frac{\partial}{\partial z} (vw) \right] &= -\frac{\partial p}{\partial y} \\ + \left[\frac{\partial}{\partial x} \left(\mu \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mu \frac{\partial v}{\partial z} \right) \right], \\ (3) \end{split}$$

O. Asgari and M.H. Saidi

$$\rho \left[\frac{\partial}{\partial x} (wu) + \frac{\partial}{\partial y} (wv) + \frac{\partial}{\partial z} (w^2) \right] = -\frac{\partial p}{\partial z} + \left[\frac{\partial}{\partial x} \left(\mu \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mu \frac{\partial w}{\partial z} \right) \right]_{(4)}^{\cdot}$$

• Energy equation:

$$\rho \left[\frac{\partial}{\partial x} (uT) + \frac{\partial}{\partial y} (vT) + \frac{\partial}{\partial z} (wT) \right] = \frac{\partial}{\partial x} \left(\frac{k}{c_p} \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(\frac{k}{c_p} \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(\frac{k}{c_p} \frac{\partial T}{\partial z} \right),$$
(5)

where $u, v, w, p, \rho, \mu, T, k$ and c_p are the velocity of *x*-direction, *y*-direction, *z*-direction, pressure, density, dynamic viscosity, temperature, thermal conductivity and specific heat at constant pressure, respectively. The hydrodynamic boundary conditions are as follows:

at the inner wall surface (no slip):

 $u = 0, \qquad v = 0, \qquad w = 0,$ (6)

at the inlet of the channel:

$$x = 0, \qquad p_f = p_{\rm in}, \qquad v = 0, \qquad w = 0,$$
 (7)

at the outlet of the channel:

$$x = L_x, \qquad p_f = p_{\text{out}} (1 \text{ atm}), \qquad v = 0,$$
$$w = 0, \tag{8}$$

the heat conduction in the solid section is:

$$\frac{\partial}{\partial x} \left(k \frac{\partial T}{\partial x} \right)_{S} + \frac{\partial}{\partial y} \left(k \frac{\partial T}{\partial y} \right)_{S} + \frac{\partial}{\partial z} \left(k \frac{\partial T}{\partial z} \right)_{S} = 0.$$
(9)

The thermal boundary conditions can be stated as:

$$0 \le x \le L_x, \qquad y = 0,$$

$$0 \le z \le L_z \Rightarrow -k_S \frac{\partial T_S}{\partial y} = q_S, \qquad (10)$$

$$0 \le x \le L_x, \qquad y = L_y,$$

$$0 \le z \le L_z \Rightarrow -k_S \frac{\partial T_S}{\partial y} = 0, \qquad (11)$$

Fluid Flow and Heat Transfer Characteristics

$$x = 0 \begin{cases} (L_y/2 - H) \le y \le L_y/2 \\ W_w/2 \le z \le (W_w/2 + W) \end{cases} \Rightarrow T_f = T_{\rm in}$$
(12)
else $\Rightarrow -k_S \frac{\partial T_S}{\partial x} = 0$
$$x = L_x \begin{cases} (L_y/2 - H) \le y \le L_y/2 \\ W_w/2 \le z \le (W_w/2 + W) \end{cases} \Rightarrow -k_f \frac{\partial T_f}{\partial x} = 0$$
(13)

else
$$\Rightarrow -k_S \frac{\partial T_S}{\partial x} = 0$$

 $0 \le x \le L_x, \qquad 0 \le y \le L_y,$
 $z = 0 \Rightarrow -k_S \frac{\partial T_S}{\partial z} = 0,$ (14)
 $0 \le x \le L_x, \qquad 0 \le y \le L_y,$

$$z = L_z \Rightarrow -k_S \frac{\partial T_S}{\partial y} = 0, \tag{15}$$

at the inner wall surface:

$$-k_{S}\left(\frac{\partial T_{S}(x,y,z)}{\partial n}\bigg|_{\Gamma}\right) = -k_{f}\left(\frac{\partial T_{f}(x,y,z)}{\partial n}\bigg|_{\Gamma}\right),$$
$$T_{s,\Gamma} = T_{f,\Gamma},$$
(16)

where Equation 10 gives the uniform heat flux boundary condition at the bottom wall of the substrate. Equations 11 to 15 assume no heat loss from the solid to the environment at the boundary, except at x = 0for the fluid, where $T_f = T_{in}$. Equation 13 assumes a thermally fully developed flow at the outlet of the microchannel. It should be noted that in reality heat losses from the heat sink to the environment should be considered by conduction and convection at the inlet and outlet and at the top surface of the heat sink. Heat transfer in the unit cell is a conjugate one, combining heat conduction in the solid and convection to the cooling fluid. The two heat transfer modes are coupled by continuities of temperature and heat flux at the interface between the solid and fluid, which are expressed by Equation 16. Γ denotes the perimeter of the inner wall of the channel. Equations 1 to 5 form a closed system from which the flow properties, u, v, w, pand T, can be solved as a function of space and time. However, in this study, only steady-state flows will be calculated.

Calculation of Incompressible Flows

Simultaneous numerical calculation of Equations 1 to 5 is computationally complex. Therefore, the equations are solved one after another, i.e. in a segregated

manner. The basic structures of Equations 2 to 5 are similar to each other, containing an unsteady term, convection, diffusion and possible source terms, and they are often called convection-diffusion equations. The flow properties, u, v, w and T, are solved from Equations 2 to 5, respectively. Therefore, continuity Equation 1 is to be modified for a pressure or pressurelike quantity. The first stage is to derive a convectiondiffusion equation up to finite form. In this study, the control volume approach is utilized. The process is to be studied by the aid of a general convection-diffusion equation for quantity Φ :

$$\frac{\partial}{\partial t}(\rho\phi) + \frac{\partial}{\partial x}(\rho u\phi) + \frac{\partial}{\partial y}(\rho v\phi) + \frac{\partial}{\partial z}(\rho w\phi)$$

$$= \frac{\partial}{\partial x}\left(\alpha\frac{\partial\phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\alpha\frac{\partial\phi}{\partial y}\right)$$

$$+ \frac{\partial}{\partial z}\left(\alpha\frac{\partial\phi}{\partial z}\right) + S,$$
(17)

where α is diffusion coefficient and source term *S* could contain, for instance, pressure gradient and/or body force, etc. Next, Equation 17 is integrated over the control volume, $V_{i,j,k}$. After the rearrangements, the integrated equation can be obtained in the following form:

$$V_{i,j,k} \frac{\partial (\rho \phi)_{i,j,k}}{\partial t} + F_{i+1/2,j,k} - F_{i-1/2,j,k} + F_{i,j+1/2,k} - F_{i,j-1/2,k} + F_{i,j,k+1/2} - F_{i,j,k-1/2} = S_{i,j,k},$$
(18)

$$F_{i-1/2,j,k} = \left[A \left(\rho u \phi - \alpha \frac{\partial \phi}{\partial x} \right) \right]_{i-1/2,j,k}, \qquad (19)$$

where A is a surface area of a face of the cell. From a physical point of view, the convection transfers the information downstream. Therefore, the approximation of convection terms must be weighted to the up-stream side. The simplest stencil is the First-Order Upwind (FOU):

$$\phi_{i-1/2,j,k} = \phi_{i-1,j,k} + O(\Delta x); \quad u_{i-1/2,j,k} > 0, \quad (20)$$

$$\phi_{i-1/2,j,k} = \phi_{i,j,k} + O(\Delta x); \qquad u_{i-1/2,j,k} < 0.$$
 (21)

The diffusion terms of Equation 19 are approximated by using the central difference scheme:

$$\left[\frac{\partial\phi}{\partial x}\right]_{i+1/2,j,k} = \frac{\phi_{i+1,j,k} - \phi_{i,j,k}}{\Delta x} + O(\Delta x^2), \qquad (22)$$

where the grid is assumed to be uniform. Also, the

pressure gradient terms of the momentum equations are approximated by using central differencing. After integration over control volume $V_{i,j,k}$, they are obtained in the form:

$$\int_{V_{i,j,k}} \frac{\partial p}{\partial x} d\Omega = (pA)_{i+1/2,j,k} - (pA)_{i-1/2,j,k}, \qquad (23)$$

where central difference schemes, like;

$$p_{i+1/2,j,k} = 0.5(p_{i,j,k} + p_{i+1,j,k}) + O(\Delta x^2), \qquad (24)$$

are applied.

To close the partial differential Equation system 1 to 5 for a certain problem, the boundary conditions must be specified. In the present solver, the boundary conditions are handled by using ghost cells, which are illustrated in Figures 2 and 3. The principal idea is to use ghost cell values to give the fixed boundary value at the boundary of the domain, i.e. the ghost cell values are extrapolated from the face and domain values. Thus, the Dirichlet and Neumann conditions are given in the form:

$$\phi_{2,j,k} = 2\phi_{\text{face}} - \phi_{3,j,k}, \tag{25}$$

$$\phi_{2,j,k} = \phi_{3,j,k} - \Delta x \frac{\partial \phi}{\partial x}\Big|_{\text{face}},\tag{26}$$

respectively.

Treatment of Pressure in Incompressible Navier-Stokes Equations

Spatial oscillations occur when central differencing is applied to both the continuity equation and the pressure gradient term in the momentum equations.



Figure 2. Ghost cells around the domain.



Figure 3. The notations of the ghost cells.

The momentum equations at the even-numbered nodes depend only on pressures at odd-numbered nodes, and vice versa. The same holds for the continuity equation. This situation permits two different pressure fields to co-exist, which is known as a checkerboard pressure field.

Nowadays, the staggered grid arrangement is no longer necessary. The remarkable turn toward the colocated grid arrangement was the study presented by Rhie and Chow [27]. In the colocated grid arrangement, all variables use the same grid and interpolation is needed. As already mentioned, the colocated grid arrangement causes problems when central differencing is applied to cell-face velocities; Rhie and Chow [26] presented a method for avoiding the usage of the staggered grid arrangement. In this method, central differencing has been applied to the pressure gradient and cell-face pressure, while the Rhie & Chow interpolation has been applied to the cell-face velocity, as follows:

$$u_{i-1/2,j,k} = \frac{1}{2} (u_{i-1,j,k} - u_{i,j,k}) - \frac{1}{2} \left(\frac{V_{i-1,j,k}}{a_{i-1,j,k}} + \frac{V_{i,j,k}}{a_{i,j,k}} \right) \times \frac{\partial p}{\partial x} \Big|_{i-1/2,j,k} + \frac{1}{2} \left(\frac{V_{i-1,j,k}}{a_{i-1,j,k}} \frac{\partial p}{\partial x} \Big|_{i-1,j,k} + \frac{V_{i,j,k}}{a_{i,j,k}} \frac{\partial p}{\partial x} \Big|_{i,j,k} \right).$$
(27)

Pressure Correction Equation and Multigrid Technique

The current research indicates that the AC-MG acceleration technique is highly efficient, reliable and robust,

which makes it feasible for CPU-intensive computations such as pressure correction equations. When compared with discretized momentum equations, the pressure Poisson equations tend to be very stiff and ill-conditioned, i.e. $a_p \equiv \sum_{nb} a_{nb}$. Because of these reasons, solving the pressure Poisson equation is usually the CPU bottle-neck for the incompressible N-S equation system, and an AC-MG technique is required. With this acceleration technique, the residuals of the large-scale algebraic equation system are guaranteed to be continuously driven down to the level of the computer machine round-off error, warranting strong conservations of mass and momentum satisfied over all the control volumes. In this cell-centered multigrid algorithm, both restriction and prolongation operators are based on piecewise constant interpolation.

The additive-correction multigrid scheme, described in [28], is used for the two cross-streamwise directions (y and z). The cell-centered two-level multigrid configuration is sketched in Figure 4 in which the grid point (i j k) on the coarse level is surrounded by four grid points on the fine level in the y and z directions, namely (i, 2j - 2, 2k - 2), (i, 2j - 2, 2k - 1), (i, 2j - 1, 2k - 2) and (i, 2j - 1, 2k - 1). The variables on the coarse and fine levels are denoted by superscripts c and f, respectively, in the following equations. Based on the idea introduced in [28], the following algebraic equation system can be used to determine the correction for the fine grid level:

$$a_{p(i,j,k)}^{c}\phi_{i,j,k}^{c} = a_{e(i,j,k)}^{c}\phi_{i+1,j,k}^{c} + a_{w(i,j,k)}^{c}\phi_{i-1,j,k}^{c}$$

$$+ a_{n(i,j,k)}^{c}\phi_{i,j+1,k}^{c} + a_{S(i,j,k)}^{c}\phi_{i,j-1,k}^{c}$$

$$+ a_{t(i,j,k)}^{c}\phi_{i,j,k+1}^{c} + a_{b(i,j,k)}^{c}\phi_{i,j,k-1}^{c}$$

$$+ Sc_{i,j,k}^{c}, \qquad (28)$$

where the coefficients on the coarse grid are calculated from the following restriction formulae:

$$a_{e(i,j,k)}^{c} = a_{e(i,2j-2,2k-2)}^{f} + a_{e(i,2j-2,2k-1)}^{f} + a_{e(i,2j-1,2k-2)}^{f} + a_{e(i,2j-1,2k-1)}^{f},$$
(29)

$$a_{w(i,j,k)}^{c} = a_{w(i,2j-2,2k-2)}^{f} + a_{w(i,2j-2,2k-1)}^{f} + a_{w(i,2j-1,2k-2)}^{f} + a_{w(i,2j-1,2k-1)}^{f},$$
(30)

$$a_{n(i,j,k)}^{c} = a_{n(i,2j-1,2k-2)}^{f} + a_{n(i,2j-1,2k-1)}^{f},$$
(31)

$$a_{S(i,j,k)}^{c} = a_{S(i,2j-2,2k-2)}^{f} + a_{S(i,2j-2,2k-1)}^{f},$$
(32)



Figure 4. Schema of a cell-centered two-level multigrid configuration.

$$a_{t(i,j,k)}^{c} = a_{t(i,2j-2,2k-1)}^{f} + a_{t(i,2j-1,2k-1)}^{f}, \qquad (33)$$

$$a_{b(i,j,k)}^{c} = a_{b(i,2j-2,2k-2)}^{f} + a_{b(i,2j-1,2k-2)}^{f}, \qquad (34)$$

$$\begin{aligned} a_{p(i,j,k)}^{c} &= a_{p(i,2j-2,2k-2)}^{f} + a_{p(i,2j-2,2k-1)}^{f} \\ &+ a_{p(i,2j-1,2k-2)}^{f} + a_{p(i,2j-1,2k-1)}^{f} \\ &- (a_{S(i,2j-1,2k-2)}^{f} + a_{S(i,2j-1,2k-1)}^{f}) \\ &- (a_{n(i,2j-2,2k-2)}^{f} + a_{n(i,2j-2,2k-1)}^{f}) \\ &- (a_{b(i,2j-2,2k-1)}^{f} + a_{b(i,2j-1,2k-1)}^{f}) \\ &- (a_{t(i,2j-2,2k-2)}^{f} + a_{t(i,2j-1,2k-2)}^{f}), \end{aligned}$$
(35)

$$Sc_{i,jk}^{c} = r_{i,2j-2,2k-2}^{f} + r_{i,2j-2,2k-1}^{f} + r_{i,2j-1,2k-2}^{f} + r_{i,2j-1,2k-1}^{f},$$
(36)

The residuals on fine grid level $r_{i,j,k}^{f}$ are calculated from the current iterative values of $\hat{\phi}_{i,j,k}^{f}$, using the following relation:

$$r_{i,j,k}^{f} = -a_{p(i,j,k)}^{f} \hat{\phi}_{i,j,k}^{f} + a_{e(i,j,k)}^{f} \hat{\phi}_{i+1,j,k}^{f} + a_{n(i,j,k)}^{f} \hat{\phi}_{i,j+1,k}^{f} + a_{t(i,j,k)}^{f} \hat{\phi}_{i,j,k+1}^{f} + a_{w(i,j,k)}^{f} \hat{\phi}_{i-1,j,k}^{f} + a_{S(i,j,k)}^{f} \hat{\phi}_{i,j-1,k}^{f} + a_{b(i,j,k)}^{f} \hat{\phi}_{i,j,k-1}^{f} + Sc_{i,j,k}^{f}.$$
(37)

A typical two-level multigrid iterative algorithm consists of restriction, relaxation on the coarse grid and prolongation. After a number of relaxation sweeps such as TDMA sweeps, on the fine grid level, the residuals are calculated using Equation 37 and are restricted to the coarse grid using Equations 29 to 36. The restricted residuals are then used as the source terms in Equation 28 and relaxation sweeps are used to solve Equation 28 on the coarse level. The solutions to Equation 28 are then utilized as the corrections to be prolongated back to the fine grid, using the following relations to update the current iterative solution of $\hat{\phi}^f_{i,i,k}$:

$$\begin{aligned} \hat{\phi}_{i,2j-2,2k-2}^{f} &= \hat{\phi}_{i,2j-2,2k-2}^{f} + \phi_{i,j,k}^{c}, \\ \hat{\phi}_{i,2j-2,2k-1}^{f} &= \hat{\phi}_{i,2j-2,2k-1}^{f} + \phi_{i,j,k}^{c}, \\ \hat{\phi}_{i,2j-1,2k-2}^{f} &= \hat{\phi}_{i,2j-1,2k-2}^{f} + \phi_{i,j,k}^{c}, \\ \hat{\phi}_{i,2j-1,2k-1}^{f} &= \hat{\phi}_{i,2j-1,2k-1}^{f} + \phi_{i,j,k}^{c}. \end{aligned}$$
(38)

Equation 38 provides the prolongation formulation in the AC multigrid calculation. Obviously, restriction (Equations 29 to 36) and prolongation (Equation 38) are second-order accuracy in space and no extra interpolations are needed. The AC-MG solution procedure contains two parts. The first part consists of a subroutine of TDMA sweeps which is controlled by the residual convergence rate. A flowchart of the TDMA solver is depicted in Figure 5. If the total residual before iteration n on the current mesh is $R^n = \sum_{i,j,k} r_{i,j,k}^n$ and the residual after iteration nis R^{n+1} , then another TDMA sweeping iteration is performed, if the residual convergence rate satisfies $R^{n+1} \leq f$. R^n , where the value for f is usually set



Figure 5. Flowchart of the TDMA solver controlled by the residual convergence rate.

to 0.5 [28]. If the convergence rate is lower than 0.5, i.e. $R^{n+1}/R^n > f$, a correction on the coarser grid is required which invokes the second part of the AC-MG solution procedure. A schematic figure of the three-level AC-MG solution procedure will be shown in Figure 6.

In order to check the sensitivity of the numerical results to mesh size, three different grid systems were tested. They consisted of $18 \times 42 \times 16$, $30 \times 82 \times 30$ and $50 \times 162 \times 58$ nodes in the x, y, and z directions, respectively. The results from the last two grid systems were very close to each other and local temperature differences were less than 0.1%. Since less computational time and computer memory were needed for the second grid system, it was employed in the final simulation.

The total grid number is 73,800 ($N_x \times N_y \times N_z$ is 30 × 82 × 30) for the domain. This type of fine grid mesh for the y and z directions was chosen in order to properly resolve the velocity and viscous shear layers, and to more accurately define the conjugate heat transfer at the surface of the channel, thereby improving the temperature resolution. Furthermore, comparison with standard theoretical or numerical results indicates that the finer the mesh size, the higher the numerical accuracy. The reasons for the comparative coarse discretization for the x-direction are:

- (i) With the exception of the inlet region, the temperature gradients are small compared to the gradients occurring in other directions;
- (ii) The CPU time as well as the memory storage required increases dramatically as the number of grid nodes is increased.



Figure 6. A schema of the three-level AC-MG solution procedure.

Fluid Flow and Heat Transfer Characteristics

VALIDATION OF THE CODE

Velocity Field

The velocity field can be determined analytically using a more direct approach. As illustrated in Figure 7, the following relations represent reasonable results for this type of problem by Shah and London [29]:

$$u = -\frac{16c_1 a^2}{\pi^3} \sum_{n=1,3,\dots}^{\infty} \frac{1}{n^3} (-1)^{(n-1)/2} \times \left[1 - \frac{\cosh(n\pi y/2a)}{\cosh(n\pi b/2a)}\right] \cos\left(\frac{n\pi z}{2a}\right),$$
(39)

$$u_m = -\frac{c_1 a^2}{3} \left[1 - \frac{192}{\pi^5} \left(\frac{a}{b} \right) \sum_{n=1,3,\dots}^{\infty} \frac{1}{n^5} \tanh\left(\frac{n\pi b}{2a} \right) \right].$$
(40)

This velocity profile is in excellent agreement with the experimental results. Since Equation 39 involves considerable computational complexity, a simple approximation in the following form for aspect ratio $\alpha^* \leq 0.5$ is proposed [29]:

$$\frac{u}{u_{\max}} = \left[1 - \left(\frac{y}{b}\right)^n\right] \left[1 - \left(\frac{z}{a}\right)^m\right],\tag{41}$$

where m and n are derived from the following relations:

$$m = 1.7 + 0.5(\alpha^*)^{-1.4},\tag{42}$$

$$n = \begin{cases} 2 & \text{for } \alpha^* \le 1/3\\ 2 + 0.3(\alpha^* - 1/3) & \text{for } \alpha^* \ge 1/3 \end{cases}$$
(43)

The integration of Equation 41 over the duct cross section yields:

$$\frac{u}{u_m} = \left(\frac{m+1}{m}\right) \left(\frac{n+1}{n}\right) \left[1 - \left(\frac{y}{b}\right)^n\right] \left[1 - \left(\frac{z}{a}\right)^m\right],\tag{44}$$

$$\frac{u_{\max}}{u_m} = \left(\frac{m+1}{m}\right) \left(\frac{n+1}{n}\right). \tag{45}$$



Figure 7. A schema of the rectangular channel.

With the fRe of Equation 46, the u_m for the rectangular ducts can be expressed in a closed form as:

$$f \operatorname{Re}_{D_h} = 24(1 - 1.3553\alpha^* + 1.9467\alpha^{*2} - 1.7012\alpha^{*3} + 0.9564\alpha^{*4} - 0.2537\alpha^{*5}), \quad (46)$$

$$u_m = \frac{8c_1 a^2}{f \operatorname{Re}_{D_h} [1 + (a/b)]^2},$$
(47)

where:

$$c_1 = \frac{(dp/dx)}{(\mu/g_c)} = \frac{(\Delta p/L_x)}{(\mu/g_c)}.$$
(48)

Substituting Equations 46 and 48 into Equation 47, the mean velocity for a given pressure drop, u_m , can be obtained. Then, using the resulting value for u_m and Equations 42 to 44, the approximate analytical velocity distribution in the microchannel, as shown in Figure 8, can be obtained. The numerically determined velocity profile developed here is illustrated in



Figure 8. Velocity field in channel from the approximate analytical expression Equation 44, $\Delta p = 50$ kPa, Re = 92.68, $T_{\text{reference}} = 20^{\circ}$ C, $u_m = 1.0779$ m/s.



Figure 9. Velocity field in channel from the numerical calculation, $\Delta p = 50$ kPa, Re = 92.383, $T_{\text{reference}} = 20^{\circ}\text{C}$, $u_m = 1.1032$ m/s, $u_{\text{max}} = 1.997$ m/s.

Figure 9. Comparison of the analytical and numerical results indicates that while the numerical code exactly represents the general trend of the results, there is some disparity between the analytical and numerical results. The small difference of velocity profile between Figures 8 and 9 is due to the approximations in the analytical solution described in Equations 41 and 44. But clearly, as evidenced by the magnitude of the mean velocities and the Reynolds numbers obtained from the different methods, the agreement between the two methods is quite good and provides sufficient evidence for validation of the numerical method. In this comparison, the thermophysical properties of water were chosen at a temperature of $293 \text{ K} (20^{\circ} \text{C})$. Because thermophysical properties are temperature dependent, especially liquid viscosity, the velocity and Reynolds numbers are different under the same pressure drop conditions. This issue will be discussed later in more detail

Figure 10 compares the analytical friction coefficient, as determined from Equation 46, with the nu-



Figure 10. Comparison among the numerical calculations, the analytical and the experimental data for the friction coefficient.

merical results obtained using the following procedure (it should be noted that $f_{\text{Darcy}} = 4f$). The mass flow rate is calculated from the velocity as:

$$\dot{m} = \sum \sum \rho_f . u \ (i = \text{constant}, j, k) \Delta y \Delta z,$$
 (49)

and then, the mean velocity is obtained as:

$$u_m = \frac{\dot{m}}{\rho_f A} = \frac{\dot{m}}{\rho_f H W},\tag{50}$$

thus, using Equation 51, the friction factor can be determined as Equation 52:

$$\Delta p = f \frac{4L_x}{D_h} \frac{\rho_f u_m^2}{2},\tag{51}$$

$$f_{\text{Darcy}} \operatorname{Re}_{D_h} = 2 \frac{\Delta p D_h^2}{u_m L_x \mu_f}.$$
(52)

In [30-32], the friction coefficient, f_{Darcy} Re, is determined numerically for different duct cross-sections. For the rectangular channel, with an aspect ratio (height to width) of 3 - 1(H : W), which approximates the geometry used here, $180 \ \mu\text{m} \times 57 \ \mu\text{m}$, the Darcy friction factor-Reynolds number product, f_{Darcy} Re, is 69. The agreement between the numerical calculations here and the calculations obtained by others [30-32] shows that the numerical code developed here is quite accurate for a fully developed laminar flow.

The length required for the formation of a fully developed laminar profile in a microchannel can be estimated by the following analytical relation which is developed for a round tube: Fluid Flow and Heat Transfer Characteristics

$$\frac{L_e}{D_h} = 0.0575 \,\,\mathrm{Re}_{D_h}.\tag{53}$$

For a hydraulic diameter of $D_h = 86.58 \ \mu \text{m}$ and for Re = 160, the entrance length is 796.5. In [30-32], it is noted that the shape of the entrance is very important, with much shorter entrance lengths occurring for square-edged entrances than for rounded ones. Thus, because the flow entrance length may be less than 5% of the total length for a rectangular channel heat sink, the assumption of fully developed laminar flow over the entire length of the microchannel is acceptable for heat transfer analysis, particularly in cases such as this where the Reynolds number is less than 200 (or low mass flow rates).

Heat Transfer

The code was first validated for one-dimensional heat conduction by comparing the results with a 1D analytical solution of heat conduction with a specified boundary condition [30]. The agreement was quite good indicating very good correlation between numerical results and the 1D analytical solution. Secondly, using conservation of energy, it can be shown that the maximum possible temperature rise between the fluid inlet and outlet can be expressed as:

$$\Delta T_{f,\text{con}} = \frac{q_s.A_s}{\dot{m}.c_p}.$$
(54)

In addition, the average temperature rise between the inlet and outlet of the channel can be determined from the numerical analysis as follows:

$$\Delta T_{f.\text{ave}} = \overline{T}_f(x = L_x) - \overline{T_f}(x = 0)$$

$$= \frac{\{\sum \sum \rho_p.u.c_p.T.\Delta y \Delta z\}_{i=\text{outlet},j,k}}{\dot{m}.c_p}$$

$$- \frac{\{\sum \sum \rho_f.u.c_p.T.\Delta y \Delta z\}_{i=\text{inlet},j,k}}{\dot{m}.c_p}. \quad (55)$$

In this work, three different cases $(q_s = 90 \text{ W/cm}^2, \Delta p = 50, 15 \text{ and } 6 \text{ kPa})$ were investigated. Comparison of the results in Table 2 indicates that the difference between $\Delta T_{f,\text{ave}}$ and $\Delta T_{f,\text{con}}$ is small. This issue is also illustrated in Figure 11. Differences of this magnitude can be attributed to:

Table 2. Comparison between $\Delta T_{f,\text{ave}}$ and $\Delta T_{f,\text{con}}$ for $\Delta p = 50, 15$ and 6 kPa, $q_w = 90 \text{ W/cm}^2$.

$\Delta p ~(\mathrm{kPa})$	50	15	6
${ m Re}$	162.68	85.60	47.32
$\Delta T_{f,\mathrm{con}}$ (°C)	14.62	36.82	78.57
$\Delta T_{f,\mathrm{ave}} (^{\circ}\mathrm{C})$	12.79	35.10	77.16



Figure 11. Comparison between the numerical and analytical results for temperature differences upon the Reynolds number.

- (i) Equation 54 is the maximum possible temperature rise in the bulk liquid from the energy balance;
- (ii) The mesh size is not as fine as required (infinitesimal), hence, the accuracy of the statistical result from Equation 55 is limited.

RESULTS AND DISCUSSION

The following four subsections are devoted to local temperature distributions, average and bulk characteristics, local heat flux distributions and the convergence performance for the pressure correction equations in microchannel heat sinks.

Local Temperature Distributions

Figure 12 shows a velocity field in a microchannel at $\Delta p = 50$ kPa. As discussed previously, the thermo-



Figure 12. Velocity field from the numerical calculation, $\Delta p = 50$ kPa, Re = 162.68, $T_{\text{reference}} = 32^{\circ}\text{C}$, $u_m = 1.44$ m/s, $u_{\text{max}} = 2.61$ m/s.

physical properties of the water are based upon the estimated liquid bulk temperature. As can be seen by comparison with the results shown in Figure 9, for the same pressure drop along the channel, thermal properties, velocity profile, mean velocity and Reynolds number are all mainly different.

As shown in Figure 12, a variation in the reference temperature, $T_{\text{reference}}$, from 20 to 32°C changes the mean velocity from 1.1032 to 1.44 m/s and results in a corresponding change in the Reynolds number from 95.38 to 162.68. Numerical results for the temperature distribution in the heat sinks are shown in Figures 13 to 17 for different locations along the channel. Figures 13 to 15 show the local cross-sectional temperature distribution in the y - z plane at x = 0, $x = L_x/2$ and $x = L_x$, respectively. As shown in Figure 13, the temperature of the liquid at the inlet is initially uniform (at 20°C). The temperature profiles



Figure 13. Local temperature distribution in y - z plane at x = 0 ($\Delta p = 50$ kPa, Re = 162.68, T_{reference} = 32°C, $u_m = 1.44$ m/s).



Figure 14. Local temperature distribution in y - z plane at $x = L_x/2$ ($\Delta p = 50$ kPa, Re = 162.68, $T_{\text{reference}} = 32^{\circ}$ C, $u_m = 1.44$ m/s).



Figure 15. Local temperature distribution in y - z plane at $x = L_x$, ($\Delta p = 50$ kPa, Re = 162.68, T_{reference} = 32°C, $u_m = 1.44$ m/s).



Figure 16. Contour of temperature in the heat sink at the cross-section of the outlet of the channel ($\Delta p = 50$ kPa, Re = 162.68, $T_{\text{reference}} = 32^{\circ}$ C, $u_m = 1.44$ m/s).



Figure 17. Contour of local temperature inside the channel at the cross-section of the outlet of the channel $(\Delta p = 50 \text{ kPa}, \text{Re} = 162.68, T_{\text{reference}} = 32^{\circ}\text{C}, u_m = 1.44 \text{ m/s}).$

shown in Figures 14 and 15 are identical in shape due to the assumption of hydrodynamic fully developed flow, but the magnitudes of temperature are different.

Figure 16 shows the temperature contours in the heat sink at the outlet of the channel and Figure 17 shows the local temperatures inside the channel. As shown in Figure 17, it is specified that the temperature is highest at the channel corner. This is due to the low velocity of the flow and the resulting high concentration of heat flux. From these calculations, it is apparent that there is a 2-3°C temperature difference between the bottom wall of the substrate and the bottom surface of the channel.

The temperature distribution can be shown obviously in Figures 18 to 20, which indicates the local temperature distribution in the x-y plane at $z = L_z/2$, for the three cases with $q_s = 90$ W/cm², at $\Delta p = 50$, 15 and 6 kPa, respectively.

The water flow is clearly specified in the above figures which, when combined with Figures 13 to 15, yield detailed information about temperature distribution in the micro-heat sink. The temperature at the outlet, for the case of $\Delta p = 6$ kPa, is higher than 100°C where boiling may occur. Therefore, the numerical solution for a single-phase flow may not be valid there.



Figure 18. Local temperature distribution in x - y plane at $z = L_z/2$ ($\Delta p = 50$ kPa, Re = 162.68, $T_{\text{reference}} = 32^{\circ}\text{C}$, $u_m = 1.44$ m/s).



Figure 19. Local temperature distribution in x - y plane at $z = L_z/2$ ($\Delta p = 15$ kPa, Re = 85.60, T_{reference} = 48°C, $u_m = 0.57$ m/s).



Figure 20. Local temperature distribution in x - y plane at $z = L_z/2$ ($\Delta p = 6$ kPa, Re = 47.32, $T_{\text{reference}} = 57^{\circ}\text{C}$, $u_m = 0.271$ m/s).

The temperature increases along the longitudinal xdirection over the channel inner walls. There are very slight changes in the temperature gradient in the xdirection at the channel bottom wall (Figure 21) and the channel top wall (Figure 22).

In fact, a linear temperature rise can be regarded as a good approximation for both planes. The temperature along the transverse z-direction is virtually constant for all the x - z planes previously discussed.

The temperatures of the side walls vary noticeably in the transverse y-direction. As expected, the temperature decreases from the unit cell bottom wall to the unit cell top wall. Figure 23 shows higher temperatures for small y values close to the channel bottom wall.

Average and Bulk Characteristics

In order to evaluate the local averaged heat transfer characteristics along the flow direction, the convective heat transfer coefficient and Nusselt number must be



Figure 21. Numerical predictions of local temperature distribution in the unit cell bottom wall.



Figure 22. Numerical predictions of local temperature distribution in the unit cell top wall.



Figure 23. Numerical predictions of local temperature distribution in the unit cell side walls.

defined. The longitudinal convective heat transfer coefficient is defined as:

$$\overline{h}_x = \frac{\overline{q}_{s,\Gamma}(x)}{\Delta \overline{T}(x)},\tag{56}$$

and the averaged longitudinal Nusselt number as:

$$\overline{\mathrm{Nu}}_x = \frac{\overline{h}_x \cdot D_h}{k_f}.$$
(57)

The averaged longitudinal local heat flux along the perimeter of the inner wall of the channel in Equation 56 is defined as:

$$\overline{q}_{s,\Gamma}(x) = -k_s \left(\frac{\partial T_s(x, y, z)}{\partial n} \bigg|_{\Gamma} \right)$$
$$= -k_f \left(\frac{\partial T_f(x, y, z)}{\partial n} \bigg|_{\Gamma} \right), \tag{58}$$

and the longitudinal mean temperature difference in Equation 56 is defined as:

$$\Delta \overline{T}(x) = \overline{T}_{s,\Gamma}(x) - \overline{T}_f(x), \qquad (59)$$

$$\overline{T}_{s,\Gamma}(x) = \frac{\sum_{\Gamma} T_{s,\Gamma}(i,j,k)}{N_{\Gamma}},$$
(60)

$$\overline{T}_f(x) = \frac{\sum \sum \{\rho_f.u.c_p.T_f.\Delta y \Delta z\}_{i=\text{constant},j,k}}{\dot{m}.c_p}, \quad (61)$$

where N_{Γ} is the total number of nodes along the perimeter of the inner wall (here, $N_{\Gamma} = 2 \times 16 + 2 \times$ 16). The averaged longitudinal inner wall temperature, $\overline{T}_{s,\Gamma}$, and the averaged local heat flux, $\overline{q}_{s,\Gamma}$, are mathematically averaged along the perimeter of the inner wall, and the longitudinal bulk liquid temperature, $\overline{T}f$, is averaged according to energy conservation. With Equations 56 to 59, the longitudinal heat transfer coefficient variation and the longitudinal Nusselt number variation for these three cases can be determined, as shown in Figures 24 and 25, respectively.

From these two figures, it can be concluded that the variation of the heat transfer coefficient and the Nusselt number along the flow direction is quite small for this type of microchannel heat sink after thermal entrance lengths. Comparisons between the average Nusselt number for the different heat flux and same Reynolds number are shown in Figure 26.



Figure 24. Longitudinal heat transfer coefficient variation for the three different cases ($\Delta p = 50$, 15 and 6 kPa).



Figure 25. Longitudinal Nusselt number variation for the three different cases ($\Delta p = 50$, 15 and 6 kPa).



Figure 26. Comparisons between the average Nusselt numbers.

These two trends are identical; this is because the Nusselt number for laminar flow is determined solely by channel geometry and local flow conditions.

Figure 27 shows the fluid bulk temperature and the average temperatures of the top, bottom and side channel walls as the functions of the longitudinal distance x, for $\Delta p = 50$ kPa. The fluid bulk temperature increases the quasi-linearity along the x-direction and almost reaches the wall temperature at the exit of the microchannel. Overall, the average temperatures of side walls are slightly higher than the top walls and lower than the bottom wall because the convective resistance is much smaller for the close space between the side walls. While, in Figure 28, i.e. $\Delta p = 6$ kPa, due to low liquid velocity and low convective heat transfer, the temperature difference will be increased between the solid and the liquid, especially at the inlet region of the channel.



Figure 27. Longitudinal mean temperature variations at the top, bottom and side walls and in bulk liquid for $\Delta p = 50$ kPa.



Figure 28. Longitudinal mean temperature variations at the top, bottom and side walls and in bulk liquid for $\Delta p = 6$ kPa.

Large temperature gradients near the inlet region are mainly to induce significant thermal stresses and, therefore, must be carefully considered in practical sink design in order to avoid mechanical failure. The quasilinear trend is not validated for low Reynolds numbers, as shown in Figure 28. In this figure a large portion of heat is conducted into the front part of the heat sink due to the low liquid velocity, and so the temperature gradient between the top and bottom walls is much smaller, approaching a zero value.

As the flow develops and the boundary layers grow in the longitudinal direction, the average heat transfer coefficients (Figure 29) gradually decrease in magnitude. The heat transfer coefficients are expected and, indeed, are larger at the side walls than at the top and bottom walls.



Figure 29. Average heat transfer coefficient distributions inside the channel.

Local Heat Flux Distributions

Figures 30 to 32 illustrate the heat flux distribution along the channel walls for $\Delta p = 50$ kPa, Re = 162.68, $T_{\text{reference}} = 32^{\circ}$ C, $u_m = 1.44$ m/s. For all the channel walls, higher heat fluxes are encountered near the channel inlet. This is attributed to the thin thermal boundary layer in the developing region. The heat fluxes vary around the channel periphery, approaching zero at the corners where the flow is weak for a rectangular channel. Figure 32 shows that the heat flux along the channel side walls is higher than along the channel top and bottom walls (almost two orders of magnitude larger than those at the top and bottom walls) due to the short distance between the channel side walls and the large velocity gradient present.



Figure 30. Numerical predictions of local heat flux distribution for the channel bottom wall.



Figure 31. Numerical predictions of local heat flux distribution for the channel top wall.



Figure 32. Numerical predictions of local heat flux distribution for the channel side walls.

The local heat fluxes at both the bottom and top walls (Figures 30 and 31, respectively) show significant variation in the transverse z-direction, unlike the fluxes at the side walls (Figure 32) which are nearly uniform everywhere but at the inlet and corner regions.

Convergence Performance for the Pressure Correction Equations

For the microchannel heat sink model, illustrated in Figure 1, the total number of control volumes in the heat sink and inside the channel were set to $N_x \times$

 $N_y \times N_z = 30 \times 82 \times 30$ and $N_x \times N_y \times N_z = 30 \times 16 \times 16$ in the three spatial directions, respectively. The multigrid scheme, as discussed in the previous section, was implemented in the two cross-streamwise directions, which consisted of four grid levels from the finest grid $(30 \times 16 \times 16)$ to the coarsest grid $(30 \times 2 \times 2)$. The multigrid behavior followed the typical V-cycle pattern. The entire residual convergence history, with the AC-MG algorithm, is depicted in Figure 33.

The results are compared with the solution procedure that used only the TDMA solver without a multigrid correction. The TDMA solver with the AC-MG algorithm was capable of efficiently driving the residual down to the level of the computer machine round-off error within 400 AC-MG cycles. The residual was driven down by about twelve orders of magnitude.

CONCLUSION

A three-dimensional mathematical model, developed by using incompressible laminar Navier-Stokes equations of motion, is capable of correctly predicting the flow and conjugate heat transfer in a microchannel heat sink. The microchannel heat sink model consists of a 10 mm long silicon substrate, with rectangular microchannels, 57 μ m wide and 180 μ m deep, fabricated along the entire length. A finite volume numerical code with a multigrid technique, based on an additive correction multigrid (AC-MG) scheme, which is a highperformance solver, was developed to solve the steady incompressible laminar Navier-Stokes (N-S) equations over a colocated Cartesian grid arrangement. Higher Reynolds numbers are beneficial at reducing both the water outlet temperature and the temperatures within the heat sink, though at the expense of greater pressure drop. By the magnitude of the mean velocities and the Reynolds numbers obtained from the analytical and numerical methods, the agreement between the



Figure 33. Residual convergence history of the pressure Poisson equation.

two methods is quite satisfactory, providing sufficient evidence for validation of the numerical method. The variations of the heat transfer coefficient and the Nusselt number along the flow direction are quite small for this type of microchannel heat sink after the thermal entrance lengths. The heat flux along the channel side walls is higher than along the channel top and bottom walls (almost two orders of magnitude larger than those at the top and bottom walls) due to the short distance between the channel side walls and the large velocity gradient present. The temperature is highest at the channel corner; this is due to the low velocity of the flow and the resulting high concentration of heat flux. The results indicate that the thermophysical properties of the liquid can significantly influence both flow and heat transfer in the microchannel heat sink. The bulk liquid temperature is shown to vary in a quasi-linear form along the flow direction for high fluid flow rates, but not for low flow rates (low Reynolds number).

NOMENCLATURE

A	surface area of a cell face; area of
	microchannel cross-section
A_S	area of substrate bottom wall
a	coefficient in momentum and pressure
	equations width of channel divided by
7	two III Equation 59
D	height of channel divided by two in Equation 39
c_1	pressure gradient parameter
c_P	specific heat at constant pressure
D_h	hydraulic diameter
F	flux
f	friction factor
g_c	proportionality constant; $g_c = 1$ in SI
	units
H	height of channel
h	convective heat transfer coefficient
Kn	Knudsen number
k	$\operatorname{conductivity}$
L	${\rm length} \ {\rm scale} \ (D_h)$
L_x	length of channel
m	exponent in Equation 41
\dot{m}	mass flow rate
N	number of control volume
Nu	Nusselt number
n	exponent in Equation 41
p	pressure
q_S	heat flux
R	total residual

Re	Reynolds number
r	residual
S	source term
Т	temperature
T_S	temperature at wall of microchannel
t	time
u, v, w	velocity components in x -, y - and z - directions, respectively
V	volume of cell
W	width of channel

Greek Symbols

ρ	density
μ	dynamic viscosity
Γ	periphery of the inner wall of channel
α	a general diffusion coefficient
λ	mean free path
$lpha^*$	aspect ratio
ε	convergence error
Φ	a general variable

 $\Omega \qquad \qquad {\rm volume} \qquad \qquad$

Superscripts

c coarse

f fin n current it

n current iteration \wedge denote to current value

Subscripts

ave	averaged value
b	bottom cell
con	conservation of energy
e	entrance effects; eastern cell
h	hydrodynamic equivalent value
in	at inlet of channel
i,j,k	spatial indices
f	fluid
m	mean value
max	maximum
n	northern cell
nb	neighbor
out	at outlet of channel
p	present cell
\$	surface of channel; solid; substrate wall; southern cell
t	top cell
w	western cell; wall
x, y, z	Cartesian coordinates

Abbreviations

AC-MG	Additive Correction Multigrid
CPU	Central Processing Unit
FOU	First-Order Upwind
N-S	Navier-Stokes
SIMPLE	Semi-Implicit Method for Pressure- Linked Equations
TDMA	Tri-Diagonal Matrix Algorithm

REFERENCES

- Tuckerman, D.B. and Pease, R.F. "High-performance heat sinking for VLSI", *IEEE Electronic Devices Let*ters EDL-2, pp. 126-129 (1981).
- Jang, S.P. et al. "Experimental investigation of thermal characteristics for a microchannel heat sink subject to an impinging jet, using a micro thermal sensor array", Sens. Actuators, A(105), pp. 211-224 (2003).
- 3. Chen, Y. et al. "Experimental investigation of fluid flow and heat transfer in microchannels", *Tamkang Journal of Science and Engineering*, 7(1), pp. 11-16 (2004).
- Wu, H.Y. and Cheng, P. "An experimental study of convective heat transfer in silicon microchannels with different surface conditions", *Int. J. Heat Mass Transfer*, 46(14), pp. 2547-2556 (2003).
- Qu, W. and Mudawar, I. "Analysis of threedimensional heat transfer in microchannel heat sinks", Int. J. Heat Mass Transfer, 45, pp. 3973-3985 (2002).
- Fedorov, A.G. and Viskanta, R. "Three-dimensional conjugate heat transfer in the microchannel heat sink for electronic packaging", *Int. J. Heat Mass Transfer*, 43(3), pp. 399-415 (2000).
- Fedorov, A. and Viskanta, R. "A numerical simulation of conjugate heat transfer in an electronic package formed by embedded discrete heat sources in contact with a porous heat sink", ASME Journal of Electronic Packaging, 119, pp. 8-16 (1997).
- Weisberg, A. et al. "Analysis of microchannels for integrated cooling", Int. J. Heat Mass Transfer, 35, pp. 2465-2474 (1992).
- Knight, R.W. et al. "Optimal thermal design of forced convection heat sinks-analytical", ASME J. Electron Packaging, 113, pp. 313-321 (1991).
- Keyes, R.W. "Heat transfer in forced convection through fins", *IEEE Trans. Electron Dev. ED-31*, pp. 1218-1221 (1984).
- Bejan, A. and Morega, A.M. "Optimal arrays of pin fins and plate fins in laminar forced convection", *ASME J. Heat Transfer*, **115**, pp. 75-81 (1993).
- Lee, D.Y. and Vafai, K. "Comparative analysis of jet impingement and microchannel cooling for high heat flux applications", *Int. J. Heat Mass Transfer*, 42, pp. 1555-1568 (1999).

330

- Peng, X.F. and Peterson, G.P. "Convective heat transfer and flow friction for water flow in microchannel structures", *Int. J. Heat Mass Transfer*, **39**(12), pp. 2599-2608 (1996).
- Kawano, K. et al. "Development of micro channels heat exchanging", Application of Heat Transfer in Equipment, Systems, and Education, R.A. Nelson et al., Eds., HTD-Vol. 361-3/PID-Vol. 3, ASME, New York, pp. 173-180 (1998).
- Wu, H.Y. and Cheng, P. "Friction factors in smooth trapezoidal silicon microchannels with different aspect ratios", *Int. J. Heat Mass Transfer*, 46(14), pp. 2519-2525 (2003).
- Cheng, P. et al. "Forced convection in the entrance region on a packed channel with asymmetric heating", *ASME Journal of Heat Transfer*, **110**, pp. 946-954 (1988).
- Koh, J.C.Y. and Colony, R. "Heat transfer of micro structures for integrated circuits", *Int. Commun. Heat Mass Transfer*, 13, pp. 89-98 (1986).
- Tien, C.L. and Kuo, S.M. "Analysis of forced convection in microstructures for electronic systems cooling", *Proc. Int. Symp. Cooling Technology for Electronic Equipment*, Honolulu, HI, August 31-September 2, pp. 217-226 (1987).
- Webb, R.L. and Zhang, M. "Heat transfer and friction in small diameter channels", *Microscale Thermophysi*cal Engineering, 2, pp. 189-202 (1998).
- Farnam, D. et al. "Development of a complete transient microchannel heat sink model", *Thermal and Thermomechanical Phenomena in Electronic Systems*, ITHERM 2008. 11th Intersociety Conference on Digital Object Identifier, 28-31 May, pp. 113-120 (2008).
- Kim, S.J. "Methods for thermal optimization of microchannel heat sinks", Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology, Taejon, 25(1), pp. 37-47 (Jan. 2004).

- Cheng, Y.J. "Numerical simulation of stacked microchannel heat sink with mixing-enhanced passive structure", Department of Refrigeration and Air-Conditioning, Chin Min Institute of Technology, Miao-Li County, TAIWAN, International communications in Heat and Mass Transfer, **34**(3), pp. 295-303 (March 2007).
- Wang, Y. et al. "Experimental investigation of heat transfer performance for a novel microchannel heat sink", Institute of Micro and Nano Science and Technology, Shanghai Jiaotong University, 3 (March 2008).
- Wong, W.H. and Ghazali, M. "Numerical simulation of a microchannel for microelectronic cooling", *Journal Technology A*, 46(A), pp. 1-16 (2007).
- 25. Eckert, E. and Drake, R., Analysis of Heat and Mass Transfer, McGraw-Hill, New York (1972).
- Ferziger, J. and Peric, M., Computational Methods for Fluid Dynamics, Berlin, Springer-Verlag, ISBN 3-540-59434-5 (1996).
- Rhie, C.M. and Chow, W.L. "Numerical study of the turbulent flow past an airfoil with trailing edge separation", AIAA Journal, 21, pp. 1525-1532 (Nov. 1983).
- Hutchinson, B.R. and Raithby, G.D. "A multigrid method based on the additive correction strategy", *Numer. Heat Transfer*, 9, pp. 511-537 (1986).
- Shah, R.K. and London, A.L., Laminar Flow Forced Convection in Ducts, Academic Press, NY, pp. 78-283 (1978).
- Bejan, A., Convection Heat Transfer, 1st Ed., John Wiley & Sons, New York (1984).
- Kakac, S. and Yener, Y., Convective Heat Transfer, 2nd Ed., CRC Press, Begell House, Boca Raton (1995).
- 32. Mills, A.F., *Heat Transfer*, 2nd Ed., Prentice Hall, Upper Saddle River (1999).