Numerical Modeling of Single-Phase Fluid-Flow in Wavy Micro-Channels

R. Durantes[1], Justin Moon[1], J. Rafael Pacheco[2], Arturo Pacheco-Vega[1,*]
[1]Department of Mechanical Engineering, California State University, Los Angeles, Los Angeles, CA 90032, USA
[2]SAP America Inc. Tempe, AZ 85281, USA
[*] Corresponding author. E-mail: apacheco@calstatela.edu

Abstract: High power- density electronic devices had achieved levels of heat generation that have exceed the current heat removal capacities. Due to increase in power density of modern electronic chips, there is a need of a higher heat flux removal capabilities. In this study, we present numerical simulations of the convective heat transfer on wavy micro-channels to investigate heat transfer enhancement in these systems, by proposing a methodology based on local and global energy balances in the device instead of the commonly used Nusselt number. The analysis is performed on a single-wave baseline micro-channel model that is exposed to a 47 W/cm^2 heat influx at the bottom. The governing equations for an incompressible laminar flow and conjugate heat transfer are first built, and then solved, for sample models by the finite element technique, under steady state conditions, with COMSOL Multiphysics. The simulation was performed for five values of wave amplitude A = (0, 50, 100, 150, 200) µm, three values of Reynolds number *Re* = (50, 100, 150), and a wavelength of 2 mm. In order to extract the data, we couple MATLAB and COMSOL via Livelink, illustrating that our numerical solutions are in good agreement with published data on single-wave channel. The simulation results indicate that wave amplitude is not an important factor, but the Reynolds number *Re*, plays a key role in the heat transfer enhancement of the device and in both the fluid and solid block temperature that are achieved.

Keywords: micro-channels, wavy geometry, heat transfer enhancement, conjugate heat transfer

Introduction

Electronic components and devices produce varying levels of heat during their operation. In cases where significant amounts of heat are generated, intervention in the form of thermal management may be required in order to prolong working life and increase reliability. Rapid miniaturization and increased power density of modern electronic devices; e.g., electronic chips, demand more compact and reliable cooling solutions to avoid shortening their life span. Although recent developments in high-heat flux heat removal techniques have been proposed to solve this problem, several years ago Tuckerman and Pease [1] suggested an approach based on micro-channels. Since then, experimental and numerical analyses of micro-channel devices using single phase fluids has been conducted with the idea of improving their performance [2,3].

Micro-channel heat exchangers are devices designed to exchange thermal energy between a solid block and a working fluid. There are methods to augment passive heat transfer with the use of fins, ribs, and other additional components that serve to increase the effective heat transfer area between solid and fluid. A tenable option for internal flows, for example, is the shift to using non-circular ducts, thus increasing the hydraulic diameter. Different micro-channel designs may offer the possibility to enhance the energy transfer in electronic equipment. A particular example is that of a wavy geometry micro-channel design. A single-phase fluid-flow wavy micro-channel offers an avenue to improve the thermal management in high power-density electronic devices. Several studies have confirmed that wavy micro-channels, designed to enhance heat transfer by improving fluid mixture and increasing surface areas, often increase the transfer of energy at the expense of slightly larger pressure drop along the channel [4,5]. In recent work, Moon et al. [6] have shown that adding harmonic surfaces to the basic wavy topology enhances heat transfer of the device. In addition, by considering a variety of conductive materials for the solid block enclosing the channel, Moon et al. [7] found that selection of specific material for the solid block highly-influences the diffusion of heat, but it is negligible on the Nusselt number for the fluid.

This study presents numerical simulations of the convective heat transfer on a single-phase fluid-flow wavy micro-channel to investigate heat transfer enhancement in these systems. These are coupled to a methodology based on local and global energy balances in the device [8] and use the heat transfer rate instead of the commonly used Nusselt number for our calculations. To this end, the governing equations are first formulated and then solved on a representative computational domain by the finite element method. From velocity, pressure, and the temperature fields; the heat rates are then computed from local and global energy balances and pressure drop – from inlet to outlet fluid – are used to evaluate the relative system performance.

Micro-channel Model and Governing Equations

In this study, the model of the device consists of a constant square cross-section channel that is enclosed by a copper block, shown in Figure 1. The micro-channel has a size of 0.5 mm by 0.5 mm by 20 mm in length, with a region of 16 mm with sinusoidal surfaces and two 2 mm inlet and outlet- straight sections. The shape of the copper block that encloses the channel is a square duct of thickness 1.5 mm and length 20 mm. Water flows inside the channel, and a 47 W/cm^2 is applied at the bottom surface of the copper solid block. Energy is advected out of the system through the internal flow of water in the channel. The micro wavy channel geometry is implemented with sinusoidal functions on the top and bottom surfaces, while the side-walls remain planar. The sinusoidal profiles are based on the following function,

$$y(x) = A \cdot \sin\left(\frac{2\pi x}{\lambda}\right), \tag{1}$$

where x and y are the streamwise and vertical coordinate directions, respectively, A is the wave amplitude and λ is corresponding wavelength.

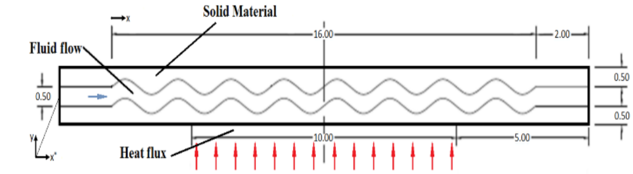

Figure 1. Schematic of a wavy micro-channel (dimensions in mm).

The three-dimensional conjugate model for a solid copper block Ω_s, surrounding the channel inside of which water flows Ω_f is based on the following assumptions: For Ω_f, we have incompressible flow of a Newtonian fluid, with constant properties, in the laminar regime, under steady state conditions, without: body forces, viscous dissipation and radiative heat transfer, along with homogeneous material properties in the solid. The governing equations are given by

$$\nabla \cdot \mathbf{u} = 0 \tag{2}$$

$$\rho_f \, (\mathbf{u} \cdot \nabla) \, \mathbf{u} = -\nabla p + \mu_f \nabla^2 \, \mathbf{u} \tag{3}$$

$$\rho_f \, c_{p,f} \, (\mathbf{u} \cdot \nabla) T = k_f \nabla^2 \, T. \tag{4}$$

In Eqs. (2)-(4), $\mathbf{u}$ is the Cartersian velocity vector; with u, v and w, being its components in the x, y, and z directions, respectively; p is the fluid pressure, ρ_f is the fluid density, and $c_{p,f}$ is the specific heat, k_f is the fluid thermal conductivity, μ_f is the dynamic viscosity and T its temperature. For Ω_s, a homogeneous solid material, the energy equation is given by

$$k_s \nabla^2 \, T_s = 0, \tag{5}$$

where k_s is the copper substrate thermal conductivity and T_s is its corresponding temperature. Table 1 and 2 show the values of the properties for the fluid (water) and solid (copper). All the properties here are taken at a reference temperature of 300 K.

Table 1. Fluid properties used in the simulations.

Parameter	*Description*	*Value*
$c_{p,f}$	Specific heat	4.182 $[\mathrm{kJ/kg \cdot K}]$
k_f	Thermal conductivity	0.61 $[\mathrm{W/m \cdot K}]$
μ	Dynamic viscosity	1.007×10^{-3} $[\mathrm{Pa \cdot s}]$
ρ_f	Fluid density	908.4 $[\mathrm{kg/m^3}]$

Table 2. Solid properties used in the simulations.

Parameter	*Description*	*Value*
$c_{p,s}$	Specific heat	0.375 $[\mathrm{kJ/kg \cdot K}]$
k_s	Thermal conductivity	403.7 $[\mathrm{W/m \cdot K}]$
ρ_s	Solid density	8933 $[\mathrm{kg/m^3}]$

The boundary conditions are as follows. For the channel domain Ω_f, we prescribe a constant streamwise velocity $u = u_{in}$ at the channel inlet; zero-pressure and zero-viscous stresses at the outlet; no-slip and no-penetration conditions at the channel walls. In addition, uniform temperature $T_{in} = 300$ K at the inlet; conditions of continuity of both temperature; i.e., $T = T_s$, and heat flux; i.e., $k_f \partial T / \partial n = k_s \partial T_s / \partial n$, at the solid-fluid interface-walls; and conditions of zero temperature gradients; i.e., $\partial T / \partial n = 0$ are at the outlet. For the solid block domain, Ω_s, the heat-generating chip is modeled as uniform heat influx of $q'' = 47$ W/cm^2, at the bottom surface (on a 10 mm × 1.5 mm area); all other surfaces of the solid are considered adiabatic. Finally, symmetric conditions are prescribed at the mid-plane z = 0 (see Figure 2), for the entire system in order to reduce CPU time.

Numerical Model and Simulation Methodology

Modeling and simulation for the current problem is performed via COMSOL Multiphysics (http://www.comsol.com). The three-dimensional representation of the wavy micro-channel design is generated within the COMSOL software using the geometry and functionality tools. The wavy shapes in the channel were built with sinusoidal functions in the top and bottom surfaces, the sidewalls remaining planar. At the bottom of the channel-block device, a heat influx boundary condition is used to represent the power transferred from the chip to the cooling device. After the micro-channel geometry is completely defined, a condition of symmetry is imposed longitudinally (along *x*-direction) at the middle *y*-*z* plane of the entire device. The symmetric condition decreases computational time by removing non-essential meshed sub-domains, simplifying the analysis without sacrificing accuracy. Figure 2 shows a typical mesh of the channel and the solid-block domains.

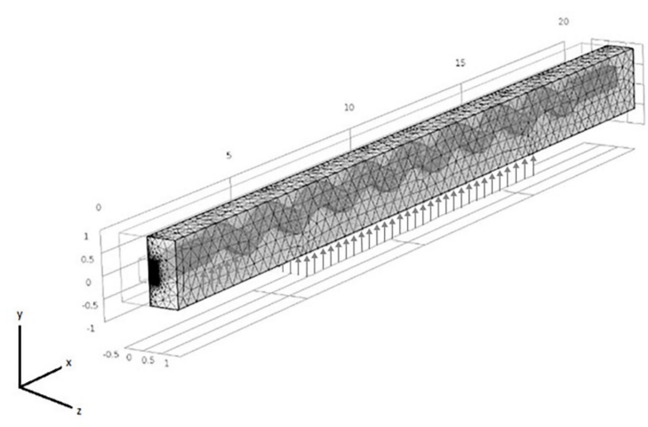

Figure 2. Typical mesh of the channel and the solid block domains.

The numerical simulations are carried out by using the Computational Fluid Dynamics and Heat Transfer modules, along with Livelink for MATLAB. The specific physics interfaces used are Heat transfer and Laminar Flow, both of which are coupled to the Multiphysics of Non-Isothermal Flow. Equations (2)-(5), for an incompressible laminar flow and conjugate heat transfer, were first discretized on the channel and solid-block domains and then solved via the finite element method (FEM). The FEM technique discretizes the domain into finite elements and the dependent variables are solved at nodes connecting these elements. This is done by multiplying the governing equations by weighting functions, and integrating them to obtain their weak form. The solution is then approximated at the element level, and by assembling the elemental equations, the entire problem can now be solved. The main advantage of the finite element method is its treatment of the boundary conditions on curved surfaces.

For each geometry under analysis, for fluid temperature, velocity and pressure, and the temperatures within the solid, we use three-dimensional unstructured meshes with four-node tetrahedral elements. In the regions close to the walls, where the boundary layers develop, we apply a mesh comprised of hexahedral elements enabling a sharp representation of the fluid-solid interface. The resulting system of algebraic equations is computed iteratively using the generalized minimum residual (GMRES) solver. The relative tolerance was set at 10^{-6}. To accurately setup the thermal condition for the fluid (T_{in}) and the solid ($\partial T/\partial n = 0$) domains, at the inlet cross-section of the system, and to sharply distinguish between them at the fluid-solid interface, we applied a discontinuous Galerkin condition [9].

Typically, simulations were carried out for five values of wave amplitude $A =$ (0, 50, 100, 150, 200) μm, three values of Reynolds number, $Re =$ (50, 100, 150}), which is defined as $Re = D_h u_{in} \rho_f / \mu_f$, and a wavelength of $\lambda = 2$ mm. In this work, we successfully developed the models in COMSOL and used Livelink for the runs and further post-processing. For our purposes, COMSOL Livelink with MATLAB allows us to connect COMSOL to MATLAB and use the Java syntax to develop models, use functions created in MATLAB within the COMSOL model, use programming logic in MATLAB such as for- and while-loops, and post-process data with ease [10]. MATLAB scripts were coded to extract temperature, velocity, and pressure values, as well as temperatures within the solid, along 321 cut planes in the $y - z$ direction (or normal to the streamwise direction) along the channel from our COMSOL models. At each of the cut planes, the average values were calculated using the function mphmean. It is to note that each model was integrated in loops within MATLAB, which is a very powerful tool. However, running a large number of models in a loop fashion requires increasingly larger amount of memory, due to COMSOL storing the entire modeling history in the model files. This was remedied by loading an interim model at the beginning of each iteration via the function mphload, thus removing the model history, and then saving the model at the end of each iteration separately.

Grid independence tests, from the numerical results, were done with various grid sizes and different values of inlet velocity. From this process, a mesh with 3.6 million elements is sufficient to achieve an accuracy within 0.5% of that obtained by the maximum number of elements (4.8 million). In the case of the coper solid-block material only 282,000 elements are required for the solution to become independent from the grid size. A total number of 3.6 million elements are used in the computational domain comprising both solid and fluid regions. A typical set of convergence tests, for a geometry with *Re*=150 and $A = 200$ μm, is illustrated in Figure 3 for the fluid pressure p, streamwise velocity u, and temperature T, at a fixed point in the domain. It is important to mention that similar numbers were obtained for all the geometries considered on this work. For each channel, typical CPU times are about 11 hours per run.

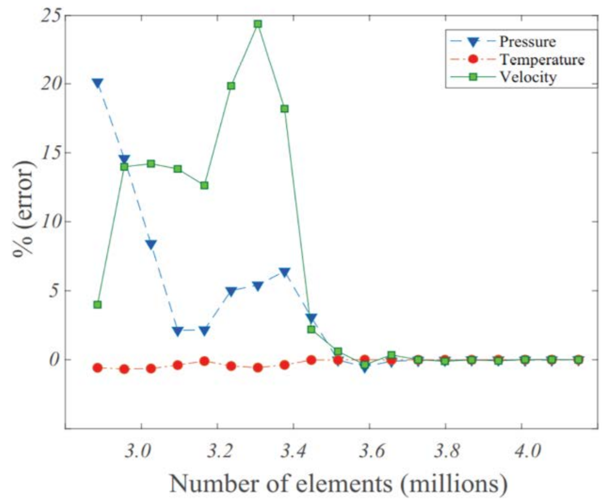

Figure 3. Grid independence tests.

Simulation Results and Discussion

Velocity, Temperature and Pressure Fields

As indicated in the previous section, the simulations were configured to analyze heat enhancement for five values of wave amplitude $A =$ (0, 50, 100, 150, 200) μm, three values of Reynolds number, $Re =$(50, 100, 150), a wavelength of 2 mm and a heat influx $q'' = 47$ W/cm^2. Figure 4 illustrates a typical set of numerical solutions of the steady-state conjugate heat transfer model as streamlines, temperature contours and isotherms, respectively, all computed at the mid-plane $z = 0$, for the entire device, for $A = 150$ μm and $Re = 100$ (results for other values of the parameters show similar behavior).

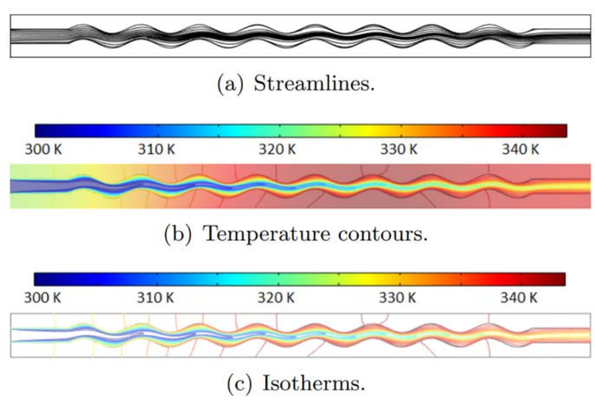

Figure 4. Streamlines, temperatures contours and isotherms for a wavy micro-channel for A= 150 µm and *Re*= 100.

From Figure 4(a), it can be seen that the flow patterns along the micro-channel evolve from uniform flow at the inlet into periodic patterns with streamlines being closer to each other near the channel centerline, and into a new-uniform flow near its outlet. On the other hand, Figures 4(b) and 4(c) show that the temperature field transitions from a uniform profile and develops – but not in a periodic fashion – as the fluid travels along the channel (with larger temperature changes occurring upstream of and at the crests of the wavy surfaces), advecting the energy supplied toward the outlet. In addition, the copper-block temperature contours are spanwise homogeneous due to its high thermal conductivity. However, they increase along the x-direction, due to the advective process from the cooling water, from a value close to that of the fluid at the inlet, to a maximum in the region where the heat influx occurs, with a subsequent decrease towards the fluid outlet.

The results from our numerical solutions on single-wave channel device are in good agreement with published data by Gong et al. [4] as illustrated in Figure 5 for the pressure drop Δp, as a function of both A and Re. The figure shows that our solutions not only qualitatively follow the same trend as those of Gong et al. [4], but quantitatively they are also very close; the maximum absolute value of the percentage difference is less than 5% demonstrating the accuracy of our simulations.

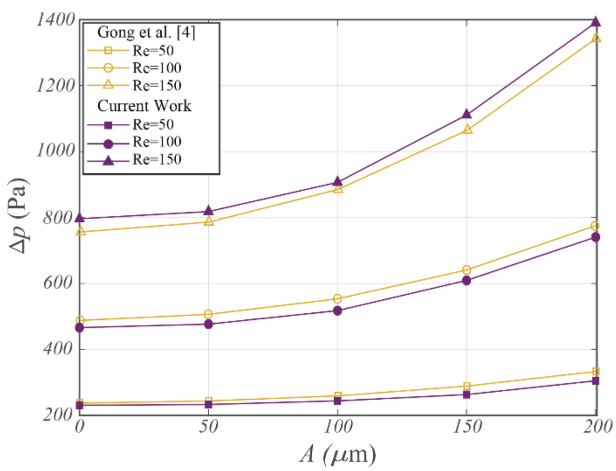

Figure 5. Comparison of present results for Δp vs. A, to ref. [4].

Cross-sectional Averaged Equations and Heat Transfer

In this work, the analysis of the thermal performance of the micro-channel is not based on the typical Nusselt number. Instead, we couple the numerical solutions of the convective heat transfer to a methodology based on local and global energy balances in the device, and use the heat transfer rate for the analysis. This forms the basis for the calculations to investigate heat transfer enhancement in these systems. The method presented here has been recently proposed by Durantes et al. [11], following the works of Cobian-Iniguez et al. [8], and Motamedi et al. [12]. Thus, the cross-sectional-averaged pressure $\bar{\boldsymbol{p}}$, streamwise flow velocity $\bar{\boldsymbol{u}}$, and the fluid temperature $\bar{\boldsymbol{T}}$, derived from the local values of the $\boldsymbol{p}$- , **u**-, and $\boldsymbol{T}$-fields, are

$$\bar{p} = \frac{1}{A}\int_A p \; dA \tag{5.1}$$

$$\bar{u} = \frac{1}{A}\int_A (\mathbf{u}\cdot\mathbf{n}) \; dA \tag{5.2}$$

$$\bar{T} = \frac{\int_A (\mathbf{u}T\cdot\mathbf{n})\, dA}{\int_A (\mathbf{u}\cdot\mathbf{n}) \; dA} \tag{5.3}$$

where A is the cross-sectional area normal to the unit vector $\mathbf{n}$, associated with the surface of interest $dA = dydz$, at any point in the streamwise direction x. The wavy-section of the channel is partitioned into $N = 8$ subsections, each of length ΔL. Upon using the averaged values $\bar{u}$, and $\bar{T}$, an energy balance at each subsection provides the net heat rate Q, advected by the fluid as

$$Q_i = \rho f \bar{u} A c_p[\bar{T}(x_i) - \bar{T}(x_{i-1})], \tag{6}$$

where, $x_{i-1} = x([i-1]\Delta L)$ and $x_i = x(i\Delta L)$, for $i = 1, 2, \ldots, N = 8$, represent the location of the cross-sectional plane of the i-th subsection and Q_i is the corresponding heat transfer rate. The total heat transferred by the device, Q_T, is given by

$$Q_T = \left(\sum_{i=1}^{N=8} Q_i\right) + Q_{L_{x-in}} + Q_{L_{x-out}}$$

$$Q_T = \rho_f \bar{u} A c_{p,f}[\bar{T}_{out} - \bar{T}_{in}] \tag{7}$$

where $\bar{T}_{in}$ and $\bar{T}_{out}$, are the terminal temperatures of the fluid in the micro-channel device, and $Q_{L_{x-in}}$ and $Q_{L_{x-out}}$, are the heat transfer contributions at the entrance and exit straight-sections.

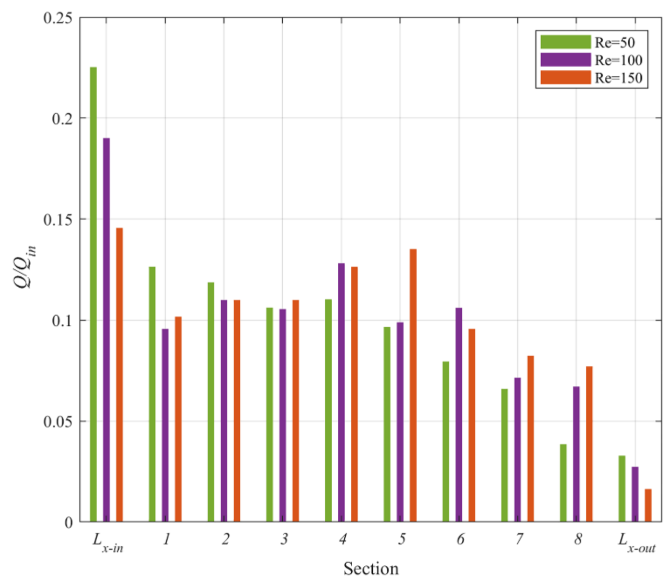

Figure 6. Fraction of influx heat rate transferred at each section, for A = 150 µm.

The corresponding results from the methodology, as a fraction of the inlet heat rate Q/Q_{in} (with $Q_{in} = q'' A_{base} = 7.05$ J), advected by the fluid at each section of the micro-channel are shown in Figure 6. The solutions shown in the figure for a sample amplitude A = 150 µm and Re = (50, 100, 150), at each section, indicate that Q/Q_{in} varies both along the streamwise direction, and with Re number. It is to note that, for all the devices analyzed, as shown in the figure, the largest values occur at the inlet section (with contributions of about 20%), followed by those in the middle sections with contributions of about 10-13%. The smallest values occur in the sections downstream along the channel (with contributions of less than 8.5%), although Q_{in}, is applied in the regions corresponding to sections 2 to 7. The Q/Q_{in}-distribution changes with Re number; e.g., a smooth decline in Q/Q_{in} occurs for Re = 50, a decrease-increase decrease transition arises for Re = 100, which is larger for Re = 150.

Conclusions

Heat generation from high power- density electronic devices have exceed the current heat removal capacities. Due to increase in power density of modern electronic chips, there is a need of even higher heat removal capabilities, and micro-channel systems offer a potentially effective solution. In this work, we have performed three-dimensional steady-state numerical simulations of the convective heat transfer on wavy micro-channels to investigate heat transfer enhancement in these systems, along with a methodology based on local and global energy balances in the device instead of the commonly used Nusselt number.

The governing equations were discretized and solved with the specialized software COMSOL Multiphysics. The software is based on the finite element method to solve physics- and equation-based problems. The simulations were implemented by using the CFD and Heat Transfer modules, along with LiveLink for MATLAB. Heat transfer and Laminar Flow physics interfaces were coupled with the Multiphysics of Non-Isothermal Flow. Once our model was successfully developed in COMSOL, MATLAB and COMSOL coupled via LiveLink to extract the desired data for further analysis.

From our numerical solutions of the three-dimensional velocity, pressure, and temperature fields, and our methodology based on local and global energy balances – including the concept of fraction of the total heat input – the results show that it is possible to design more efficient devices. At the expense of some generality, the numerical approach is useful in providing accurate and clear information on the distribution of both the average fluid temperature along the device, and the corresponding fraction of inlet heat transfer rate at specific sections to assess its relative performance. Results demonstrate that the Reynolds number *Re* (i.e., fluid velocity), plays an important role in the heat transfer enhancement of the device and in both the fluid and solid block temperatures that are achieved. In addition, as wave amplitude increases, so does the effectiveness of the device.

References

[1] D. Tuckerman, and R. Pease, "High-performance heat sinking for VLSI", IEEE Electron Device Letters, **2**(5), pp. 126–129, (1981).

[2] P. Lee, S. Garimella, and D. Liu, "Investigation of heat transfer in rectangular microchannels", *International Journal of Heat and Mass Transfer*, **48**(9), pp. 1688-1704, (2005).

[3] W. Qu, and I. Mudawar, "Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink", *International Journal of Heat and Mass Transfer*, **45**, pp. 2549-2565, (2002).

[4] L. Gong, K. Kota, W. Tao, and Y. Joshi, "Parametric numerical study of flow and heat transfer in microchannels with wavy walls" *ASME Journal of Heat Transfer*, **133**(5), Paper No. 051702, (2011).

[5] G. Xie, J. Liu, Y. Liu, B. Sunden and W. Zhang, "Comparative study of thermal performance of longitudinal and transversal-wavy microchannel heat sinks for electronic cooling", *ASME J. Electronic Packaging* **135**(2), Paper No. 021008, (2013).

[6] J. Moon, J.R. Pacheco and A. Pacheco-Vega, "Heat transfer enhancement in wavy micro-channels through multiharmonic surfaces", in *Proceedings of the 2018 ASME International Mechanical Engineering Congress and Exposition*, Pittsburgh, PA, USA, IMECE2018-86425, (2018).

[7] J. Moon, J.R. Pacheco and A. Pacheco-Vega, "Heat transfer enhancement in wavy micro-channels: Effect of block material" in *Proceedings of the 4th World Congress on Momentum, Heat and Mass Transfer* (*MHMT'19)*, Rome, Italy, ENFHT-120, (2019).

[8] J. Cobian-Iniguez, A. Wu, F. Dugast, and A. Pacheco-Vega "Numerically-based parametric analysis of plain fin and tube compact heat exchangers", *Applied Thermal Engineering*, **86**, pp.1–13, (2015).

[9] D. Arnold, F. Brezzi, B. Cockburn, and L. Marini, "Unified analysis of discontinuous Galerkin methods for elliptic problems". *SIAM J. of Numerical Analysis*, **39**, pp. 1749–1779, (2002).

[10] COMSOL Multiphysics, LiveLink for MATLAB User's Guide version 4.3, Chapter 1: Introduction, p. 8, (2012).

[11] R. Durantes, J. Moon, J.R. Pacheco and A. Pacheco-Vega, "Averaged energy-balance analysis of wavy micro-channels", Submitted to: *8th European Thermal Sciences Conference* (2021).

[12] A. Motamedi, A. Pacheco-Vega, and J.R. Pacheco, "Numerical analysis of a multi-row multi-column compact heat exchanger" *Journal of Physics: Conference Series*, **395,** pp. 1–9, Paper No. 012121, (2012).

Acknowledgements

The authors would like to acknowledge support for this work by NSF under award No. HRD-1547723.