Proceedings of the 7th World Congress on Momentum, Heat and Mass Transfer (MHMT'22)
~~Lisbon, Portugal~~ Virtual Conference – April 7 – 9, 2022
Paper No. ENFHT 160
DOI: 10.11159/enfht22.160

Two-Dimensional Approximation of a Three-Dimensional Wavy Microchannel

Roxana Durantes[1], J. Rafael Pacheco[2], Arturo Pacheco-Vega[1]
[1]California State University, Los Angeles
Los Angeles, CA 90032, USA
rdurant@calstatela.edu; apacheco@calstatela.edu
[2]SAP America Inc.
Tempe AZ 85281, USA
rafael.pacheco@sap.com

Abstract - In this work, we present a systematic approach to establishing the validity of a two-dimensional approximation of three-dimensional single-phase thermal convection in wavy microchannels. The system consists of a square cross-section single-wave channel enclosed by a copper block with a constant thickness of 0.5 mm on all the walls, but its width can be modified in accordance with the width set for the channel. Since the overall objective of the study is the heat transfer enhancement via thermal convection process occurring in these systems, the working fluid is water, exposed to a heat influx of 47 W/cm^2 at the bottom. On the other hand, the main goal is to determine when the two-dimensional approximation for different values of width departs from the three-dimensional original model, and at what values of width it is reasonable to make a two-dimensional simplification. To this end, the width of the channel and the copper block is increased systematically in the range of [0.5,20] mm, while keeping all other geometrical parameters constant. The numerical simulations for an incompressible laminar flow and conjugate heat transfer are based on three-dimensional versions. For purposes of comparison, a two-dimensional model is also built and solved. The numerical solutions are performed by the finite element technique, under steady-state conditions, for wave amplitudes A = (0, 50, 200) μm, Reynolds number of Re = 50, and wavelength of 2 mm. The results are shown as velocity and temperature profiles at the mid-plane z = 0, and averaged energy values at different streamwise locations to find the value in width representing the departure point of the two-dimensional approximations from the three-dimensional original model. As the width is increased to a value beyond 20 mm, the velocity and temperature values in the three-dimensional model tend to approach the ones of the two-dimensional model.

***Keywords*:** Wavy micro-channel; Numerical analysis; Conjugate heat transfer; Two-dimensional approximation.

1. Introduction

Over the past decades, electronic devices have become more powerful and substantially smaller, with increasing heat flux densities. For these reasons, innovative solutions in thermal managements are extremely important [1]. One of those solutions was suggested several years ago by Tuckerman and Pease [2], who proposed the use of microchannels as a way to provide thermal management in devices using integrated circuits. Since their seminal paper, different designs of microchannels, e.g., wavy, pin-fin, oblique fin and double-layered, have been analyzed both numerically and experimentally. With feasible pressure drops, these microchannel designs have shown success in dissipating the heat flux. A particular example is the case of microchannels with wavy geometries, which, when compared to straight channels, have achieved better heat transfer performance regardless of the slightly pressure drop along the channel [3–6]. Recent comprehensive numerical studies by Grajeda et al. [7] and Moon et al. [8, 9], have shown that by adding harmonic surfaces to the basic wavy topology a substantial enhancement heat transfer as compared to that of the original device is possible.

Though experiments are essential to understand the physics in a system; e.g., microchannel equipment, many times they can be extremely expensive. Thus, numerical simulations are very useful in analyzing the system before physical models are built and tested with the idea of enabling the experiment results to be interpreted with reliability. The usefulness of numerical models is particularly advantageous not only in the design process of the device, but in optimizing it, i.e., finding the best among all possible designs. However, numerical simulations of mathematical models need to accurately reflect the physics of the problem. In this regard three-dimensional models of any system are the most accurate; they,

however, can be very expensive computationally, in particular when simulating coupled physics, as it is the case of the conjugate heat transfer in microchannels. Therefore, simpler models, e.g., two-dimensional, may also be particularly in the context of performing parameter optimization, since they require less memory and computational time to solve the equations [10]. The main emphasis of this study is to examine the validity of two-dimensional approximations of three-dimensional single-phase thermal convection in wavy microchannels. To this end, we will be utilizing a systematic approach to determine when the two-dimensional approximation for different values of channel width depart from the three-dimensional original model, and at what values of width it is reasonable to make a two-dimensional simplification. The width of the channel-solid system will be increased in the range of $[0.5,\ 20]$ mm. After describing the system and the mathematical model, the model equations are solved to obtain values of velocity, pressure and temperature for the fluid in the microchannel, and temperature in the solid domain. These values are further compared to those of the three-dimensional models to assess the accuracy of the two-dimensional model.

2. System Description

The device corresponds to a three-dimensional microchannel with square cross sectional area that has measurements of 0.5 mm by 0.5 mm by 20 mm for width CW, height H, and length L, respectively; a region of 16 mm with sinusoidal top- and bottom-surfaces located at the middle of the channel and two straight sections of 2 mm length, one for the inlet section and another for the outlet section. The channel is enclosed by a copper block with dimensions of 1.5 mm by 1.5 mm by 20 mm for width, height and length, respectively. A heat flux of 47 W/cm^2 is applied at the bottom of the copper solid block surface. The device corresponds to a heat exchanger device of a circuit-dissipating equipment used in a typical electronic-cooling application. The working fluid in the system is water, which works by internally flowing inside the channel to advect the heat flux out of the system. The device is shown schematically, with all dimensions being in [mm], in Figure 1.

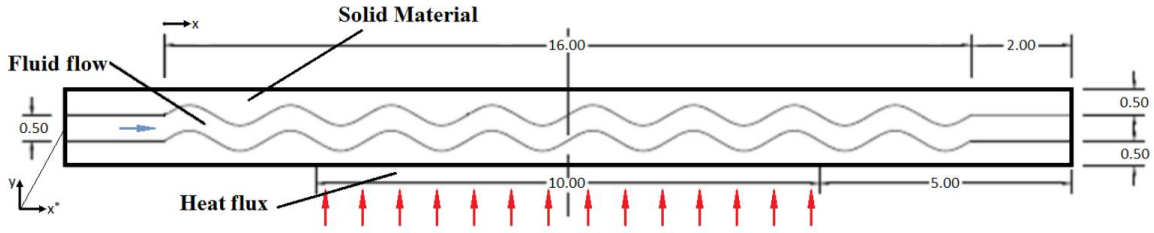


Figure 1: Side view of a wavy microchannel.

3. Mathematical Model

In this study, we focus on two and three-dimensional simulations of wavy microchannels with amplitudes $A = (0, 50, 200)$ μm and a Reynolds number of $Re = 50$. The models are based on the schematic illustrated in Figure 1, here referred to as the *wavy geometry* which, within the sinusoidal region, the top and bottom surfaces of the base-design channel were built using the function

$$y(x) = A \cdot \sin(2\pi x/\lambda) \tag{1}$$

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

The governing equations are for an incompressible flow of a Newtonian fluid, with constant properties, in the laminar regime, under steady state conditions and without radiative heat transfer, body forces and viscous dissipation. For the copper solid, the material properties are assumed to be those of a homogeneous material. The two comprise a conjugate heat transfer problem, which can be written, for the fluid (water) flowing inside the wavy channel Ω_f, as

$$\rho_f \nabla \cdot \mathbf{u} = 0 \quad (2)$$

$$\rho_f (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} \quad (3)$$

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

and for the heat diffusion in the solid (copper substrate), Ω_s, as

$$k_s \nabla^2 T_s = 0 \quad (5)$$

In the above equations, $\mathbf{u}$ is the Cartesian velocity vector; p is the fluid pressure, ρ_f is its density, and $c_{p,f}$ is the specific heat, k_f the fluid thermal conductivity, μ is the dynamic viscosity and T its temperature. In addition, k_s and T_s are the copper substrate thermal conductivity and its corresponding temperature, respectively.

In the case of a three-dimensional model, the velocity vector has components u, v and w, in the x, y, and z-directions, and p, T and T_s depend on these three directions, with a total number of 5 equations to be solved. On the other hand, in a two-dimensional model, the velocity vector reduces to components u and v, in the x- and y-directions, with p, T and T_s depend on these two directions, disregarding the contributions in the z-direction; with a total number of 4 equations to be solved.

The boundary conditions corresponding to the fluid flow simulation consist of a prescribed inlet velocity $u = u_{in}$ along the channel direction, and normal to the inlet cross section, which is coupled with prescribing a zero pressure with no viscous stresses at the outlet. At the channel walls, where the copper solid and the fluid are in contact, no-slip and no-penetration conditions are imposed. For the thermal simulation, there is a prescribed inlet temperature T_{in} of 300 K, and conditions of continuity of both temperature; i.e., $T = T_s = T_w$, and heat flux; i.e., $k_f \partial T/\partial n = k_s \partial T_s/\partial n = k_s \partial T_w/\partial n$, at the solid-fluid interface walls. For the solid domain a uniform heat flux of $q'' = 47\ \mathrm{W/cm^2}$ boundary condition is used to model the heat generating chip; the heat flux is applied at the bottom surface along a 10 mm line in the streamwise direction (× 1.5 mm width in the case of a three-dimensional model). The rest of the walls of the solid, where the heat flux is not applied are considered adiabatic.

4. Discretization and Grid Independence Test

The numerical simulations for this study were carried out using the commercial software, COMSOL Multiphysics (http://www.comsol.com). The governing equations (2)-(5) are discretized on the channel and solid block domains and then solved by the finite element method (FEM), which has the advantage – over other numerical techniques [11–16], in the treatment of the boundary conditions on complex geometries. For each channel-block geometry considered, whether two- or three-dimensional models, unstructured meshes are built inside both the channel domain Ω_f , and the copper block Ω_s. The system of algebraic equations that resulted from the discretization are computed iteratively by using the generalized minimum residual (GMRES) solver, which solves with a relative tolerance of 10^{-6} for the calculations of velocity, pressure and temperature for the fluid domain and temperature in the solid domain. A discontinuous Galerkin condition [17] was applied at the inlet of the channel to accurately setup the thermal condition for the fluid (T_{in}) and the solid ($\partial T_s/\partial n = 0$) domains, and to sharply distinguish between them at the fluid-solid interface. To calculate the temperature values within the solid structure Ω_s, the GMRES solver with a relative tolerance of 10^{-6} is also used.

In order to ensure grid independence of the numerical results for the two- and three-dimensional models, various grids sizes were assessed using different values of the inlet velocity (i.e., Reynolds number). For the three-dimensional model, for example, Figure 2 illustrates a typical set of convergence tests for the pressure p, streamwise velocity u and temperature T, at a fixed point in the fluid domain Ω_f, for the single wavelength configuration, with a wave amplitude of

200 μm and a Reynolds number $Re = 150$ based on the hydraulic diameter ($D_h = 500$ μm). For the process, the number of elements was varied from 3.0 to 4.0 million. From the figure it can be seen that, essentially with 3.5 million elements, all the quantities have reached grid independence with an accuracy within 0.05%. On the other hand, the copper domain requires only 282,000

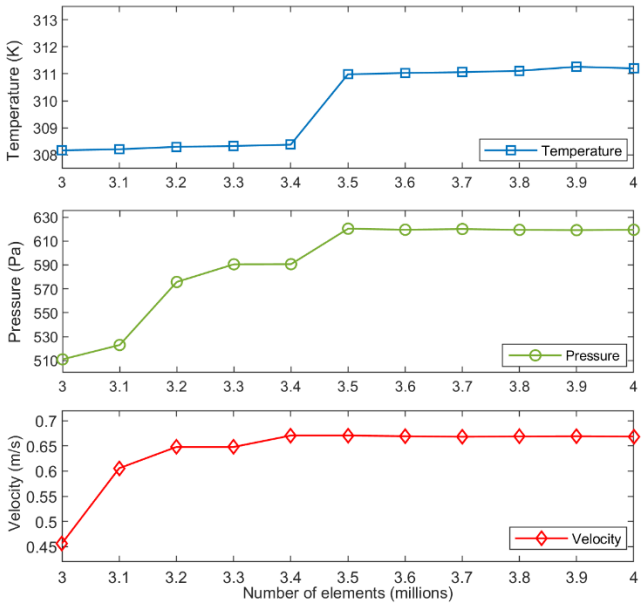


Figure 2: Grid independence tests for 3D model.

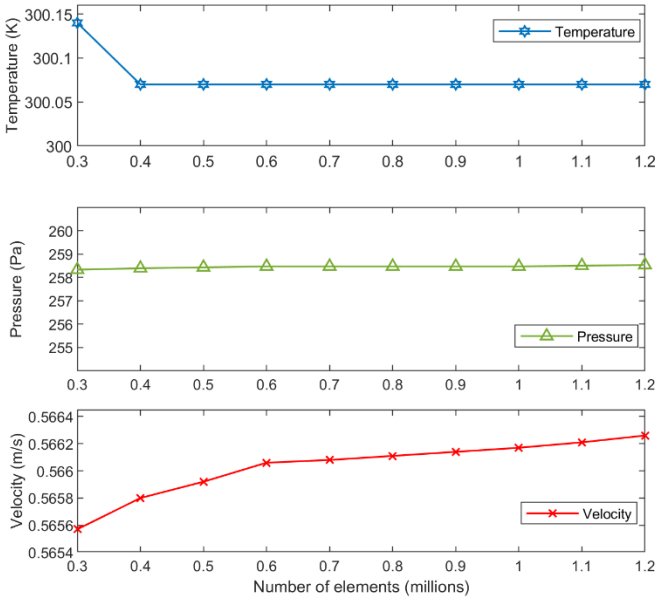


Figure 3: Grid independence tests for 2D model.

elements for the solution to become independent from the grid size. The total number of elements comprising both solid and fluid regions for the three-dimension model is 3.6 million. It is to be noted that as the width of the microchannel is increased, the number of elements also increases. Finally, typical CPU times for solving each model is around 11 hours.

For the two-dimensional model, on the other hand, Figure 3 illustrates the convergence tests for the temperature T, pressure p and streamwise velocity u, at a fixed point in the two-dimensional fluid domain Ω_f, for the single wavelength configuration with a wave amplitude of 200 μm and a Reynolds number $Re = 150$. From the figure, it can be observed that a mesh with 0.3 million elements is sufficient to obtain accurate results for all the quantities. The total number of elements in the computational comprising both the solid and the fluid domain for the two-dimensional model is 0.5 million elements. Typical CPU times for this model is of only 30 minutes.

5. Results and Analysis

In all the simulations, the properties used for both the fluid (water) and solid (copper) are shown in Tables 1 and 2 for a reference temperature of 300 K.

Table1: Fluid properties used in 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	$1.007\times10^{-3}\,[\mathrm{Pa\cdot s}]$
ρ_f	Fluid density	$998\,[\mathrm{kg/m^3}]$

Table 2: Solid properties used in 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	$8933\,[\mathrm{kg/m^3}]$

Since the goal of this study is to establish the point at which the two-dimensional approach is representative of the three-dimensional system, we first focus on the three-dimensional versions of the mathematical model (2)-(5) for the original system, and then systematically increase its width CW. The original model was solved for five values of the wave amplitude $A = (0, 50, 100, 150, 200)\,\mu$m, two values of Reynolds number $Re = \{50, 150\}$, and a wavelength of $\lambda = 2$ mm, and the solutions compared to the results of Gong et al. [6] who reported an analysis of a similar system. As an example, our results for the pressure drop Δp, are shown in Figure 4 as functions of both A and Re. From the figure it can be observed that our numerical solutions not only qualitatively follow the same trend as those of [6], but quantitatively they are also very close, as the maximum absolute value of percentage error is less than 5%.

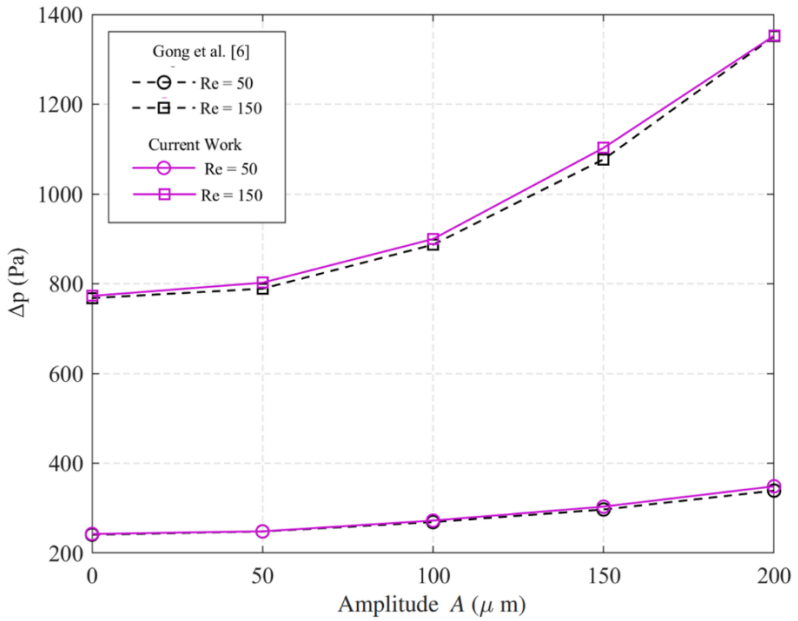
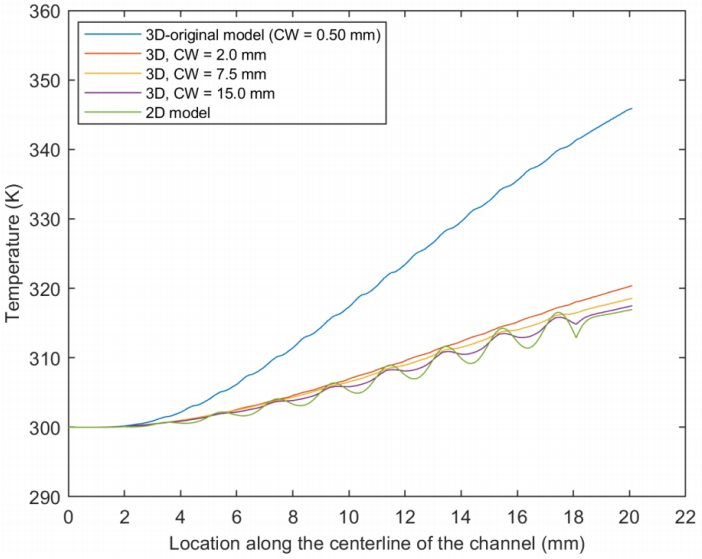


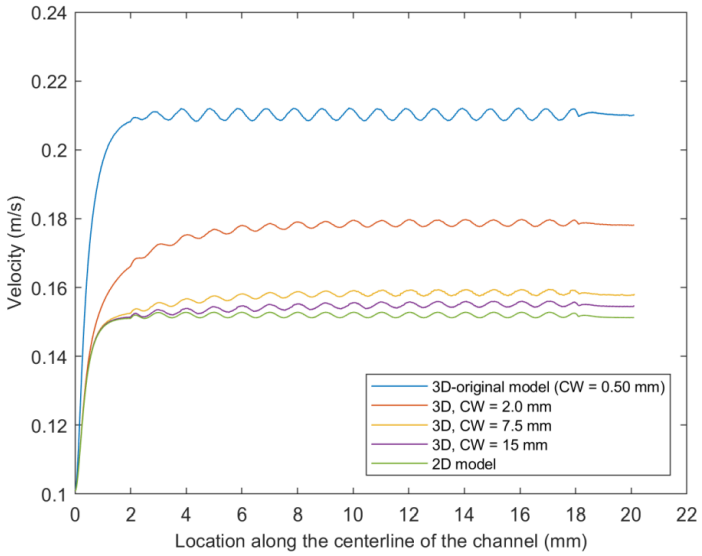
Figure 4: Comparison of results for Δp vs. A for $Re = \{50, 150\}$.

Once our three-dimensional models, with channel width of $CW = 0.5$ mm, were validated, CW of each model systematically increased in the range of $[0.5, 20]$ mm, while keeping all the other geometrical parameters constant. purposes of comparison, a two-dimensional model of the system was also built and solved under the same operating conditions. In all cases, the solutions are obtained in terms of velocity, pressure and temperature for the fluid in the microchannel, and temperature distribution in the solid.

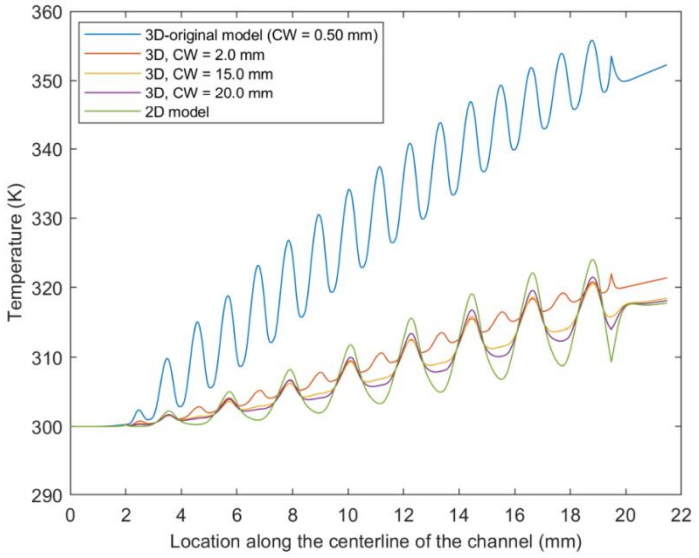
Figures 5 and 6 present, respectively, the results for fluid temperature and velocity along the *centerline* of the channel, for $Re = 50$ and $A = 50\ \mu$m and $A = 200\ \mu$m. These results are presented for the three-dimensional original model,



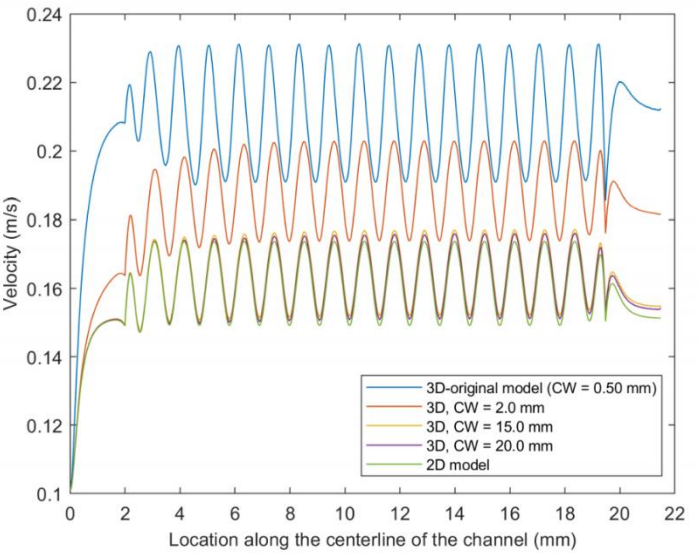
(a) Temperature



(b) Velocity



(a) Temperature.



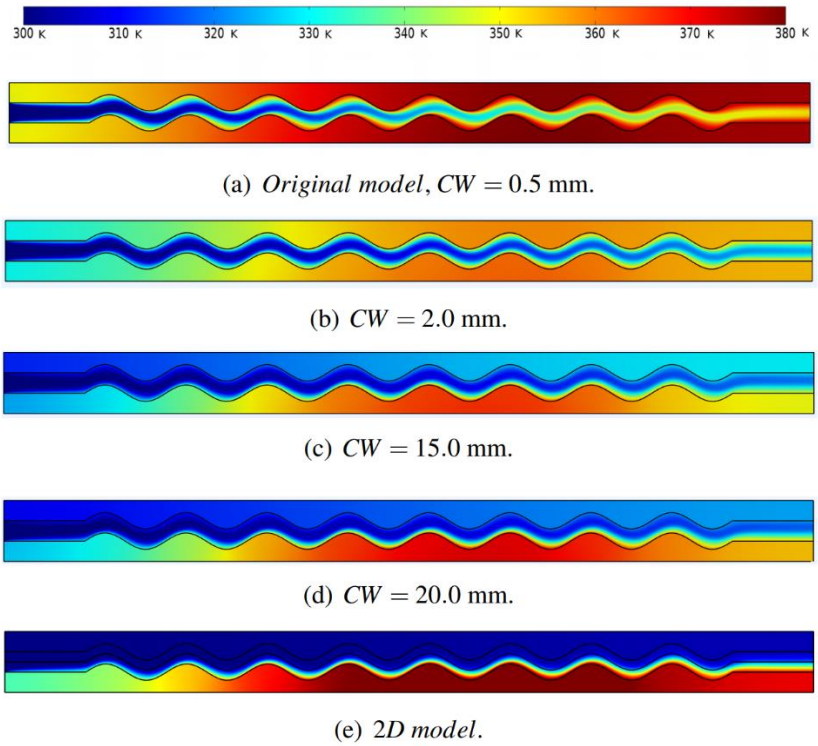
(b) Velocity.

Figure 5: Temperature and velocity profiles along centerline of the channel for $Re = 50,\ A = 50\ \mu$m.

Figure 6: Temperature and velocity profiles along centerline of the channel for $Re = 50,\ A = 200\ \mu$m.

$CW = 0.5$ mm, representative channel width models and the two-dimensional model. From the figures it can be observed that as the microchannel width increases, the three-dimensional numerical results for centerline velocity and temperature converge to those of the two-dimensional model. This is expected because as the channel width increases, the effect from the lateral walls diminishes. For instance, the two figures show that the centerline temperature and velocity profiles of a model with channel width of 15 mm (and 20 mm) not only qualitatively follows the same trend as those of the two-dimensional model but quantitatively they are very close. An interesting feature, observed from Figures 5 and 6, is the development of periodic profiles for both the centerline temperature and velocity along the length of the microchannels for

models with $A = 50\ \mu$m and $A = 200\ \mu$m. Notably, the amplitude of these profiles correlates with the channel amplitude A.



(a) *Original model*, $CW = 0.5$ mm.

(b) $CW = 2.0$ mm.

(c) $CW = 15.0$ mm.

(d) $CW = 20.0$ mm.

(e) *2D model*.

Figure 7: Temperature contours for $A = 200\ \mu$m and $Re = 50$.

Solutions of the conjugate heat transfer for the different three-dimensional models, as well as the two-dimensional model, are provided in Figure 7, in terms of temperature contours, all computed at the mid-plane $z = 0$, of the entire device. These results are presented for the original model, $CW = 0.5$ mm, representative channel widths $CW = 2$ mm, 15 mm and 20 mm, and the two-dimensional approximation. The parameter values in these calculations are wave amplitude $A = 200\ \mu$m, heat influx $q'' = 47$ W/cm^2, and $Re = 50$. From the figure, it is to note that the temperature in the copper block of the original channel, in all the models, are qualitatively similar – in general. For instance, due to the advective process from the cooling water, it increases along the streamwise direction 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. However, there are significant differences as well. For instance, whereas the fluid and channel-block temperature has higher values over-all, as the channel width is increased, the temperature not only for the fluid but also for the solid decreases. This is in part due to the increased heat transfer area and the reduced effect from the lateral walls. However, a specific feature is the drastic temperature stratification between the lower and upper sections of the channel-block assembly. Whereas the block temperature does not show stratification in the vertical directions, as the width of the channel-block increases, so does the level of stratification, with the largest temperature difference being shown by the three-dimensional model with $CW = 20$ mm, which approaches the solutions obtained from the two-dimensional model. Such behavior is due to the fact that as CW increases, particularly beyond $CW = 15$ mm, the heat transfer by diffusion in the transversal direction decreases significantly, until it becomes zero with the two-dimensional model. Quantitatively, the results for velocity and temperature between the two- and three-dimensional models with large widths ($CW > 15$ mm), are confined to 1.7%.

6. Conclusion

Thermal management of electronic equipment is becoming more challenging with continues development of miniature high power-density electronic devices. For this reason, numerical simulations are very useful in analyzing the system before physical models are built and tested, offering advantage not only in the design process but its optimization. Though three-dimensional models of any system are the most accurate, they can be very expensive computationally, and simpler models can also be effective.

In this work, we have presented a systematic approach to establish the validity of a two-dimensional approximation of three-dimensional single-phase thermal convection in wavy micro-channel. The results of this study indicate that as the channel width is increased, the results in the three-dimensional models converges to that in the two-dimensional model. For the parameters considered, it is reasonable to make a two-dimensional simplification of a channel-block device with a width larger than 15 mm. Thus, in the context of improving microchannel designs with high heat transfer performance and low pressure drop, it may be possible to use a two-dimensional model in the parameter optimization process, since not only it provides qualitatively the same results from that of a more complex three-dimensional model; but provides the results at a fraction of the CPU time, since the simpler model only takes minutes to solve.

Acknowledgements

This work has been supported by the National Science Foundation under Award No. HRD-1547723.

References

[1] S.S. Murshed. “Introductory chapter: electronics cooling—an overview” in *Electronics Cooling*, Croatia: InTech, 2016, pp. 1-10.

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

[3] Y. Sui, C.J. Teo, P.S. Lee, Y.T. Chew, and C. Shu. Fluid flow and heat transfer in wavy microchannels. *Int. J. Heat Mass Transfer*, 53:2760–2772, 2010.

[4] Z. Parlak. Optimal design of wavy microchannel and comparison of heat transfer characteristics with zigzag and straight geometries. *Heat and Mass Transfer*, 54(11):3317–3328, 2018.

[5] D. Yuan, W. Zhou, T. Fu, and C. Liu. Experimental and numerical investigation of heat and mass transfer in non-uniform wavy microchannels. *International Journal of Thermal Sciences*, 152, 2020. Article 106320.

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

[7] E. Grajeda, C. Gaunthier, F. Dugast, J.R. Pacheco, and A. Pacheco-Vega. Conjugate heat transfer analysis of multi-harmonic wavy micro-channels. In *Proceedings of the International Conference on Energy Systems*, pages 1–8, Istanbul, Turkey, 2015. Istanbul, Turkey.

[8] J. Moon, J. Pacheco, and A. Pacheco-Vega. Heat transfer enhancement in wavy micro-channels through multiharmonic surfaces. In ASME 2018 International Mechanical Engineering Congress and Exposition, American Society of Mechanical Engineers Digital Collection. IMECE2018-86425.

[9] J. Moon, J. Pacheco, and A. Pacheco-Vega, 2019. 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)*. ENFHT-120.

[10] S. Zeng, and P.S. Lee. Topology optimization of liquid-cooled microchannel heat sinks: An experimental and numerical study. *International Journal of Heat and Mass Transfer*, 142, 2019. Paper No. 118401.

[11] U. Ghia, K. Ghia, and C. Shin. 1982. High-*Re* solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. *J. Comput. Phys*., 48:387–411, 1982.

[12] C. Canuto, M. Hussaini, A. Quarteroni, and T. Zang. Spectral methods in fluid mechanics. Berlin, Germany: Springer-Verlag, 1988.

[13] I. Demirdzic, Z. Lilek, and M. Peric. Fluid flow and heat transfer test problems for non-orthogonal grids: Benchmark solutions. *Int. J. Numer. Methods Eng.*, 15(3):329–354, 1992.
[14] J. Pacheco, and A. Pacheco-Vega. Analysis of thin film flows using a flux vector splitting. *ASME Int. J. Fluids Engineering*, 125:365–374, 2003.
[15] J. Pacheco, A. Pacheco-Vega, T. Rodic, and R. Peck. Numerical simulations of heat transfer and fluid flow problems using an immersed-boundary finite-volume method on nonstaggered grids. *Numer. Heat Transfer, Part B*, 48(1):1–24, 2005.
[16] A. Pacheco-Vega, J. Pacheco, and T. Rodic. A general scheme for the boundary conditions in convective and diffusive heat transfer with immersed boundary methods. *ASME J. Heat Transfer*, 129:1506–1516, 2007.
[17] D.N. Arnold, F. Brezzi, B. Cockburn, and L.D. Marini. Unified analysis of discontinuous Galerkin methods for elliptic problems. *SIAM Journal of Numerical Analysis*, 39(5):1749–1779, 2002.