Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **1**, No. 2, pp. 231-241 (2009)

April 2009

Numerical Simulation of Heat Transfer in Rectangular Microchannel

Jun Yao[1], Yufeng Yao[1,*], Peter J. Mason[1] and Mayur K. Patel[2]

[1] *Faculty of Engineering, Kingston University, 104 Friars Avenue, Roehampton Vale, London SW15 3DW, United Kingdom.*

[2] *School of Computing and Mathematical Sciences, University of Greenwich, 30 Park Row, Greenwich, London SE10 9LS, United Kingdom.*

Received 11 November 2008; Accepted (in revised version) 24 December 2008

Available online 17 March 2009

Abstract. Numerical simulation of heat transfer in a high aspect ratio rectangular microchannel with heat sinks has been conducted, similar to an experimental study. Three channel heights measuring 0.3 mm, 0.6 mm and 1 mm are considered and the Reynolds number varies from 300 to 2360, based on the hydraulic diameter. Simulation starts with the validation study on the Nusselt number and the Poiseuille number variations along the channel streamwise direction. It is found that the predicted Nusselt number has shown very good agreement with the theoretical estimation, but some discrepancies are noted in the Poiseuille number comparison. This observation however is in consistent with conclusions made by other researchers for the same flow problem. Simulation continues on the evaluation of heat transfer characteristics, namely the friction factor and the thermal resistance. It is found that noticeable scaling effect happens at small channel height of 0.3 mm and the predicted friction factor agrees fairly well with an experimental based correlation. Present simulation further reveals that the thermal resistance is low at small channel height, indicating that the heat transfer performance can be enhanced with the decrease of the channel height.

AMS subject classifications: 76D05, 76M12, 80A20

Key words: Computational fluid dynamics, rectangular microchannel, scaling effect, thermal resistance.

1 Introduction

In an experimental study of microchannel flow about twenty years ago, Tuckeman

*Corresponding author.
URL: `http://engineering.kingston.ac.uk/people/details.php?AuthorID=3826`
Email: J.Yao@kingston.ac.uk (J. Yao), y.yao@kingston.ac.uk (Y. Yao), m.k.patel@gre.ac.uk (M. K. Patel), P.Mason@kingston.ac.uk (P. J. Mason)

and Pease [1] discovered, for the first time, that the heat transfer in a narrow flow channel can be enhanced by reducing the height or diameter down to microscale. This finding initiated numerous in-depth studies over the last decades that lead to the microchannelling technology. The technique has now been widely used in various fields, e.g., the development of high efficient cooling devices by applying enhanced heat transfer [2]. Recently, research activities have been primarily focused on the heat transfer performance in microchannel for growing number of microengineering and biomedical applications [3, 4]. While the experimental approach has been proved to be very successful in microchannel flow study [5], the methodology is often constrained by two key facts: first, the current ability for machining microstructures with the restriction of instrumentation and second, the limitation of measurements on heat transfer parameter along the channel wall due to its very small scale and three-dimensionality associated with other complex physical phenomena. Because of these two reasons, experimental measurements often exhibit significant discrepancy, and one typical example is the dependency of fraction factor on the channel spacing, for which the experimental measurements have shown significant scattering. Furthermore, some contradictive findings were often found in published results by various researchers. As an example, Pfahler et al. [6] performed a study of rectangular microchannel flow with small hydraulic diameters of 1.6$\sim$3.4 μm and Reynolds number of 50$\sim$300, and they revealed that the measured friction factor was consistently lower than theory. Peng and Peterson [7] also performed a similar study applying water flow through rectangular machined steel grooves with large hydraulic diameters of 133$\sim$367 μm and found that the friction factor was not inversely proportional to the Reynolds number in the laminar region, in contrary to theory. On the other hand, studies by Rahman and Gui [8] and Qu and Mudawar [9] concluded that the measured friction factor and pressure drop agree well with theory in the laminar regime. Thus there are clear demands of further refinement studies to clarify these contradictive findings.

With the advancement of numerical simulation technique, one recent trend is to perform numerical study for microchannel flow problem, such that any difficulties associated with micro-manufacturing and instrumentation could be easily avoided. Among various methods, two methodologies are often adopted for microchannel flow simulations as (1) the lattice Boltzmann modelling [20, 21] with a kinetic approach, and (2) the Navier-Stokes modelling with a continuum approach. The present study adopts the latter approach, following some previous studies [10,11]. By referencing to the experiment of Gao et al. [5], Gamrat et al. [10] carried out a numerical simulation of flow configuration same as that in the experiment. While the simulation results agree well with the test data for channel heights greater than 0.3 mm, no scaling effect was found at a small channel height of 0.3 mm, which in fact has been observed by Gao et al. [5] and other researchers. To verify this, present authors re-visited the case by performing a numerical study [11] and not surprising, the scaling effect at small channel height was predicted, in agreement with the experimental observations. Our study was extended further to a smaller channel height of 0.1 mm where the scaling effects are much more significant. A correlation relating the Nusselt number and channel height was derived that fits well with the available test data. Following this successful work, here we continue the study to include the thermal characteristics, in order to evaluate the influence of Reynolds number and channel heights on heat transfer performance, particularly the Poiseuille number, the friction factor and the thermal resistance. The numerical predictions will be compared with available test data, empirical corrections and theory.

2 Problem definition, governing equations and numerical solution

We considered a microchannel flow with laminar characteristics, similar to that studied experimentally by Gao et al. [5] and numerically by Gamrat et al. [10]. Fig. 1 depicts a sketch of the configuration used in the present study. Following the experimental setup, the channel walls have two 'bronze' blocks, separated by a 'stainless' plate (at a thickness of 'e' indicating the channel height) and a hollow section at the centre, in which the size of width 'w' is 25 mm and the length of 'L' is 86 mm. Two rectangular sumps are 'machined' in the solid blocks with inlets and outlets positioned at the block ends. The chambers are also linked to the entrance and the exit of the channel. The channel height can be adjusted by varying the 'steel' plate thickness in a range of 0.1$\sim$1 mm. Four electric cartridges are inserted inside two blocks in symmetry manner and are surrounded by a layer of insulting material. The length of the electric cartridges is slightly shorter than the microchannel length L. Comparing to a curved microchannel entrance described in Gao et al.'s experimental study [5], we have used rectangular entrances to achieve a uniform inlet velocity and minimise any possible distortions at the channel entrance. The same mass flow rate, at the two chamber inlets, is applied to ensure that the flow remains symmetric while entering the channel entrance. The heat losses from such an arrangement are mainly concentrated at the channel exit area as the heat sinks are approximately 25% shorter than the channel length.

We consider the governing Navier-Stokes (NS) equations for incompressible flow in conservation form as

$$
\begin{aligned}
&\frac{\partial u_j}{\partial x_j} = 0,\\
&\rho\frac{Du_i}{Dt} = -\frac{\partial p}{\partial x_i} + \mu\frac{\partial \tau_{ij}}{\partial x_j},\\
&\rho\frac{DE}{Dt} = -\frac{\partial (pu_j)}{\partial x_j} + \frac{\partial u_i\tau_{ij}}{\partial x_j} + \frac{\partial}{\partial x_j}(\mu\frac{\partial T}{\partial x_j}),
\end{aligned}
$$

where x_j are the Cartesian coordinates; u_j the Cartesian velocity components; ρ the density; p the pressure; μ the dynamics viscosity; τ_{ij} the shear stress tensor; E the total energy; and T the temperature, respectively.

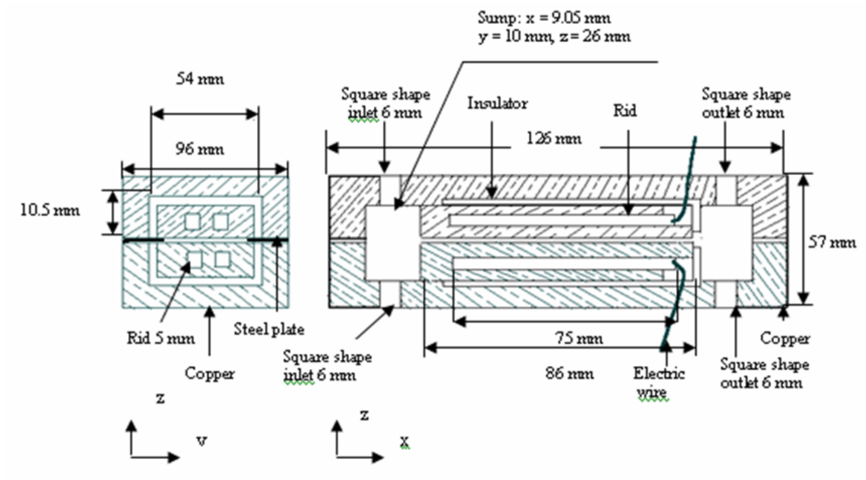

Figure 1: A sketch of configuration with plain view at two cross-sections.

In computation, the domain has two components, fluid and solid. The fluid domain consisting of two chambers, each having two inlets/outlets and are linked with microchannel channel in the middle. The solid domain, surrounding the channel and chambers, is embedded with four square shaped heat sinks. Based on the geometry, quality multi-block structured mesh was generated for each rectangular sub-domain. Across the fluid and the solid domains, mesh lines were connected smoothly to ensure no numerical errors, due to interpolation, were introduced. Standard hydraulic boundary conditions of uniform inflow at the two chamber inlets were applied and the thermal boundary conditions for solving heat conduction equation were (1) The heat flux of electrical cartridge was kept constant at 180 W and the heat source was uniformly distributed over the fluid/solid interfaces; (2) Water at ambient temperature (300 K) and heat transfer coefficient of 10 $\mathrm{Wm^{-2}K^{-1}}$; and (3) Adiabatic wall conditions were prescribed for the remaining surfaces. These thermal conditions follow an earlier experimental study of Gao et al. [5] and a later numerical study of Gamrat et al. [10], both showed that these thermal conditions are suitable for the case studied.

The numerical simulation of this type of flow involving conjugate heat transfer was preformed using the commercial software PHOENICS [12]. The governing equations for incompressible fluid flow as described in the above were solved by conventional finite volume methodology together with the pressure-velocity coupling technique, commonly referred to as the SIMPLE algorithm for satisfying the continuity of flow field. The fluid (water in present simulation) in the domain was assumed to be incompressible, having laminar flow characteristics with constant fluid properties such as density and temperature-dependent fluid properties such as viscosity. The buoyancy forces and radiation effects were neglected. At meantime, the temperature-dependent effect of thermal development in the solid domain was resolved through the heat conducting equation and local momentum transport, providing the constraints for the boundary conditions in the flow domain. The conventional non-slip boundary condition and adiabatic thermal wall condition are used, following some previous studies of Shah and London [13], Gamrat et al. [10] and others. However, the suitability of applying the non-slip conditions in the continuum NS modelling for micro-scale flows is still debatable and some recent studies using the lattice Boltzmann approach [20, 21] has revealed noticeable differences in results between the 'non-slip' and the 'slip' conditions. It is generally accepted that for micro-scale of very small Knudsen number, the 'non-slip' boundary condition has certain limitations and thus the 'slip' boundary condition treatment would be more reasonable of representing the flow physics of the solid boundary effects.

Simulations for a series of grids at different density levels were carried out in order to identify a baseline mesh, which would provide grid-independent solutions. For each computation, the convergence criteria for the velocity residuals were below 10^{-5}, and in most cases, this can be achieved after around 500 iterations. The final baseline mesh has 98400 cells for all simulations (with channel heights of 0.3$\sim$1 mm), i.e. $164\times30\times20$, in streamwise, spanwise and wall-normal, respectively. This grid resolution was comparable to that used by Gamrat et al. [10] in their numerical study, where they found that the computed channel velocity profile was collapsed for 20, 30 and 40 grids in the wall-normal direction, but some differences appeared for grid densities between 80 and 164 in the streamwise direction. Thus, the higher limit of 164 grid points was used in the present study. As the problem was basically two-dimensional (due to a high aspect ratio applied), there were no significant influences from a spanwise resolution, as confirmed by Gamrat et al. [10]. A validation simulation of a laminar flow at Reynolds number of 2166 was performed. The present temperature distributions agree qualitatively well with that reported by Gamrat et al. [10]. The predicted temperature has a value of 295.8 K at the channel entrance, which is exactly the same as that reported by Gamrat et al., and the temperature increases to around 296.7 K at the channel exit, about 0.3% lower than that estimated by Gamrat et al.

3 Results and discussions

The numerical study of Gamrat et al. [10] suggested that the characteristics of flow and heat transfer in the microchannel have significant dependency on the Reynolds number, the Prandtl number and the shape of geometry. Our previous study [11] confirmed these observations and in addition, had predicted the scaling effect at small channel height of less than 0.3 mm, in consistent with the experiment of Gao et al. [5]. In this paper, we focus on key parameters related to the hydraulic and thermal performance, notably the Poiseuille number, the friction factor and the thermal resistance in the microchannel subject to the Reynolds number and the channel height change.

Same as that in [11], the characteristic hydraulic diameter length (D_h) is defined as D_h=$4C_A/W_p$, where C_A is the cross-section area and W_p the wetted perimeter, both measured at the channel entrance. The Reynolds number considered varies from 300 to 2360, similar to that used in the experimental investigation of Gao et al. [5] and the numerical study of Gamrat et al. [10].

Previously, numerous studies have revealed that the hydrodynamic properties of laminar flow, along the microchannel, are characterised by the Poiseuille number variations with respect to the dimensionless coordinate. Shah and London [13] proposed an analytical formula based on developing laminar flow in a two-dimensional channel as

$$P_{O_{lam}}(x^+) = \frac{3.44}{\sqrt{x^+}} + \frac{24 + 0.1685/x^+ - 3.44/\sqrt{x^+}}{1 + 2.9 \times 10^{-5}/(x^+)^2}, \tag{3.1}$$

where the dimensionless coordinate is defined as $x^+ = x/(D_h R_e)$ and x is the channel streamwise coordinate.

Fig. 2 illustrates the predicted Poiseuille number distributions along the dimensionless coordinate x^+ in comparison with the numerical prediction by Gamrat et al. [10] and theory of Shah and London [13] at two chosen Reynolds numbers. While the Nusselt number variations agree very well with theory (not shown here, but can be found in Yao et al. [11]), the Poiseuille number variations from two numerical studies have quite similar trend, but neither of them has shown satisfactory agreement with the theoretical estimation. In details, the prediction by Gamrat et al. [10] under-predicts the theory throughout with higher deficits at smaller x^+ while the deficit reduces at larger x^+. The present results agree fairly well with theory for $x^+ = [0.006, 0.009]$, but over-predict at small x^+ and under-predict at large x^+. The cause of the discrepancy is not very clear yet and the possible reasons may be attributed to the boundary layer development along the channel wall and the entrance/exit effects, as previously discussed by Kandlikar and Grande [14]. The numerical predictions of Gamrat et al. [10] also exhibits certain data scattering at the channel entrance region, providing another supporting evidence for this claim. Downstream near the exit, results are well collapsed for two channel heights.

Simulation continues for another ten Reynolds numbers at two channel heights of 1 mm and 0.3 mm. Fig. 3 gives the Poiseuille number variation against dimensionless length of L^+ defined as $L^+ = L/(D_h R_e)$, note that here L is the channel length, not the x-coordinate shown in Fig. 2. For all Reynolds numbers tested, L^+ varies from 0.01 for high Re and large channel height to 0.17 for low Re and small channel height. Numerical predictions show an initial sharp drop, followed by a moderate decrease until a final near content value of about 20 reached for the L^+ greater than 0.12, where the entrance effect would be almost negligible. Based on the Eq. (3.1), the classical theory for the fully developed laminar flow would have the Poisueille number of 24 (for infinitely large L^+) that is higher than numerical predictions by a factor of 1.2. Despite of this, at small L^+ (i.e., large Re and channel height), numerical results compared reasonably well with the theory, thus leads to some doubts on the applicability of theory for small Re and channel height conditions. Further experimental and numerical studies are thus desirable, particularly for low Reynolds number and small channel spacing to verify the cause of this discrepancy.

The Darcy friction factor for laminar developing flow in two-dimensional channels

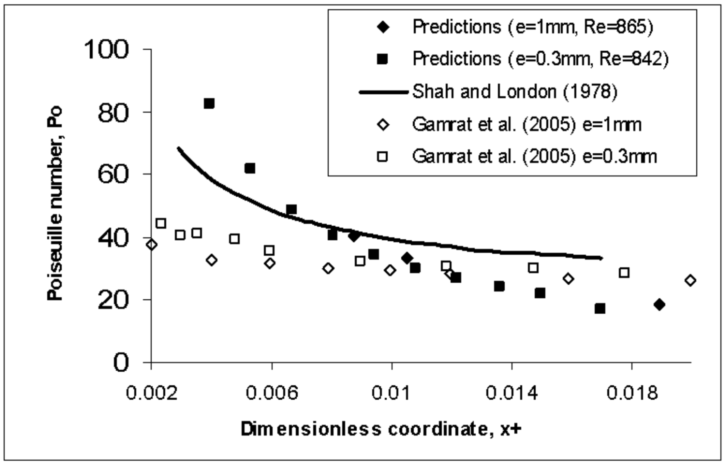

Figure 2: Comparison of numerical predictions with theoretical estimation for the Poiseuille number variations along the channel.

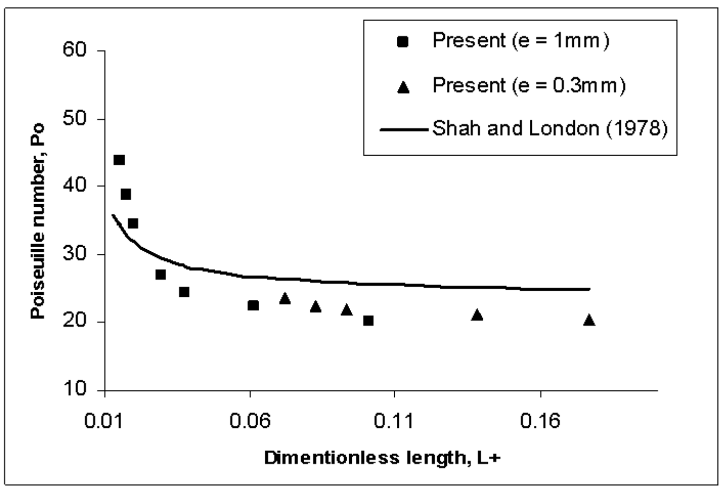

Figure 3: Effects of Reynolds number and channel height on numerical predictions in comparison with theoretical estimation.

can be determined by a theoretical solution proposed by Hartnett and Kostic [15] by using a polynomial fitting for various aspect ratios (α) as

$$fR_e = 96(1 - 1.3553\alpha + 1.9467\alpha^2 - 1.7012\alpha^3 + 0.9564\alpha^4), \tag{3.2}$$

where the friction factor $f{=}\triangle p/(0.5\rho V_{ave}^2) \times D_h/x$ can be evaluated by the pressure drop over the flow developing length scale x, the hydrodynamic diameter D_h, and V_{ave} is the averaged velocity in the channel. The density of water is kept constant.

Fig. 4 depicts the predicted friction factor variations against the Reynolds number for three channel heights of 0.3 mm, 0.6 mm and 1.0 mm. Additional results from theoretical estimation of Eq. (3.2) by Hartnett and Kostic [15] and an experimental based correlation proposed by Shen et al. [16] are also plotted for comparisons. At low Reynolds number of about 300, the influence of channel height is evident in numerical results of the friction factor that over-predict theory and test data. With the Reynolds number increases to greater than 500, the difference between three channel heights decreases and the predictions agree test data fairly well until maximum Reynolds number simulated. Results also show that for R_e>800, the friction factors are almost identical for two large channel heights (0.6 mm and 1.0 mm), but have higher value for small channel height of 0.3 mm. This observation proves that the scaling effect only appears at channel height less than 0.3 mm, in agreement with the experimental observation by Gao et al. [5]. While the theory of Hartnett and Kostic [15] agrees reasonably well with test data at small Re, there is a clear departure from test data for large R_e. Furthermore, the influence of channel height via aspect ratio α is almost negligible. While numerical simulation predicts the scaling effect for small channel height which has been verified by other researchers, somehow this does not echoed in the correlation of Shen et al. [16]. One possible reason might be the incorrect account for the wall roughness in their experiment. Several evidences could be used to support this. In an experimental study, Kandlikar et al. [17] revealed the relationship between the wall roughness and the friction factor by testing two capillary tubes of 1.076 mm and 0.6 mm in diameter with higher wall roughness ranging from 0.00178 to 0.33. They concluded that the channel wall roughness effect could be neglected for large tube (diameters >1 mm) where smooth wall condition can normally be used, whereas for small tube (diameters <0.6 mm), the surface roughness has considerable affects on the friction factor and it increases proportional to the surface roughness. Although it is not the purpose for this study to consider the wall roughness effect, at least it provides a good topic for future studies, numerically and experimentally.

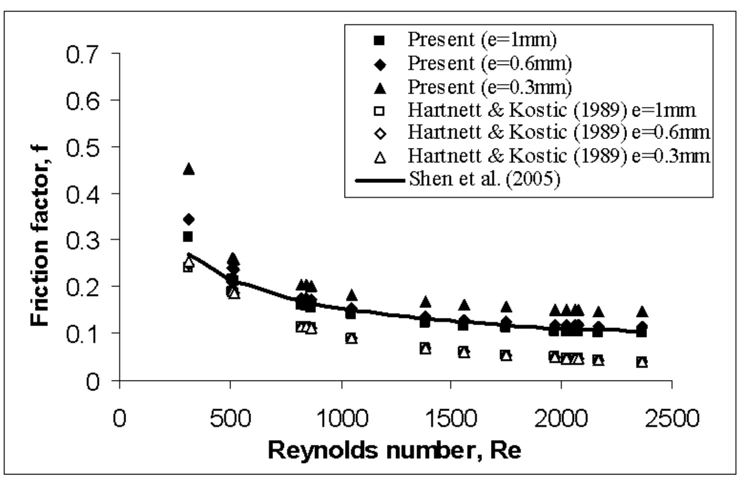

Figure 4: Comparison of the friction factor variation at different Reynolds number and channel heights.

Finally, we evaluate the thermal resistance defined as $R_t = \triangle T_m / q$, where the temperature deficit $\triangle T_m$ is calculated in the same manner as that in Shen et al. [16]. Fig. 5

shows the predicted thermal resistance variations against all tested Reynolds numbers of wider range 300∼2360. For three channel heights considered, simulation results have shown quite good consistency with higher thermal resistance value for larger channel height and lower value for smaller channel height, indicating that small channel would have low thermal resistance thus the better thermal performance. As this type of data are limited in public domain, here comparisons are only made qualitatively by referencing to the numerical results of Fedorov and Viskanta [18] and the experimental measurements of Kawano et al. [19], both are at fairly low Reynolds numbers up to 500, while present results are at high Reynolds number up to 2360. In general, three sets of results exhibit similar trends, although at low R_e, our predictions are higher than that of Fedorov and Viskanta [18] and Kawano et al. [19], probably due some model uncertainty issues. By extending to high Re, our results further predicts a low limit boundary of heat resistance about 0.05 as seen in Fig. 5. This finding would be valuable for practical application, subject to further experimental verification.

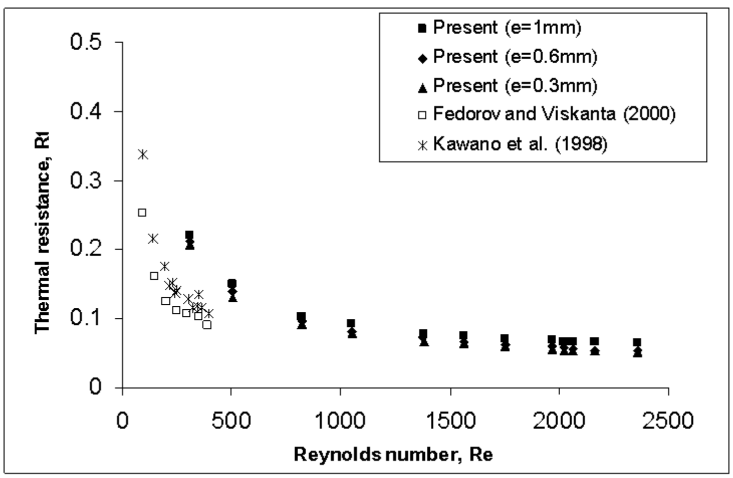

Figure 5: Effects of channel heights on the variation of thermal resistance with the Reynolds number.

4 Conclusions

We present a numerical study of heat transfer in a high aspect ratio rectangular microchannel with heat sinks using a three-dimensional CFD model. Following configuration of an experimental study by Gao et al. [5], our study has successfully verified a previous numerical study carried out by Gamrat et al. [10] and furthermore, confirmed the experimental observation of scaling effect at small channel height of less than 0.3 mm, which was not produced in the study of Gamrat et al. [10]. Further comparisons of thermal characteristics such as the friction factor and thermal resistance with theory and other available test data reveals the existence of scaling effect in these quantities.

In summary the following findings can be drawn based on the present study: (1) while the predicted Nusselt number agrees well with theory, the predicted Poiseuille number shows some under-predictions compared to theory, in consistent to that reported by Gamrat et al. [10]; (2) scaling effect on the friction factor variation is evident at small channel height of less than 0.3 mm. For large channel heights, scaling effect does not exist, in agreement of the experimental observations. The numerical results are also in good comparisons with an experimental based correlation, while theoretical estimation shows clear under-prediction with no scaling effects at small channel height; (3) scaling effect also affects the thermal resistance of microchannel flow with low thermal resistance is observed in the case of small channel height, indicating better thermal performance. At low Reynolds number regime, the qualitative comparison with available data confirms the similar trend, although further experimental and numerical studies are suggested, particularly at high Reynolds number regime to verify the low limit boundary of thermal resistance predicted in present study.

References

[1] D. B. Tuckeman and R. F. W. Pease , *High performance heat sink for VLSI*, IEEE Electron Device Letters EDL-2, 5 (1981), pp. 126-129.
[2] P. S. Lee and S. V. Garimella, *Thermally developing flow and heat transfer in rectangular microchannels of different aspect ratios*, Int. J. Heat and Mass Transfer, 49 (2006), pp. 3060-3067.
[3] T. Chovan and A. Guttman, *Microfabricated devices in biotechnology and biochemical processing*, Trends in Biotechnol, 20(3) (2002), pp. 116-122.
[4] B. Gromoll, *Micro cooling systems for high density packaging*, Revue Generale de Thermique, 37(9) (1998), pp. 781-787.
[5] P. Gao, S. Le Person and M. Favre-Marinet, *Scale effects on hydrodynamics and heat transfer in two-dimensional mini and microchannel*, Int. J. Thermal Science, 41 (2002), pp. 1017-1027.
[6] J. Pfahler, J. Harley and H. Bau, *Liquid transport in micron and submicron channels*, Sensors and Actuators, A21-A23 (1990), pp. 431-434.
[7] X. F. Peng and G. P. Peterson , *Frictional flow characteristics of water flowing through rectangular microchannels*, J. Exp. Heat Transfer, 7 (1994), pp. 249-264.
[8] M. M. Rahman and F. Gui, *Experimental measurements of fluid flow and heat transfer in microchannel cooling passages in a chip substrate*, Advances in Electronic Packaging ASME EEP, 4 (1993), pp. 495-506.
[9] W. Qu and I. Mudawar, *Experimental and numerical study of pressure drop and heat transfer in a single-phase microchannel heat sink*, Int. J. Heat Mass Transfer, 45 (2002), pp. 2549-2565.
[10] G. Gamrat, M. Favre-Marinet and D. Asendrych, *Conduction and entrance effects on laminar liquid flow and heat transfer in rectangular microchannels*, Int. J. Heat Mass Transfer, 48 (2005), pp. 2943-2954.
[11] J. Yao, Y. Yao, M. K. Patel and P. J. Mason, *On Reynolds number and scaling effects in microchannel flows*, Eur. Phys. J - Appl. Phys., 37(2) (2007), pp. 229-235.
[12] PHOENICS version 3.4 user's manual CHAM Ltd., (2001).
[13] R. K. Shah and A. L. London , *Laminar flow forced convection in ducts*, Advances in Heat Transfer Academic Press, New York, (1978).

[14] S. G. KANDLIKAR AND W. J. GRANDE , *Evolution of microchannel flow passages*, Thermohydraulic performance and fabrication technology Heat Transfer Engineering, 24(1) (2003), pp. 3-17.

[15] J. P. HARNETT AND M. KOSTIC , *Heat Transfer to Newtonian and non-Newtonian fluids in rectangular ducts*, Advances in Heat Transfer, 19 (1989), pp. 247-356.

[16] S. SHEN, J. L. XU, J. J. ZHOU AND Y. CHEN, *Flow and heat transfer in microchannels with rough wall surface*, Energy Conversion and Management, 47 (2006), pp. 1311-1325.

[17] S. G. KANDLIKAR, S. JOSHI AND S. TIAN , *Effect of channel roughness on heat transfer and fluid flow characteristics at low Reynolds numbers in small diameter tubes*, Proc. of NHTC'01, 35th Nacional Heat Transfer Conference, Anaheim, California, NHTC01-12134, (2001).

[18] A. G. FEDOROV AND R. VISKANTA, *Three-dimensional conjugate heat transfer in the microchannel heat sink for electronic packaging*, Int. J. Heat Mass Transfer, 43 (2000), pp. 399-415.

[19] K. KAWANO, K. MINAKAMI, H. IWASAKIAND AND M. ISHIZUKA, *Micro channel heat exchanger for cooling electrical equipment*, Heat Transfer Equip. Syst. Educ., ASME HTD-361-3/PID-3 (1998), pp. 173-180.

[20] Y. T. CHEW, X. D. NIU AND C. SHU, *Three-dimensional lattice Boltzmann BGK model and its application to flows with heat transfer in a rectangular microchannel*, Int. J. Num. Methods in Fluids, 50(11) (2006), pp. 1321-1334.

[21] F. VERHAEGHE, L. S. LUO, AND B. BLANPAIN, *Lattice Boltzmann modeling of microchannel flow in slip flow regime*, J. Comput. Phys., 228(1) (2009), pp. 147-157.