Numerical approach for fluids flow and thermal convection in microchannels

Wagd Ajeeb1,2*, Monica S A Oliveira1, Nelson Martins1 and S M Sohel Murshed2

1 Centre for Mechanical Technology and Automation (TEMA), Department of Mechanical Engineering, University of Aveiro, 3810-193 Aveiro, Portugal.
2 IDMEC, Department of Mechanical Engineering, Instituto Superior Técnico, University of Lisbon, 1049-001 Lisbon, Portugal.

*Email: wagdajeeb@tecnico.ulisboa.pt

Abstract. The heat transfer performance of conventional thermal fluids in microchannels is an attractive method for cooling devices such as microelectronic applications. Computational fluid dynamics (CFD) is a very significant research technique in heat transfer studies and validated numerical models of microscale thermal management systems are of utmost importance. In this paper, some literature studies on available numerical and experimental models for single-phase and Newtonian fluids are reviewed and methods to tackle laminar fluid flow through a microchannel are sought. A few case studies are selected, and a numerical simulation is performed to obtain fluid flow behaviour within a microchannel, to test the level of accuracy and understanding of the problem. The numerical results are compared with relevant experimental results from the literature and a proper methodology for numerical investigation of single-phase and Newtonian fluid in laminar flow convection heat transfer in microscale heat exchangers is defined.

1. Introduction

The rapid development in microsystem technology including microelectronic cooling systems, chemical processing, aerospace and other applications [1], demands the development of a particular heat exchange system for achieving the required temperature and increasing the device shelf life. The latter presents a continuing challenge to surpass the limitations of such thermal systems when overall size, saving energy and cost reduction is mandatory. Microchannels for heat transfer applications present high thermal performance since they detain a wide heat transfer surface area between the operating fluid and the device, lower fluid requirement and lower operational cost [2], [3]. They have attributed to microchannel-based cooling devices a higher thermal performance in comparison with traditional macro-channel based heat exchange systems [4]. However, a literature review by Rosa et al. [5] considered the channels whose hydraulic diameter lies between 1 µm and 1 mm as a microchannel, following a study carried by Celata et al. [6].

On the other hand, the increase in pressure drop in a microchannel leads to a considerable rise in pumping power. Moreover, proper consideration should be given to the scaling effects since they may influence fluid flow and heat transfer characteristics through the microchannels [7].

The literature highlights a few studies on convection heat transfer, concerning the performance of microchannels and their effectiveness regarding heat transfer enhancement[8], [9]. The studies have
mentioned that factors such as shape, viscous heating, dependent-temperature thermophysical properties and entrance region (scaling effects) can’t be neglected when studying heat transfer and fluid flow, and these factors become very influential in mini/micro channels in comparison with conventional channels [5]. The scaling effects have been highlighted in detail by Morini et al. [10] and they reported significant influence on heat transfer and fluid flow in microchannels. Moreover, as the hydraulic diameter of the channel decreases, the effects of variation of thermophysical properties and viscous dissipation increases and significantly affect the thermal and rheological behaviour of fluid flow through the microchannels, affecting heat transfer performance [11], [12].

However, an accurate and validated single-phase numerical model of microscale heat transfer systems is the key element to justify and clarify the performance of fluid flow in a microchannel for various operating conditions. Also, several issues should be considered in terms of the validation of some assumptions such as continuum fluid, applying Navier-Stokes equations and boundary conditions. Therefore, this study provides a proper methodology for establishing a numerical approach to investigate fluid flow and heat transfer of conventional fluids such as water through microchannels. This study takes a part in the current trend toward elements miniaturization and higher energy efficiency of all heat transfer applications in industries.

2. Microchannel numerical model development and validation

2.1. Numerical modelling and methodology

In order to reproduce numerically the experimental studies of fluid flow through microchannels, the numerical approach is achieved using the laminar model of CFD tools by ANSYS-FLUENT version 19.0. Where 2D Navier–Stokes and energy equations are used to describe the laminar flow and heat transfer of single-phase and Newtonian fluid such as water, including the viscous dissipation effect in the simulations. The finite volume method (FVM) is employed to discretize the governing equations, and the SIMPLE algorithm is chosen to couple pressure and velocity. Also, a second-order upwind method is used to include the convective and diffusive terms. In addition, a pressure-based solver is implemented. The convergence criteria are set such that the residual errors for continuity, momentum and energy are reduced to less than $10^{-6}$. Moreover, some assumptions were considered for the current simulations as following: 1- incompressible fluid (for water); 2- continuum fluid flow and steady-state; 3- laminar flow; 4- no-slip boundary condition at the wall; 5- variable thermophysical properties for the fluid (temperature-dependent); 6- constant and uniform wall heat flux. The latter, numerical method and assumptions, were chosen according to the problem of the selected experimental studies, following the steps and suggestions of the relevant literature on the subject [13]. The microchannel geometry is chosen to have a hydraulic diameter $D_h > 25.9 \, \mu m$, where it was found in a study by Wu et al. [14] that applying numerical approach for the fluid flow in microchannel can be valid for a minimum diameter of 25.9 \, \mu m.

In order to take into consideration the variable (temperature-dependent) thermophysical properties of the fluid (distilled water), third-order polynomial fitting curves of the experimental data from the literature (i.e., [15], [16]) are used to obtain the correlations for the dependent-temperature thermal conductivity $k$ (1) and viscosity $\mu$ (2) of distilled water. Thus, the correlations of the thermal conductivity and viscosity are as following:

\[
k = -1.470E-07 \, T^3 + 1.275E-04 \, T^2 - 3.531E-02 \, T + 3.701E+00 \tag{1}
\]

\[
\mu = -5.098E-09 \, T^3 + 4.955E-06 \, T^2 - 1.6156E-03 \, T + 1.772E-01 \tag{2}
\]

Moreover, the mesh optimization and numerical results-independent process were carried out individually for each case of the reproduced studies. The optimal mesh is chosen so that it achieves results-independency and good agreement with the experimental data, and the nodes number are intensified near the microchannel wall where the velocity and temperature gradients are significant.
2.2. The validation of the numerical methodology

The obtained numerical results are compared with the experimental data from the studies of Zhuo et al. [17] and Lelea et al. [18], as shown in figures 1, 2 and 3. The chosen experimental studies investigated the fluid flow through microchannels with different tube geometries and flow conditions, as well as provide the needed information (parameters and data) for establishing the numerical approach and run the simulations at the same conditions presented in their studies (such as the dimensions of the test section, temperature and fluid flow values). The heat transfer coefficient is obtained by Newton’s law of cooling, as it can be depicted from Equation (3), where $T_w$ is the temperature at the wall, $T_f$ the bulk fluid temperature and $q$ is the imposed heat flux.

$$h = \frac{q}{T_w - T_f} \quad (3)$$

The $Nu$ is calculated based on the heat transfer coefficient value by equation (4).

$$Nu = \frac{h \cdot D}{k} \quad (4)$$

Also, the total pressure drop can be numerically calculated as the pressure difference between the inlet and the outlet of the test section model by the Area-Weighted Average method.

First selected study is by Zhuo et al. [17], where the numerical simulation (in the current study) is carried out for water flow through a stainless steel microtube with an inner diameter of 373µm, outlet diameter of 670 µm and length of 270 mm at a $Re$ of 200, under wall heat flux condition of 36.05 kW/m².

![Figure 1](image_url)

**Figure 1.** Distribution of local $Nu$ along the tube length ($D_h$=373µm) at $Re$=200.

The numerical results of local $Nu$ are compared with the experimental data along the tube and an acceptable agreement is observed with an average deviation of 5%, considering dependent-temperature thermophysical properties (Num-variable, figure 1). It should be noted that a higher average deviation of 17% is observed when the simulation is repeated considering independent-temperature thermophysical properties (Num-constant, figure 1) for water, highlighting the significant impact of dependent-temperature properties. The latter can be explained by the significant effect of the gradient viscosity and gradient conductivity on the fluid velocity and temperature profiles, particularly near the wall where the heat flux is applied.

Second selected study is by Lelea et al. [18], where the numerical model (in the current study) is built for a tube diameter of 0.3 mm and the suitable mesh is chosen to be of 579726 nodes. The investigation carried out for the case of fluid flow without heat flux conditions ($Q = 0$ W) and for the case of applying uniform heat flux of $q$=13.409 kW/m² ($Q = 2$ W). In this study, the numerical results are evaluated and compared with the experimentally obtained ones, for $Nu$ and friction constant ($f$.$Re$). It shows an acceptable agreement between the numerical results and the experimental data with an average deviation of 2% for friction constant and with an average deviation of 5% for $Nu$ set within the experimental uncertainty <10% (figure 2 and figure 3).
Therefore, this study confirms the validity of the used numerical approach (Navier-Stokes equations and energy equation of the 2D physical model of laminar flow) to describe the heat transfer and the fluid flow in microchannels for the mentioned conditions, mainly single-phase, Newtonian fluids and straight microchannels under forced convection heat flux. The latter agrees with the finding of some investigations studying the validity of using a numerical approach for microscale fluid flow [19].

3. Conclusions
In this study, several research works on conventional fluid flow and heat transfer through microchannels have been reviewed for a better understanding of the overall phenomena. This allowed defining a proper methodology for a numerical investigation on single-phase and Newtonian fluids for laminar flows in microchannels considering some parameters such as dependent-temperature fluid properties, viscous heating, hydraulic diameter, and transition Re.

Also, two experimental research in the area were reproduced numerically for the same test section and the operating conditions. The numerical results were in good agreement with experimental data which provides confidence for the used numerical methodology for future numerical investigations of fluids flow and convection heat transfer evaluation in microchannels.

References
[1] I Mudawar 2013 J. Therm. Sci. Eng. Appl. 5 021012–15
[2] Mohammed H A et al 2011 Renew. Sustain. Energy Rev. 15 1502–1512
[3] Tuckerman D B and Pease R F W 1981 IEEE Electron Device Lett. 2 126–129
[4] Wu Z and Sundén B 2014 Renew. Sustain. Energy Rev. 40 11–27
[5] Rosa P, Karayiannis T. G. and M. W. Collins 2009 Appl. Therm. Eng., 29, 3447–3468
[6] Celata G P, 2004 Heat Transfer and Fluid Flow in Microchannels (New York: Begell House)
[7] Herwig H and Hausner O 2003 Int. J. Heat Mass Transf. 46 935–937
[8] Mirzaei M and Dehghan M 2009 Appl. Therm. Eng., 29, 3447–3468
[9] Wu H Y and Cheng P 2006 Int. J. Heat Mass Transf. 49 1803–1811
[10] Morini G L 2006 Heat Transf. Eng. 27 64–73
[11] Chiu H. C et al 2011 Int. J. Heat Mass Transf. 54 34–42
[12] Hooman K and Ejiali A 2010 Int. Commun. Heat Mass Transf. 37, 34–38
[13] Qu W and Mudawar I 2002 Int. J. Heat Mass Transf. 45, 3973–3985
[14] Wu H Y and Cheng P 2003 Int. J. Heat Mass Transf. 46, 2519–2525
[15] ASHRAE 2005 ASHRAE Handbook: Fundamentals. (American Society of Heating)
[16] Bergman T L et al. 2011 Fundamentals of Heat and Mass Transfer (John Wily & Sons 7th ed.)
[17] Li Z, He Y L, Tang G H and Tao W Q 2007 Int. J. Heat Mass Transf. 50 3447–3460
[18] Lelea D and Nishio S 2004 Int. J. Heat Mass Transf. 47 2817–2830