On the application of a modified k-epsilon model for predicting hydrodynamic and thermal turbulent Microchannel flow

A. Balabel[1], A. F. Khadrawi[2]| Ali S. Al-Osaimy
[1,2](CFD-Lab, Mechanical Engineering Dept., Taif University, Al-Haweiah, P.O. 888, Z.C. 21974, Taif, Saudi Arabia, khadrawi99@yahoo.com)

Abstract—*In the present work turbulent thermal and hydrodynamic behaviors of fluid flow in microchannel is investigated numerically using modified k-epsilon model. Different micro-geometries are adopted in this study such as microtubes, isothermal micro two fixed plates and micro Couette flow. The numerical procedure is developed in the present work based on the control volume approach proposed by Patankar. Validation between numerical and experimental or analytical hydrodynamics and thermal behaviors is investigated. It is found that this numerical technique gives the same predictions which are given by the experimental or analytical.*

Keywords—numerical simulation, turbulent, micro-channel, thermo-fluid, modified k-epsilon.

1. INTRODUCTION

Micro-electromechanical devices (MEMs) refer to devices and products whose characteristic length is less than 1 mm but larger than 1 micrometer. MEMs devices are usually part of integrated systems where microchannel flow and heat transfer are found. Micropumps, microturbines, micro heat exchangers, micronozzles, microvalves, micropipes and ducts, biological cells separation reactors and blood analyzers are just a few examples of micro channel flow and heat transfer. Ever continuous demand of miniaturizing devices and gadgets to increase power/weight ratio of products and tools in biological, medical, manufacturing processes, aerospace and many other industrial applications increased the interest of researchers to model and simulate such devices to better understand its physics and the governing laws as well as the fundamental differences of these physics and laws when regarding microscale devices rather than macroscale ones and the dominant parameters and factors imposing these differences. Experimental works have showed that conventional physics of fluid flow and heat transfer described successfully the macroscale devices and processes break down regarding microgeometry flow and heat transfer, where surface-to-volume ratio becomes drastically large and rarified gas flow with nonlinear pressure drop for gaseous flow and granular flows of liquids are observed [1-2]. The research works of momentum and heat transport behaviors in microdomain is the fundamental for developing these new techniques, and is also a foreground in the heat transfer field. The developing microscale thermal and fluidic systems typically have characteristic length of the order of 1–100 μm and often operate in gaseous environments at standard conditions, where the molecular mean free path is in the order of 100 nm. Knudsen number, which is defined as the ratio of the mean free path of gas molecules to the characteristic length, is in the range from to, 0.001- 0.1 [3]. There has been an enormous interest in research works of micro-system over the last decade [4-9]. The single-phase forced flow convection of water and methanol flowing through microchannels with rectangle cross section is investigated [10]. They found that the fully developed turbulent flow was achieved at Re = 1000–1500. The friction factors of water in stainless steel and fused silicon microtubes with diameters ranging from 50 to 254 μm is measured [11]. It is found that an early transition from laminar to turbulent flow occurs when at Re = 600–900, while the fully developed turbulent flow was achieved at Re > 1500. Although early transition in microtubes or microchannels has been observed by some researches, it is interesting to note that, in the works of other researchers, there is no evidence of early transition occurring at all. Judy and Maynes [12] performed experiments on frictional characteristics of flow in microtubes of fused silica and stainless steel. The capillary diameters varied from (15 – 150) μm and three different fluids (water, methanol and isopropanal) were tested. The authors found no evidence of transition to turbulent flow in the range Re < 2000 as reported by other works. Recently, Celata et al. [13,14] studied single-phase heat transfer and flow in circular micro pipes in diameter from (70 to 326) μm through pressure measurement. It is found that the classical Hagen Poiseuille law for friction factor is respected for all diameters measured and Re > 300. Integrated pressure sensors in microchannels pressure drop in microchannels with hydraulic diameters ranging from (25-100) μm are investigated and developed [15]. The results suggest that friction factors for microchannels can be accurately determined from data for standard large channels. Additionally, possible reasons that caused the deviation from classical theory based on macro flow had been discussed in several previous works. The large inconsistencies in previously published data are probably due to the entrance, disturbance from roughness and viscous dissipation effect for liquid flows in micro channels [16-18]. Sharp et al. [19] investigated the transition to turbulent flow for liquid in micro tubes with diameters between (100 and 247) μm by pressure drop and micro-PIV measurements. Transitional and turbulent flow in a circular micro tube is investigated experimentally [20]. In this study the stream wise mean velocity profiles and turbulence intensities were measured at Reynolds number ranging from 1540 to 2960. Experimental results indicate that the flow transition from laminar to turbulent occurs at Re = 1700–1900 and the turbulence becomes fully developed at Re > 2500.

In the present work turbulent thermal and hydrodynamic behaviors of fluid flow in microchannel is investigated numerically using modified *k-epsilon* model. Different micro-geometries are adopted in this study such as microtubes, isothermal micro two fixed plates and micro Couette flow. The numerical procedure developed in the present work is based on the control volume approach proposed by Patankar [21]. Validation between numerical and experimental or analytical hydrodynamics and thermal behaviors is investigated.

2. MATHEMATICAL FORMUKATION

This study is particularly concerned with the hydrodynamic and thermal behaviors of micro turbulent, unsteady, Newtonian and incompressible flow of constant property. There are basically three flow cases considered in this study: fluid flow in micro-tubes, micro Coutte flow and micro two fixed plates. These are shown schematically in Fig. 1. In microchannel environment, the thermally turbulent flow is governed by the averaged-Reynolds form of the continuity, momentum and energy equations for unsteady, incompressible and Newtonian flow. Following the Reynolds averaging procedure [22], the unsteady Reynolds-averaged Navier-Stokes (RANS) equations coupled with the standard *k-epsilon* model can be written as follows:

$$\nabla \cdot (\rho \bar{u}) = 0 \quad (1) \qquad \frac{\partial(\rho \bar{u})}{\partial t} + \nabla \cdot (\rho \overline{uu}) + \nabla p = \nabla \cdot (2\mu \hat{S} + \hat{\Re}_t) \quad (2)$$

$$\frac{\partial(\rho T)}{\partial t} + \nabla \cdot (\rho \bar{u} T) = \nabla \cdot (\bar{q} / C_p) \quad (3)$$

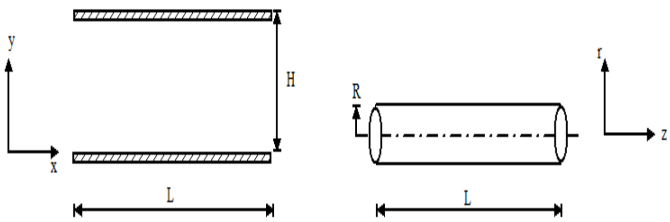
$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k \bar{u}) = \nabla \cdot (\mu + \mu_t / \Pr_k) \nabla k + 2\mu_t \hat{S}\hat{S} - \rho\varepsilon \quad (4)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \varepsilon \bar{u}) = \nabla \cdot (\mu + \mu_t / \Pr_\varepsilon) \nabla \varepsilon + (2C_{1\varepsilon} \mu_t \hat{S}\hat{S} - C_{2\varepsilon} \rho \varepsilon)(\varepsilon / k) \quad (5)$$

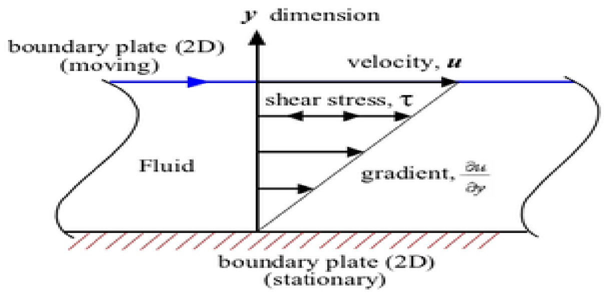
In the above system of equations, ρ is the density, $\bar{u}$ is the velocity vector, p is the static pressure, T is the temperature, $\bar{q}$ is the heat flux vector, $C_p$is the specific heat capacity at constant pressure, μ is the molecular viscosity, μ_t is the turbulent viscosity, k is the turbulent kinetic energy, ε is the turbulent dissipation, $\hat{S}$ and $\hat{\Re}_t$ are the strain rate tensor and turbulent stress tensor respectively. The components of the strain rate tensor can be described as:

$$S_{ij} = 0.5\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad (6)$$

and the turbulent stress tensor is given by:



(Case A): Two fixed plate Case (B): Micro tube



Case (C): Micro Couette flow

Fig. 1: Schematic diagram of the problems under consideration.

$$\Re_{ij} = -\rho \overline{u_i' u_j'} = -\frac{2}{3} \rho k \delta_{ij} + 2\mu_t S_{ij} \quad (7)$$

where δ_{ij} is the kronecker delta and $\overline{u_i' u_j'}$ is the average of the velocity fluctuations. The turbulent viscosity is defined as:

$$\mu_t = \rho C_\mu k^2 / \varepsilon \quad (8)$$

The coefficients for the modified turbulence model adopted in the present work are given as:

$$C_\mu = 0.01, \Pr_k = 1, \Pr_\varepsilon = 1.3, C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92$$

It should be pointed out that the above group of constants is similar to those developed by [23], except that for the value of $C\mu$. The heat flux vector consists of a molecular and turbulent part:

$$\bar{q} = -C_p\left(\frac{\mu}{\Pr} + \frac{\mu_t}{\Pr_t}\right) \nabla T \quad (9)$$

The turbulent Prandtl number is Pr_t= 0.9, while the molecular Prandtl number Pr and the specific heat capacity at constant pressure are defined according to the fluid considered.

3. NUMERICAL PROCEDURE

The numerical procedure is developed in the present work based on the control volume approach proposed by [21]. In such approach, a general differential equation for the dependent variables (u, v, T, k, ε) is written for unsteady, Newtonian, two-dimensional ($\Im = 0$)/axis symmetric ($\Im = 1$) and incompressible flow as follows:

$$\frac{\partial}{\partial t}(\rho\varphi) + \frac{1}{r^{\Im}}\frac{\partial}{\partial x}(\rho r^{\Im} u\varphi) + \frac{\partial}{\partial r}(\rho r^{\Im} v\varphi) = \frac{1}{r^{\Im}}\frac{\partial}{\partial x}(r^{\Im}\Gamma_{\varphi}\frac{\partial\varphi}{\partial x}) + \frac{\partial}{\partial r}(r^{\Im}\Gamma_{\varphi}\frac{\partial\varphi}{\partial r}) + S_{\varphi} \quad (10)$$

where φ is the dependent variable, Γ_{φ} is the diffusion coefficient for φ, and S_{φ} is the source term. The quantities Γ_{φ} and S_{φ} are specific to a particular meaning of φ. Using the control volume arrangement proposed by [21], the above general differential equation can be written in terms of the total fluxes over the control volume faces and the resulting equation is integrated over each control volume. In similar manner, the continuity equation is integrated over the control volume.

In our algorithm, one can assume that the velocity field reaches its final value in two stages; that means

$$\mathbf{u}^{n+1} = \mathbf{u}^{*} + \mathbf{u}_{c} \quad (11)$$

where, $\mathbf{u}^{*}$ is an imperfect velocity field based on a guessed pressure field, and $\mathbf{u}_{c}$ is the corresponding velocity correction. Firstly, the 'starred' velocity will result from the solution of the momentum equations. The second stage is the solution of *Poisson* equation for the pressure

$$\nabla^2 p_c = \frac{\rho}{\Delta t}\nabla\cdot\mathbf{u}^{*} \quad (12)$$

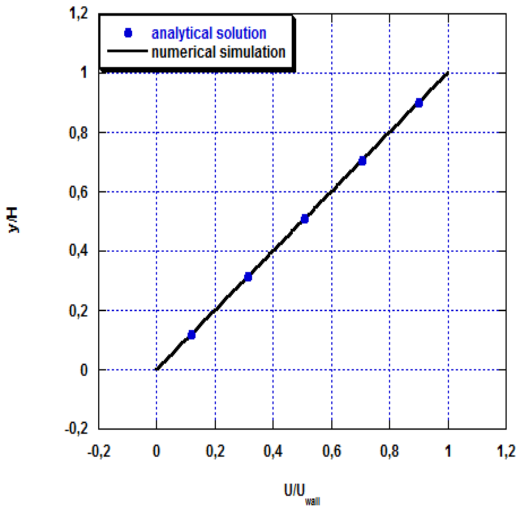
Where, p_c will be called the pressure correction and Δt is the chosen time step. Once this equation is solved, one gets the appropriate pressure correction, and consequently, the velocity correction is obtained according to:

$$\mathbf{u}_{c} = -\frac{\Delta t}{\rho}\nabla p_c \quad (13)$$

The fractional step non-iterative method described above ensures the proper velocity-pressure coupling for incompressible flow field. It should be pointed out that the above numerical method has been developed and successfully applied for simulating a variety of engineering applications [24-26].

4. Results and Discussion

The numerical procedure is developed in the present work based on the control volume approach proposed by Patankar. Validation between numerical and experimental or analytical hydrodynamics and thermal behaviors is investigated. Figure (2) shows the validation between the numerical and analytical hydrodynamics behavior for laminar micro Couette flow. It is clear from this figure that the numerical solution is identical with the analytical solution. Also, the validation between the numerical and experimental hydrodynamics behavior is shown in Figs. (3-5). It is clear from these figures that the numerical solution



gives the same predictions which are given by the experimental solution for laminar and turbulent flow [21].

Fig 2: Validation for laminar velocity distribution in micro Couette flow (Case C)

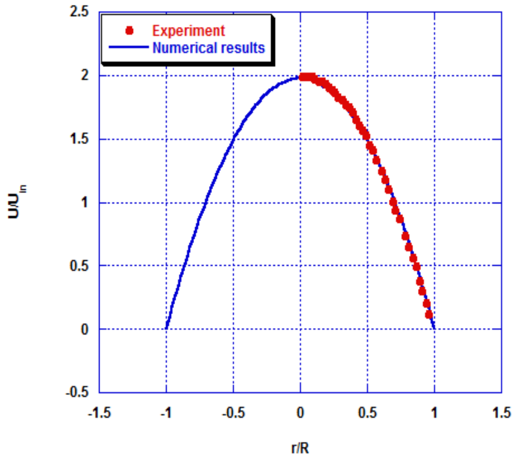


Fig. 3: validation for laminar velocity distribution in micro tube for Re=1587, Exp. Ref. [21], (Case B)

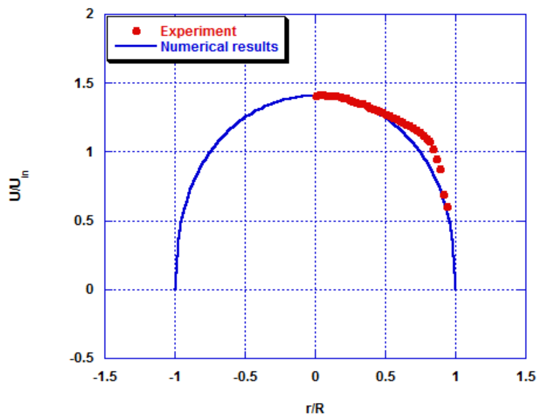


Fig. 4: validation for turbulent velocity distribution in micro tube for Re=2960, Exp. Ref. [21], (Case B)

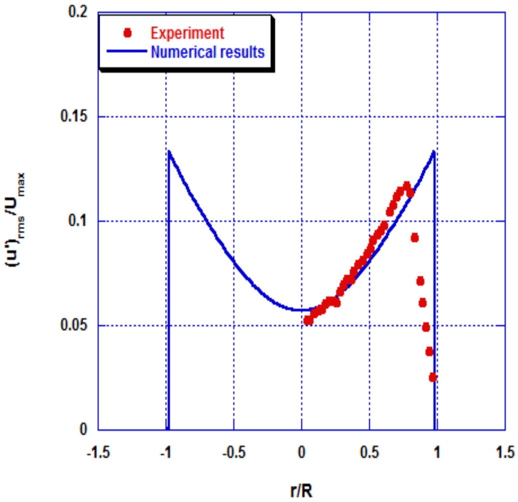


Fig. 5: validation for normalized streamwiseturbulent velocity fluctuation in micro tube, Re = 2960, Exp. Ref. [21], (Case B)

Figure (6) shows the thermal validation between numerical and analytical solution for microtube. It is obvious from this figure that the numerical solution matches the analytical solution.The spatial temperature and velocity distribution for laminar and turbulent flow for two micro fixed plates is shown in Figs. (7&8). It is clear from these figures that the maximum dimensionless velocity at the center of the channel for laminar and turbulent flow is 1.5 and 1.2 respectively. Also, the maximum temperature is 309.5 ^{o}C for laminar flow and 305 ^{o}C for turbulent flow.

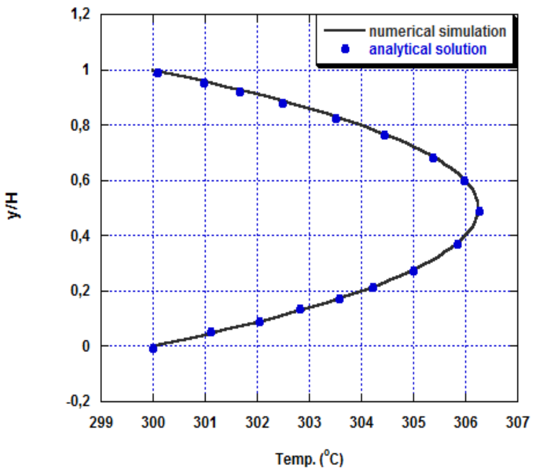


Fig. 6: Validation for temperature distribution for micro couette flow (Case B)

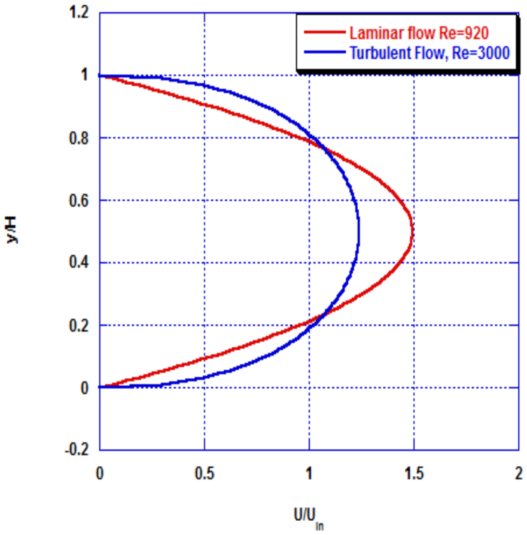


Fig. 7: Spatial dimensionless velocity distribution for laminar and turbulent two micro parallel fixed plate, (Case A)

Figure 9 shows the temperature distribution for laminar and turbulent micro tube flow. It is obvious from this figure that the maximum temperature at the center of the pipe is 309.5 ^{o}C for laminar flow, while it is 302 ^{o}C for turbulent flow.The dimensionless turbulent velocity distribution in micro tube at different Re is shown in Fig. 10.

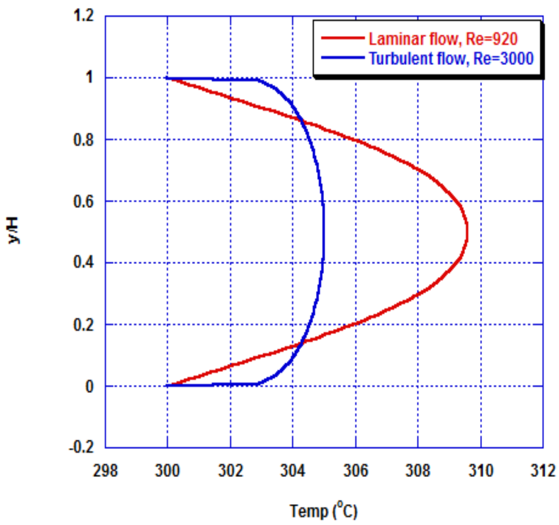


Fig. 8: Spatial temperature distribution for laminar and turbulent two micro fixed plate, (Case A)

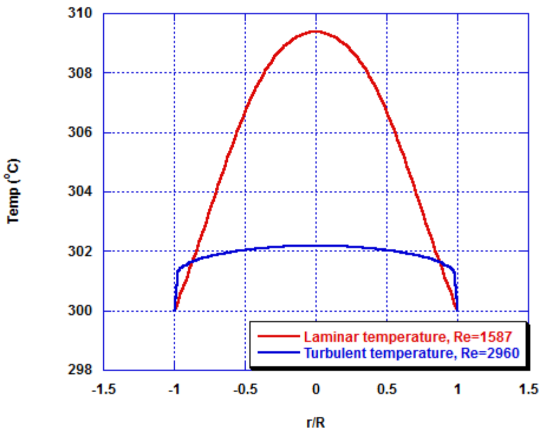


Fig. 9: Temperature distribution for laminar and turbulent micro tube flow, (Case B)

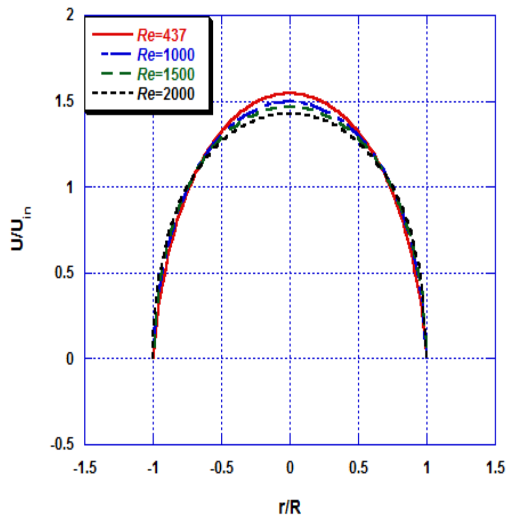


Fig. 10: Dimensionless turbulent velocity distribution in micro tube at different Re, (Case B)

5. CONCULSION

Turbulent thermal and hydrodynamic behaviors of fluid flow in microchannels are investigated numerically using modified*k-epsilon*model. Different micro-geometries are adopted in this study such as microtubes, isothermal micro two fixed plates and micro Couette flow. The numerical procedure is developed in the present work based on the control volume approach proposed by Patankar [21]. It is found that this numerical technique gives the same predictions which are given by the experimental or analytical.

REFERENCES

[1] Gad-el-Hak, Mohamed, MEMs Introduction and Fundamentals, 2ed ed. Taylor and Frances Group,LLC, 2006, ch. 4.

[2] G. Karniadakis, M. Beskok, Micro flows fundamentals and simulation. New York: Springer 2002.

[3] Y. Zohar, Heat convection in micro ducts. Kluwer 2003.

[4] M. A. Al-Nimr, and A. F. Khadrawi, Thermal behavior of a stagnant gas convicted in a horizontal microchannel as described by the dual-phase-lag heat conduction model, Int. J. Thermophysics, Vol. 25, pp. 1953, 2004.

[5] A. F. Khadrawi, A. Othman and M. A. Al-Nimr, Transient free convection fluid flow in a vertical microchannel as described by the hyperbolic heat conduction model, Int. J. Thermophysics, Vol. 26, pp.905, 2005.

[6] J. Al-Jarrah, A. F. Khadrawi, and M. A. Al-Nimr, Film condensation on a vertical micro-channel, Int. Communication in Heat and Mass Transfer, Vol. 35(9), pp. 1172-1176, 2008.

[7] M. A., Al-Nimr, A. M., Maqapleh, A. F. Khadrawi, and Ammourah S. A.: "Fully developed thermal behaviors for parallel flow microchannel heat exchanger", International Communications in Heat and Mass Transfer Vol. 36, pp 385–390, 2009.

[8] A. F. Khadrawi and Ahmad Al-Shyyab, Slip Flow and Heat Transfer in Axially Moving Micro-Concentric cylinders, International Communications in Heat and Mass Transfer, 37, 8, pp.1149–1152 (2010).

[9] M. Maqableh, A. F. Khadrawi, M. A. Al-Nimr, S. A. Ammourah and A. C. Benim, "Heat Transfer Characteristics of Parallel and Counter Flow Microchannel Heat Exchangers with Varying Wall Resistance" Progress in Computational Fluid Dynamics, 11(5), pp. 318-328 (2011).

[10] B.X. Wang, X.F. Peng, Experimental investigation on liquid forced convection heat transfer through microchannels, Int. J. Heat Mass Transfer 37 (Suppl. 1) (1994) 73–82.

[11] G.M. Mala, D. Li, Flow characteristics of water in microtubes, Int. J. Heat Fluid Flow 20 (1999) 142–148.

[12] J. Judy, D. Maynes, Characterization of frictional pressure drop for liquid flow through microchannels, Int. J. Heat Mass Transfer 45 (2002) 3477–3489.

[13] G.P. Celata, M. Cumo, G. Zummo, Thermal-hydraulic characteristics of single-phase flow in capillary pipes, Exp. Thermal Fluid Sci. 28 (2004) 87–95.

[14] G.P. Celata, M. Cumo, S. McPhail, G. Zummo, Characterization of fluid dynamic behaviour and channel wall effects in microtube, Int. J. Heat Fluid Flow 27 (2006) 135–143.

[15] M.J. Kohl, S.I. Abdel-Khalik, S.M. Jeter, D.L. Sadowski, An experimental investigation of microchannel flow with internal pressure measurements, Int. J. Heat Mass Transfer 48 (2005) 1518–1533.

[16] P. F Hao, F. He, K.Q. Zhu, Flow characteristics in a trapezoidal silicon microchannel, J. Micromech. Microeng. 15 (2005) 1362–1368.

[17] S.G. Kandlikar, D. Schmitt, Characterization of surface roughness effects on pressure drop in single-phase flow in minichannels, Phys. Fluids 17 (2005) 100606-1–100606-11.

[18] G.L. Morini, Scaling effects for liquid flows in microchannels, Heat Transfer Eng. 27 (2006) 64–73.

[19] K.V. Sharp, R.J. Adrian, Transition from laminar to turbulent flow in liquid filled microtubes, Exp. Fluids 36 (2004) 741–747.

[20] Peng-Fei Hao *, Xi-Wen Zhang, Zhao-Hui Yao, Feng He, Transitional and turbulent flow in a circular microtube, Experimental Thermal and Fluid Science 32 (2007) 423–431

[21] Patankar, S. V, 1980.Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation.

[22] Anderson, D. A., Tannehill, C. J., Pletcher, R. H., 1984. Computational Fluid Mechanics and Heat Transfer, Heimsphere Publishing Corporation.

[23] Launder, B. E., Spalding, D. B., 1974. The numerical computation of turbulent flow, Computer Methods in Applied Mechanics and Engineering 3(2), 269-289.

[24] Balabel, A., Numerical Prediction of Turbulent Thermocapillary Convection in superposed Fluid Layers with a free Interface, International Journal of Heat and Fluid Flow, 32(6), 1226-1239, 2011.

[25] Balabel, A., (2012). Numerical simulation of two-dimensional binary droplets collision outcomes using the level set method, International Journal of Computational Fluid Dynamics, vol. 26, no. 1, pp. 1-21.

[26] Balabel, A., (2012). Numerical Modelling of Turbulence Effects on Droplet Collision Dynamics using the Level Set Method. Computer Modeling in Engineering and Sciences (CMES), vol. 89, no.4, pp. 283-301.