Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2011
(ICMERE2011) 22- 24 December 2011, Chittagong, Bangladesh

ICMERE2011-PI-212

ANALYSIS OF A MICRO-CHANNEL HEAT EXCHANGER

Md. Arif Hasan Mamun[1], Md. Rakibul Hasan[1], Mohammad Jane Alam Khan[1], Shuichi Nagata[2], Toshiaki Setoguchi[2]

[1]Department of Mechanical Engineering, Bangladesh University of Engineering and Technology (BUET)
Dhaka-1000, Bangladesh
[2]Institute of Ocean Energy, Saga University
1 Honjo-machi, Saga 840-8502, Japan
E-mail: [1]arifhasan@me.buet.ac.bd, mahmamun1@gmail.com, [1]rakibhasan_490@hotmail.com,
[1]khan_me06@yahoo.com, [2]nagata@ioes.saga-u.ac.jp, [2]setoguci@me.saga-u.ac.jp

Abstract- *An efficient micro-channel heat Exchanger was designed which can be used for heat removal from the components requiring micro level heat extraction. The heat exchanger was designed and simulated numerically using the non symmetric pattern multi frontal finite element method for predicting the flow behavior in Micro channels. Three geometries were proposed and simulated in two dimensions to show that geometry directly affects the velocity distribution in the micro channel. The simulation demonstrated that the efficiency of the micro-heat exchanger vary with pressure difference, velocity distribution and heat transfer. The results were quite useful for the design of micro fluidic device for IC cooling.*

Keywords: CFD, Micro-heat exchanger, Micro-fluid

1. INTRODUCTION

It is quite obvious that over the past few decades and with the refinement of micro fabrication techniques, the use of micro-channels has become promising for a variety of industrial applications. Examples include biological applications for front end sample preparation (purification, separation, and concentration), cell sorting, and even more. Micro-channels continue to play an integral part in thermal management (microchip cooling, micro reactors) and energy systems (fuel cells, micro combustion) applications. Moreover, Energy conservation and sustainable development demands have been driving research efforts, within the scope of thermal engineering, towards more energy efficient equipments and processes. In this context, the scale reduction in mechanical fabrication has been permitting the miniaturization of thermal devices, such as in the case of micro-heat exchangers [1]. Recent review works [2,3] have pointed out discrepancies between experimental results and classical correlation predictions of heat transfer coefficients in micro-channels. Such deviations have been stimulating theoretical research efforts towards a better agreement between experiments and simulations, through the incorporation of different effects that are either typically present in micro-scale heat transfer or are effects that are normally disregarded at the macro-scale and might have been erroneously not accounted for in micro-channels.

Decreasing feature sizes and increasing package densities are making thermal issues extremely important in integrated circuit (IC) design. According to recent statistics, a large proportion of field failures can be attributed to overheating, which in turn is caused by inappropriate thermal design. Uneven thermal maps and “hot spots” in ICs cause physical stress and reduce reliability. High die temperatures affect circuit behavior in a number of adverse ways. For example, the mean-time-to-failure due to electro migration decreases exponentially with rise in temperature.

A number of techniques have been proposed in the literature for chip cooling. Initially Tuckerman and Pease [4] proposed micro channel heat sinks fabricated onto the back of a silicon chip substrate. Later on, other means were employed to cool the chips by: immersing the electronic chips in a pool of inert dielectric liquid [5], thermosyphons, where a liquid evaporates with applied heat and condenses dissipating that heat elsewhere, in a closed system [6], and heat pipes, where the liquid evaporates, condenses at another region and reaches the hot area through wick structures that line the heat pipes thus ensuring uniform distribution of heat [7]. These techniques are inherently unsuitable for compact, embedded systems. Thus, there remains a technology void and a pressing need for dynamically cooling hot spots in embedded ICs and compact packages used for portable applications.

A wide range of research has been carried out on different processes and application. For instance, Kang and Tseng [8] developed a theoretical model for predicting the thermal and fluid characteristics of a cross flow micro-heat exchanger. In micro-structured devices, homogeneity of the fluid is very important. Tomonomura et al. [9] designed a micro device using CFD. The simulation illustrates that the uniformity of flow greatly depends on the geometry of the shape of micro channels.

The present research effort was related to the fundamental analysis of flow behavior and forced convection within micro-channels without slip flow, as required for the design of micro-heat exchangers. Three different geometries were simulated using non symmetric pattern multi frontal finite element method and compared.

2. DESIGN AND SIMULATION

In this work, different micro-channel geometries were designed and analyzed but only three of them are discussed. These geometries were simulated and results were compared to identify the best channel configuration.

2.1 Mathematical Formulations

The basic mathematical equations in determining the flow behavior through the micro-channel are Navier-Stokes equation for momentum conservation and continuity equation for mass conservation.

$$\rho\frac{\partial u}{\partial t} + \rho u.\nabla u = -\nabla p + \mu\nabla^2 u - \rho g \qquad (1)$$

$$\nabla.u = 0 \qquad (2)$$

For getting temperature distribution, law of convection and conduction for energy conservation was used.

$$\nabla.(-k\nabla T) = Q - \rho C_p u.\nabla T \qquad (3)$$

2.2 Assumptions

In the present work, there were some assumptions made for simplifying the problem. The assumptions taken were:

a. Fluid flow (M=0.04) was incompressible though density varied with temperature.

b. Fluid was considered as continuum. In this case, Knudsen number (Kn) < 0.01 where Knudsen number is defined as:

$$Kn = \frac{\Lambda}{d_{min}} \qquad (4)$$

Here Λ is the mean free path [10] which is defined as:

$$\Lambda = \frac{RT}{\pi\sqrt{2}PA\sigma^2} \qquad (5)$$

c. The flow regime was laminar. So, Reynolds number should be less than 2300 for rectangular micro-channel [11]. Reynolds number (Re) is defined as:

$$Re = \frac{\rho V_m D_H}{\mu} \qquad (6)$$

2.3 Micro-channel Design

For simulation process various geometries were evaluated and the best three convenient ones are stated here. The basic difference among the three was in the inlet and outlet geometries. For Type-01, inlet and outlet accommodation of the flow were sidewise and geometries were triangular. For Type-02, the fluid inflow was in the central part and the distribution was triangular. In the third proposal (Type-03), the distribution is circular and central inflow. The Type-01 geometry is given below for detailed information:

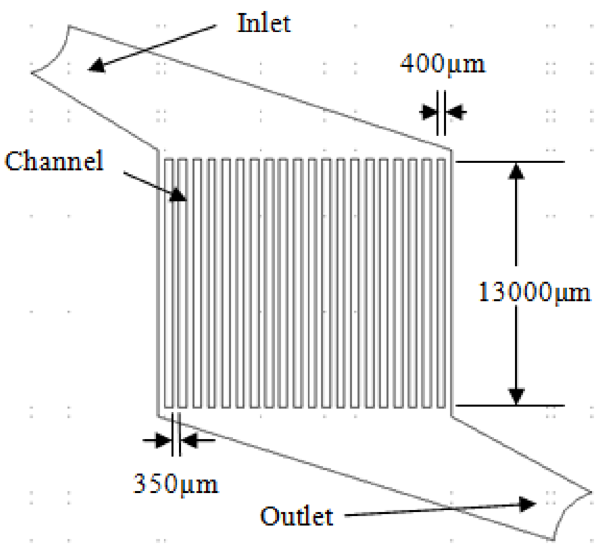

Figure 1: Type-01 geometry

Three micro-channels (Type 01, Type 02, and Type 03) were designed in COMSOL Multiphysics according to the design specifications of Table 1.

Table 1: Proposed Micro-channel dimensions

Characteristics			Types		
Symbol	Name	Unit	01	02	03
n	No. of channel	Dim-entionless	21	21	21
W_c	Channel width	µm	350	350	350
L	Channel length	µm	13000	13000	13000
W_a	Channel wall width	µm	400	400	400

2.4 Boundary Conditions

The boundary conditions as for fluid flow behavior taken for the specific problem were:

a. No slip flow at channel walls

b. Zero relative pressure at the outflow

c. Fully developed inflow velocity

For temperature distribution, the boundary conditions were as follows:

d. Constant inlet temperature

e. Constant channel wall temperature

f. Convective flux condition at outlet

2.5 Grid Independent Test

Grid independency test is important due to the complexity of the computational domain. Therefore, several grid size sensitivity tests were conducted on Type-03 to determine the sufficiency of the mesh scheme and to ensure that the solutions were grid independent. Five different non uniform grids of various numbers of elements were considered for the grid refinement test. It was found in Fig. 2 that 51264 non regular elements were sufficient to provide accurate results.

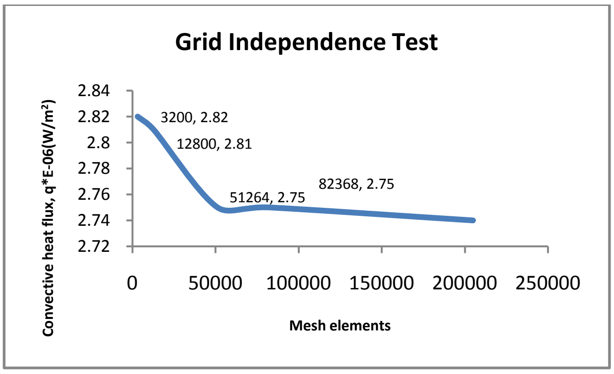

Figure 2: Grid refinement test for Type-03

2.6 Simulation

The proposed geometries were simulated under the mathematical model which is governed by the coupled equations of continuity, momentum and energy and was solved by employing non symmetric pattern multi frontal finite element method. For the simulation purpose, Nitrogen gas of inlet average velocity of 10 m/s was considered. The channel wall temperature was taken as constant 350 K and the inlet temperature of the gas was taken 300 K.

The physical problem was considered as damped Newtonian and highly non linear problem. The analysis was considered continuum and is divided in to a set of non-overlapping regions called elements. In addition, to discretize the physical domain triangular mesh elements were taken for velocity and temperature simulations. Substitutions of the proposed assumptions in to the system of governing equations and boundary conditions yield a very good result. More details about the solving method are available at Rahman et al.[12] as well as Zienkiewicz and Taylor [13].

2.7 Code Validation

The present results were validated based on the problem of Vasquez-Alvarez et al. [14]. Some of the present results were compared with those reported in Vasquez-Alvarez et al. [14]. The comparison of the average velocity has been shown in Table 2. The present results have a very good agreement with the results obtained by Vasquez-Alvarez et al. [14].

Table 2: Code validation for Type-01

Channel no	Average velocity(m/s)	
	present	reference
1	2.15	2.3
2	2.46	2.35
10	2.46	2.6
11	2.5	2.65
20	2.58	2.7
21	2.62	2.71

3. RESULTS AND DISCUSSIONS

The simulation results (velocity and temperature distribution) for all the proposed three geometries were as followings:

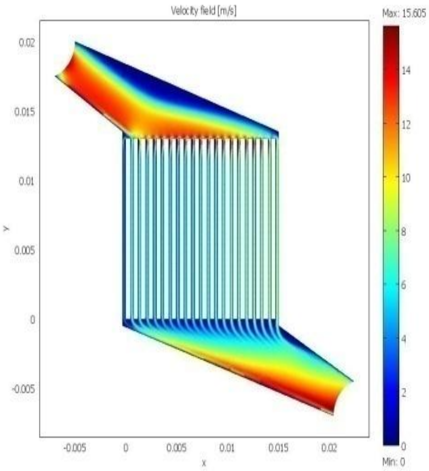

Figure 3: Velocity distribution for Type-01

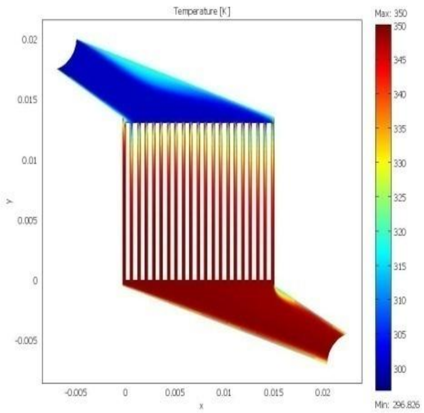

Figure 4: Temperature distribution for type-01

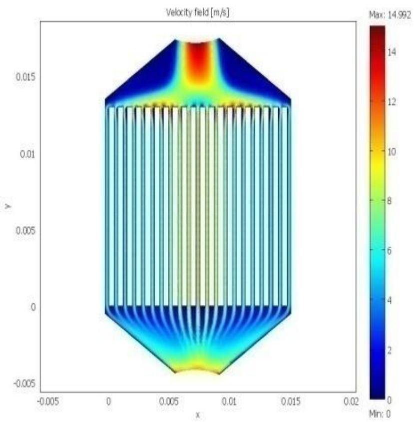

Figure 5: Velocity distribution for Type-02

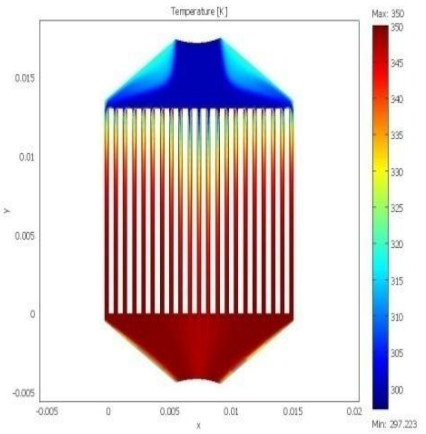

Figure 6: Temperature distribution for type-02

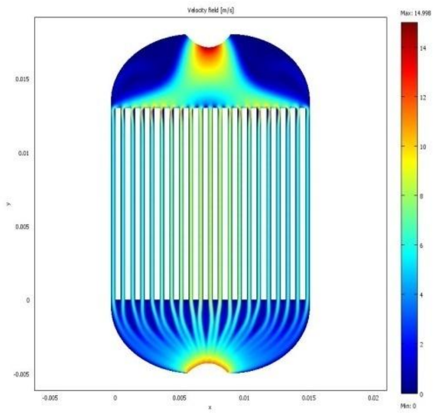

Figure 7: Velocity distribution for Type-03

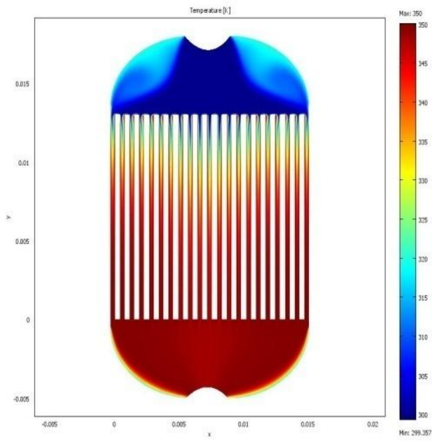

Figure 8: Temperature distribution for type-03

The various parameters obtained in different geometries are given in Table 3. From simulation data, in case of Type-01 geometry the maximum velocity was obtained 15.605 m/s. Reynolds number was 327.667 which ensures the laminar flow. Pressure difference was 300.11 Pa. For geometry Type 02 and Type-03 maximum velocity obtained was approximately same but Type-03 had lower Reynolds number which depicted good laminar flow. Moreover pressure difference between the inlet and outlet was the lowest. So, it will require less pumping effort. Velocity distribution among the channel was quite sound.

Table 3: Results

Param-eters	Unit	Types		
		01	02	03
V_{max}	m/s	15.605	14.992	14.991
Re_{max}	Dimensionless	327.667	274.576	158.25
ΔP	Pa	300.11	284.089	241.29
q_{max}	W/m^2	5.378E6	5.194E6	5.193E6

4. CONCLUSION

The numerical investigations of flow and thermal fields and heat transfer behaviors in a micro channel heat exchanger have been presented in this study. The basic differences in the results were because of the difference in inlet and outlet configurations of various geometries. After simulation process, velocity field, pressure difference, temperature distribution etc were analyzed for solutions of current field problems. Three geometries were proposed and Reynolds number, pressure difference and velocity, temperature distributions were obtained.

5. ACKNOWLEDGEMENT

We would like to pay gratitude towards Department of Mechanical Engineering, BUET for cordial cooperation in this study.

6. REFERENCES

[1] C.B. Sobhan and G.P. Peterson, *Microscale and Nanoscale Heat Transfer: Fundamentals and Engineering Applications*, CRC Press, Boca Raton, FL (2008)

[2] Yener, S. Kakaç, M. Avelino, and T. Okutucu, *Single-phase Forced Convection in Micro- channels –a State-of-the-art Review, in: S. Kakaç, L.L. Vasiliev, Y. Bayazitoglu, Y. Yener (Eds.), Microscale Heat Transfer Fundamentals and Applications*, NATO ASI Series, Kluwer Academic Publishers, The Netherlands, pp. 1–24 (2005).

[3] G.L. Morini, “Single-Phase Convective Heat Transfer in Micro channels: a Review of Experimental Results”, *Int. J. of Thermal Sciences*, 43, 631–651 (2004).

[4] D.B. Tuckerman and R.F.W. Pease, “High Performance heat sinking for VLSI,” *IEEE Electron Devices Letters*, vol. 2, 126-129, 1981.

[5] A.E. Bergles and A. Bar-Cohen, “Direct liquid cooling of microelectronic components,” *In Adv. in thermal modeling of electronic components and systems*, v. 2, 241-250, 1990.

[6] S.S. Murthy, et al., “Single chamber compact

thermosyphons with micro-fabricated components," *Intersociety Conf. Thermal and Thermomechanical Phenomena in Electronic Systems*, pp. 321-327, 2000.
[7] H. Xie et al., "The use of heat pipes in personal computers," *Intersociety Conf. Thermal and Thermomechanical Phenomena in Electronic Systems*, pp. 442-448, 1998.
[8] Kang, S-W., Tseng, S-C., "Analysis of effectiveness and pressure drop in a micro cross-flow heat exchanger." *Applied Thermal Engineering*, 27, 877-885 (2007).
[9] Tonomura, O., Tanaka, S., Noda, M., Kano, M.Hasebe, S., Hashimoto, I., "CFD-based optimal design of manifold in plate-fin microdevices." *Chemical Engineering Journal*,101, 397-402(2004)
[10] Commenge, J-M., Falk, L., Corriou, J-P., Matlosz, M., "Optimal Design for flow uniformity in Microchannel Reactors." *AICHE Journal*, 48, No 2, 345-358 (2002).
[11] Kockmann, N., "Transport Phenomena in Micro Process Engineering." Springer, Germany (2008).
[12] M.M. Rahman, M.A. Alim, M.A.H. Mamun, "Finite element analysis of mixed convection in a rectangular cavity with a heat-conducting horizontal circular cylinder," *Non linear analysis, Model. Control* 14 (2) (2009) 217-247.
[13] O.C. Zienkaewicz, R.L. Taylor, *The finite element method*, fourth ed, McGraw-Hill, 1991.
[14] E. Vasquez-Alvarez, F.T. Degasperi,L.G. Morita,M.R. Gongora-Rubio and R. Giudici, "Development of a micro-heat exchanger with stacked plates using LTCC technology," *Brazilian Journal of Chemical Engineering*, Vol.27 No.3 pp. 483-497.

7. NOMENCLATURE

Symbol	Meaning	Unit
A	Avogadro number	mol^{-1}
$\mathbf{C_p}$	Heat capacity	J/Kg.K
$\mathbf{D_H}$	Hydraulic diameter	m
$\mathbf{d_{min}}$	Minimal characteristic dimension	m
g	gravitational acceleration	m/s^2
Kn	Knudsen number	Dimensionless
k	Thermal conductivity	W/m.K
L	Channel length	μm
M	Mach number	Dimensionless
n	Channel number	Dimensionless
P	minimum operating pressure	Pa
ΔP	Pressure difference	Pa
Q	Heat source	W/m^3
$\mathbf{q_{max}}$	Maximum convective heat flux	W/m^2
R	universal gas constant	J/mol.K
Re	Reynolds number	Dimensionless
$\mathbf{Re_{max}}$	Maximum Reynolds number	Dimensionless
T	Temperature	K
u	fluid velocity	m/s
$\mathbf{V_m}$	average flow velocity used to calculate the Reynolds	m/s
$\mathbf{V_{max}}$	maximum velocity	m/s
$\mathbf{W_a}$	Channel wall width	μm
$\mathbf{W_c}$	Channel width	μm
μ	Dynamic viscosity	Pa.s
Λ	Molecular mean free path	M
ρ	Density	Kg/m^3
σ	Molecular diameter	m