Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 75 (2015) 912 – 918

Energy Procedia

The 7th International Conference on Applied Energy – ICAE2015

Investigation of the enhancement effect of heat transfer using micro channel

Liang Xia*, Yue Chan+

*Department of Building Services Engineering and Building Physics, Faculty of Science
and Engineering, The University of Nottingham Ningbo China, Ningbo, China
+School of Mathematical Sciences, Faculty of Science and Engineering,
The University of Nottingham Ningbo China, Ningbo, China

Abstract

This paper reports an investigation of the enhanced heat transfer effect in micro sized channels. In particular, a simulation model has been established using computational fluid dynamics (CFD) platform, i.e., Fluent to investigate such micro effect. Here, hot fluid flow is squeezed into the micro channel and the heat is gradually transferred into the cold surface boundaries, i.e., top and bottom surfaces of the channel having rectangular cross-sections. The area of inlet is varied to investigate and identify the enhanced heat absorption arising from the size effect and it is found that heat transfer rate per unit effective heat transfer area is greater if the inlet area becomes smaller. Lower pressure drop can also be deduced if smaller radii channels are considered. Least but not last, a new design of heat exchanger is proposed based on these extra micro-effects.

Keywords:micro channel heat transfer, heat transfer enhancement, reduction in pressure loss, micro channel heat exchanger

1. Introduction

Due to the increasing demands in micro manufacture, micro channel heat transfer is getting more attentions. It also enjoys advantages in the weight reduction and the improvement of the compactness of equipment, micro-channel heat exchanger takes an important role in heat transfer augmentation, micro-electronics and micro-electromechanical systems (MEMS), miniaturized chemical reactors and

---

* Corresponding author. Tel.: +86-0574-88189839; fax: +86-0574-88180188.
E-mail address: liang.xia@nottingham.edu.cn

doi:10.1016/j.egypro.2015.07.234

combustors, aerospace, and biomedical systems. Since the concept of micro-channel heat exchanger was first proposed and used by Tuckerman and Pease [1] in 1981, the performance of microscopic heat exchanger continuously gains researchers' interests.

Compared with the conventional heat exchanger, micro-channel heat exchanger reveals distinctive flow characteristics and heat transfer characteristics due to its structural and physical effects. In particular, some phenomena and new laws emerge in micro channel such as enhanced fluid compressibility effects, which can be simpely attributed to scale effects [2,]. As surface area to volume ratio increases, the mechanical effects associated with the area (surface, forces, viscous forces etc) in the micro channel will be strengthened [3], and the impact of axial heat conduction on micro channel wall is also enhanced [4-6].

Heat transfer enhancement is observed in micro channel as well. Dirker et al. [7] found experimentally the inlet type, i.e., the shape of inlet, significantly influenced the heat transfer coefficients and friction factors in laminar and transitional flow regimes. Li et al. [8] numerically studied air side performance, i.e., Colburn *j* factor of an integrated fin and micro-channel heat exchanger in different fin geometrics (fin height, pitch, depth and thickness).

This work focuses on the computational study of heat transfer effect in micro channels. A commercial computational fluid dynamics (CFD) package, i.e. Fluent, is used in simulating heat transfer between hot fluid and the cold channel inner surfaces in various inlet areas. Pressure drop is also studied in this case. Based on the simulation results, a new design of regular sized heat exchanger embedding micro-channel bundles is proposed.

Nomenclature

x, y, z	Cartesian coordinate system, m
k	turbulent kinetic energy, m^2/s^2
ε	Turbulent dissipation rate, m^2/s^3

2. Simulated model

Geometrical model of micro channel

In order to simulate the heat transfer taking place in it, micro channel model is generated using Design Modeler in ANSYS Fluent. Figure 1a shows the model geometry while Figure 1b shows the grids which possess cuboid shape with the same volume and Figure 1c shows the geometry set-up of model. As shown in Fig.1c, inlet and outlet are the two terminating surfaces in z direction. Other four surfaces which are perpendicular to the inlet are the walls of the channel. Therefore, the walls perpendicular to the y axis are the top and bottom walls, and the ones perpendicular to the x axis are left and right walls, respectively.

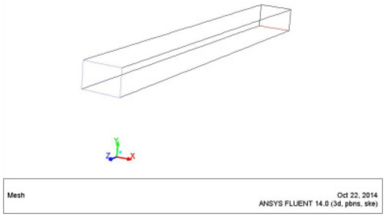

Figure 1a Model geometry of the micro channel

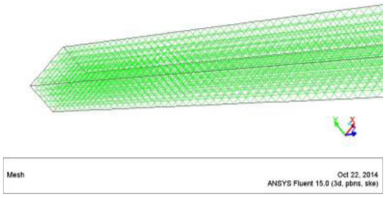

Figure 1b Grids of the micro channel

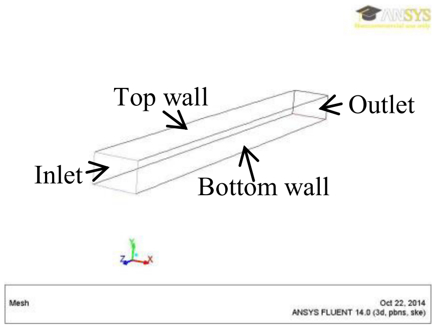

Figure 1c Geometry set-up of the channel

To investigate the size effect of micro channel on heat transfer, the cross-sectional area of the channel is varied. Table 1 shows the variation of length (in x direction as shown in Fig.1a) and width (in y direction as shown in Fig.1b), the variation of area, and the ratio of heat transfer area to volume of channel. It can be seen that the ratio of heat transfer area to volume decreases as the inlet area increases. It should be mentioned the number of grids is proportionally increased if the area increases.

Table 1.Variation of the cross-sectional area of micro channel

No.	Length (μm)	Width (μm)	Area (μm^2)	Ratio of heat transfer area to volume (μm^{-1})
1	3	2	6	0.5
2	6	4	24	0.25
3	12	8	96	0.125
4	18	12	216	0.0833
5	24	16	384	0.0625
6	30	20	600	0.05

Boundary Conditions and calculation models used in simulation

Hot air flows from the inlet towards the outlet and exchanges its heat with the cold top and bottom walls. Boundary conditions which are shown in Table 2 are applied in the simulation. In this simulation, the continuity, momentum and energy conservation equations [9] are solved. Standard *k*-*ε* model [9] is also used to modeling the momentum transport.

Table 2.Boundary conditions

Boundary	Conditions
Inlet	T: 400K, v:0.001m/s
Outlet	Outflow
Top and bottom walls	Non-slippery walls and surface temperature: 300K
Right and left walls	Non-slippery and adiabatic walls

3. Simulation results

In this section, simulation results for micro channels with different cross-sectional areas are presented. Figure 2, where the unit of x and y axis is m and K, respectively, shows the distribution of air temperature in terms of middle line for different size channels. It can be found that the air temperature decreases from inlet to outlet in different speeds. The inlet is at −0.00003m and the outlet is at 0m. The speed drop is the highest when the inlet area is the smallest, i.e., $3{\times}2\mu m^2$, and it decreases with an increase of the inlet area.

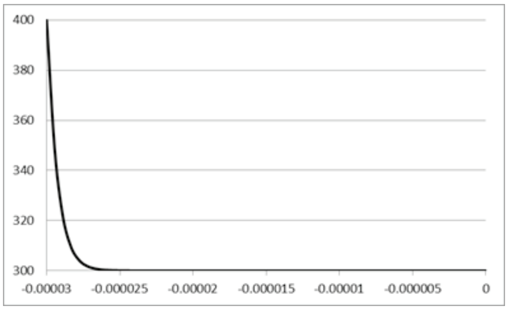

Figure 2a.Temperature distribution on middle line for inlet area of $3\times2\ \mu m^2$

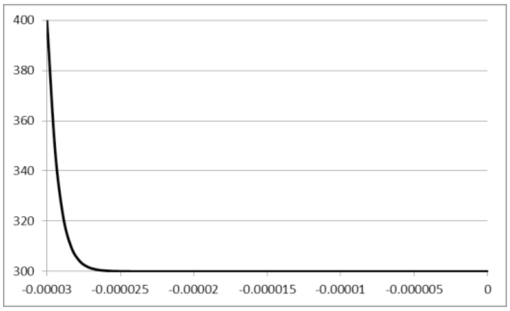

Figure 2b.Temperature distribution on middle line for inlet area of $6\times4\ \mu m^2$

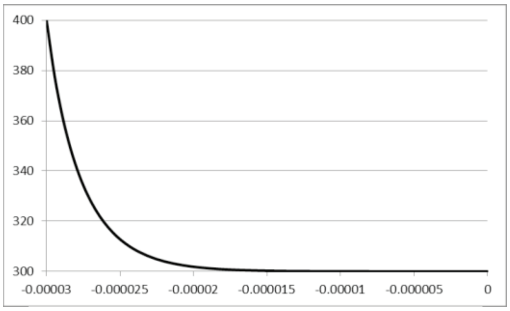

Figure 2c.Temperature distribution on middle line for inlet area of $12\times8\ \mu m^2$

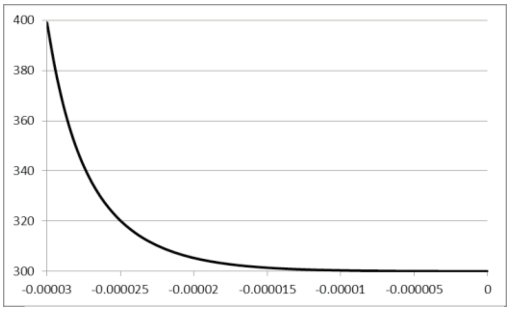

Figure 2d.Temperature distribution on middle line for inlet area of $18\times12\ \mu m^2$

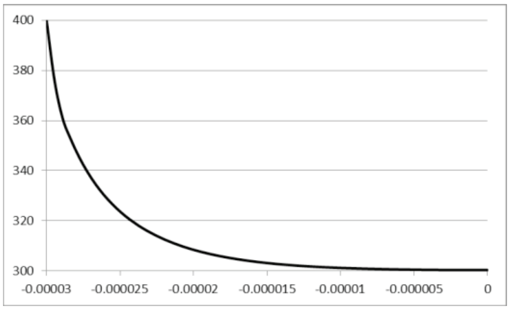

Figure 2e.Temperature distribution on middle line for inlet area of $24\times16\ \mu m^2$

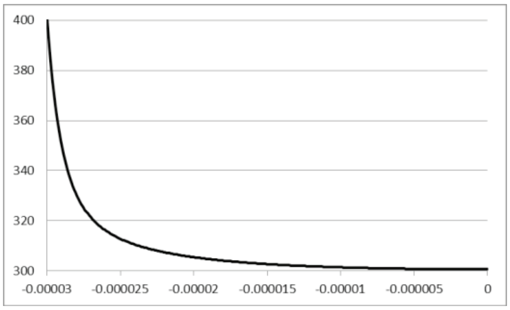

Figure 2f.Temperature distribution on middle line for inlet area of $30\times20\ \mu m^2$

Higher rate in the temperature drop illustrates the stronger heat transfer ability owing to the fact that the heat is dissipated more quickly. Therefore, we find that the heat transfer can be strengthened as the inlet area of the channel is decreased, i.e., the ratio of heat transfer area to volume is increased. This result indicates the size effect on heat transfer varies as the ratio varies. Enhancement of heat transfer can be obtained by increasing the ratio, which has been validated by previous relative researches. For example, Li and Guo [10] concluded the higher Nusselt number for a small scale and Xie et al. [11] reported a lower thermal resistance in microchannel of smaller width.

In Figure 2f, the air temperature decays to 300K at the vicinity of outlet. We can conclude that the air temperature remains higher than 300K if the inlet area is further increased. However, for the smaller sized channel, air temperature decreased to 300K occurring far away from the outlet. The location in channel where air temperature decreases to 300K is different when the inlet area of channel is different. Upon considering the negligible heat transfer as air temperature is lower than 301K, the side area of channel where the bulk air temperature is higher than 301K is defined as effective heat transfer area, and the corresponding flow length is defined as effective channel length. Such that result shown in Fig.2 illustrates that the effective heat transfer area is smaller as the inlet area decreases. For different inlet areas, the heat transfer rate per unit effective heat transfer area is also calculated. Table 3 illustrates the ratio of effective heat transfer area to the total side area, and the heat transfer rate per unit effective heat transfer area.

Table 3.Distance ratio and heat transfer rate per unit effective heat transfer area

No.	Area (μm^2)	ratio	Heat transfer rate (W/m^2)
1	6	0.268	496376
2	24	0.289	486207
3	96	0.393	405381
4	216	0.565	284998
5	384	0.711	227364
6	600	0.875	187118

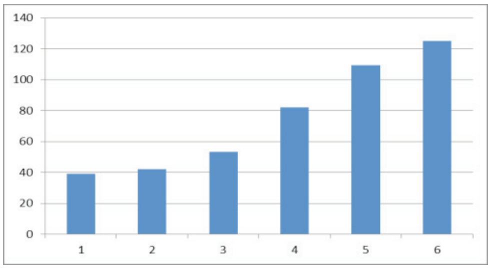

Figure 3.Pressure drop along the channel

4. Heat exchanger shape optimization

From Fig.2 and Table 3, we find the benefit of using micro channel having smaller inlet area, which has more intensive heat transfer on effective heat transfer area. Shorter channel can be applied, for example the total length of the channel can be reduced to 26.8% from its original total length to achieve the same amount of heat transfer rate for a channel of smaller inlet area. The total pressure loss during the flow can be also decreased in shorter channel. These reveal advantages for choosing micro channel of smaller inlet area. Assuming a constant inlet pressure, the pressure loss along the middle line as air flows within the effective length is calculated for different channels and shown in Fig.3, where the unit of pressure loss (y axis) is Pa. This figure shows that the pressure loss is the lowest for the channel of the smallest inlet area.

Current heat exchanger is made of fin-tube design and the tube used is in regular size, not micro one. Two benefits are the enhancement of heat transfer and the reduction of pressure loss of using micro channel with small inlet area are shown and discussed in Section 3. Based on these two benefits, we now propose a more efficient design of heat exchanger. For improving the heat transfer efficiency and reducing the pressure loss of fluid flowing through the channels, we can design a tube numerous micro embedding micro-sized channels, which are shown in Fig.4. Higher energy efficiency and lower pressure loss result from using this design. However, the effect of heat transfer enhancement and the cost for manufacturing such structure have to be further investigated.

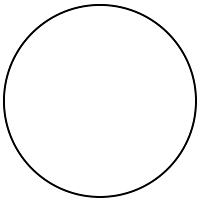

Figure 4a.Regular size tube in heat exchanger

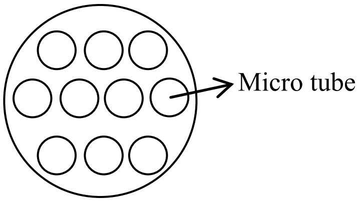

Figure 4b.Regular size tube including numerous micro tubes in heat exchanger

5. Conclusions and discussion

Enhancement of heat transfer performance of a micro-sized channel has been investigated in this paper. Using a CFD simulation, we find that the temperature of bulk flow in channel responses more quickly when the inlet area of channel becomes smaller resulting in the stronger heat transfer ability for micro-sized channels. The heat transfer rate per unit effective heat transfer area provides another path to prove the enhancement in the heat transfer using small radii channel. The reduction of pressure drop using small radii channel is also demonstrated based on the simulation results of pressure loss. A new design of heat exchanger based on the above effects is proposed accordingly.

Acknowledgement

We gratefully acknowledge the financial support from the small research grant, UNNC (01.03.02.01.2052).

6. References

[1] Tuckerman DB, Pease RFW. High-performance heat sinkingfor VLSI. *IEEE Electron Device Lett.* 1981;5:126-9.

[2] Du XD. *Affect of compressibility and roughness on the micro-pipe flow and heat transfer*, PhD thesis, Tsinghua University,2000.

[3] Guo ZY. Characteristics of microscale fluid flow and heat transfer MEMS. *Proceeding of the international conference on Heat and Transport phenomena in microscale*, Banff Canada ,2004:24~31.

[4] Li ZX, WangW, Guo ZY. Effects of axial heat conduction in wall on convection inmicrotubes.Conf.on Microchannels and Minichannels 2003;24:327~333.

[5] Sakakibara MS. Steady heat transfer to laminar flow in a circular tube with conduction in tube wall. *Heat Transfer-Jpn.Res*. 1974;3(2):27-46.

[6] Shah RK, London A. *Laminar flow forced conduction in ducts*. Advances in Heat Transfer ,Acadernic Press,New York,San Francisco,London,1978.

[7] Dirker J, Meyer JP, Garach DV. Inlet flow effects in micro-channels in the laminar and transitional regimes on single-phase heat transfer coefficients and friction factors. *International Journal of Heat and Mass Transfer* 2014;77:612-626.

[8] Li J, Wang, SF, Wang C, Zhang WJ. Numerical study on air-side performance of an integrated fin and micro-channel heat exchanger. *Applied Thermal Engineering* 2010;30:2738-2745.

[9] ANSYS Help Document. *ANSYS Fluent* 15.0 2013.

[10] Li ZX, Guo ZY. Size effect on microscale single-phase flow and heat transfer *International Journal of Heat and Mass Transfer* 2003;46:149~159.

[11] Xie XL, Liu ZJ, He YL, Tao WQ. Numerical study of laminar heat transfer and pressure drop characteristics in a water-cooled minichannel heat sink. *Applied Thermal Engineering* 2009;29:64-74.