Al-Nahrain University, College of Engineering Journal (NUCEJ) Vol.19 No.1, 2016 pp.66 - 75

A Numerical Study of the Heat Transfer and Fluid Flow in Different Shapes of Microchannels

Ammar A. Abed
Mechanical Eng. Dep.
Coll. of Engineering, Al-Anbar University
ammar.alhete22@yahoo.com

Wissam H. Khalil
Mechanical Eng. Dep.
Coll. of Engineering, Al-Anbar University

Abstract

Microchannels are the current interest for use in compact heat exchanger , micro reactors , very large scale integrated system where there is a desire of high heat transfer performance. The mentioned electronic equipment are important part of modern life. The dissipation of heat from these equipment is very necessary for the proper functioning of these instruments. Microchannels provide high heat transfer coefficients because of their small hydraulic diameter. In this work, a numerical investigation of fluid flow and heat transfer in different shapes of microchannels have been presented, square notches with different number are added to the rectangular microchannel to create different shapes of MCHS. A three dimensional computational fluid dynamics were built using a commercial package FLUENT to investigate the conjugate fluid flow and heat transfer phenomena in aluminum-based rectangular microchannel heat sink. The MCHS performance is evaluated in terms of temperature profile , Nusselt number and friction factor. Water is used in the present study as the working fluid. The results show that the heat transfer rate and Nusselt number are increased for the shapes that have notches compared with the original channel as Reynolds number increase. The Results conducted from this study were compared with that published in the previous literature and there were a good agreement obtained.

Introduction

One of the most important issues in the electronic industry is how to dissipate greater amount of heat generated by electronic devices to ensure a high performance for these devices. **Tuckerman and Pease [1]** introduced the concept of microchannel heat sink. They demonstrated that the laminar flow in a rectangular microchannel has higher heat extraction capabilities than turbulent flow in conventional size tubes. This finding opened up a new research field and it has been followed by many more studies by numerous researchers. **Fedorov and Viskanta[2]**developed a three dimensional model to investigate the conjugate heat transfer in a micro channel heat sink. This investigation indicated that the average channel wall temperature along the flow direction was nearly uniform except in the region close to the channel inlet, where very large temperature gradients were observed. This allowed them to conclude that the thermo–properties are temperature dependent. The modifications of thermo-physical properties in the numerical process are very difficult as the temperature and velocity are highly coupled. **Rahman[3]**measured the heat transfer coefficient and pressure drop of water flowing through multiple rectangular microchannels of hydraulic diameter 299-491 µm. The larger heat transfer was believed to be caused by the breakage of the velocity boundary layer adjacent to the wall by surface roughness. No sharp transition from laminar to turbulent flow was ever observed with Reynolds numbers up to 3250. **Leleaet *al*. [4]** performed experiments on laminar (Re<800) heat transfer and fluid flow using water in three microtubes of 0.1, 0.3 and 0.5 mm in diameter. The experimental results confirmed that, including the entrance effects, the conventional or classical theories are applicable for water flow through microchannels of the above sizes. **Qu and Mudawar[5]**studied the critical heat flux in a water-cooled microchannel heat sink consisting of 21 parallel215 x 821 µm channels. They observed that CHF was independent on the inlet temperature but it increased with increasing the mass flux. The yalso noticed that, as CHF was approached, flow instabilities induced backflow into the heat sink's upstream plenum .**Kuan and Kandlikar[6]**experimentally investigated the critical heat flux of water in six parallel 1054 x 157 µm microchannels. Their results show that CHF increases with mass flux but decreases with increasing exit vapor fraction. Through simultaneous flow visualization, they could observe a change in flow pattern at CHF, from annular flow to liquid streams travelling in the core of the flow. **Liu and Garimella[7]** and **Xu et al. [8]**confirmed in their studies that the behavior of heat transfer and fluid flow through microchannels is quite similar to that of conventional channels .**Hetsroni et al**. **[9]**has verified the capacity of conventional theory to predict the hydrodynamic characteristics of laminar Newtonian incompressible flows in micro channels in the range of hydraulic diameter from D_h = 15to D_h= 4010 µm. They have compared their results with the data available in open literature. **Sabbah et al. [10]** observed that the prediction of heat transfer in micro-channels becomes difficult with increase in complicacy of the geometry of the microchannels. **Allen [11]** had investigated fluid flow and heat transfer in microchannels experimentally and numerically. Fluid flow and heat transfer experiments were conducted on a copper microchannel heat exchanger with constant surface temperature.

Very recently, **Mathew and Hegab[12]**theoretically analyzed the thermal performance of parallel flow micro heat exchanger subjected to constant external heat transfer, The results show that for the same volume of heat exchanger, increasing the number of channels leads to an increase in both effectiveness and pressure drop. The present work will take in to account different shapes of notches inside the microchannel and study their effect on the behavior of performance of microchannel.

Nomenclature

A	channel flow area, m^2
Cp	specific heat, J/kg K
D_h	hydraulic diameter, µm (D_h=2HW/(H+W)
f	friction factor, (f=2D_h ΔP/ρ $u_{in}^2 L_{ch}$)
Nu	Nusselt number (Nu=ρ u D_h /µ)
H	channel height of rectangular, µm
W	channel width of rectangular, µm
k	thermal conductivity, W/m.K
L_c	channel length, µm
P	channel wet perimeter, µm
q_w	heat flux at microchannel heat sink top plate, W/m^2
Re	Reynolds number, (Re=ρ $D_h u_{in}$/µ)
S	distance between two microchannels, µm
t	substrate thickness, µm
T_{in}	fluid inlet temperature, K
T_s	microchannel heat sink solid temperature, K
w	fluid velocity, m/s
w_{in}	inlet fluid velocity, m/s
U	dimensionless velocity in x-coordinate
V	dimensionless velocity in y-coordinate
W	dimensionless velocity in z-coordinate
X, Y, Z	dimensionless Cartesian coordinates

Greek symbols

µ	viscosity, kg/m s
ρ	density, kg/m^3

Subscripts

avg	average
ch	channel
h	hydraulic
i	inlet
o	outlet
s	solid
str	straight

Model Geometry

The microchannel heat sink used in this research is schematically described in Figure (1) and (2). The unvaried total area being cooled is (W *L_z) with individual minichannel flow passage dimensions of w, h. The wall separating the two channels is of thickness W_w and acts like a fin. The bottom plate thickness is S_b, and the coolant flow velocity U_{in} are the parameters of interest in designing a microchannel heat sink .Heat supplied to the aluminum MCHS substrate through a bottom plate, is removed by flowing water through a microchannels. the dimensions of all channel shapes and rectangular cross-section are given in (Table-1).In this study, the effect of using different channel shapes on heat transfer the change in shape is by add notches along of rectangular microchannel, the number of notches is 2,4 and 6 for two side (The dimension in Table-2) ,and fluid flow characteristics of MCHS is considered

Table1: dimension for rectangular cross-section channel

W	W	H	h	t	S_t	S_b	L_z	D_h
340µm	280µ	680µm	480µm	30µm	100µm	100µm	10000µm	355µm

Table2: dimension for rectangular with notches

W	W	H	h	t	S_t	S_b	w_{not}	h_{not}
340µm	280µ	680µm	480µm	30µm	100µm	100µm	60µm	60µm

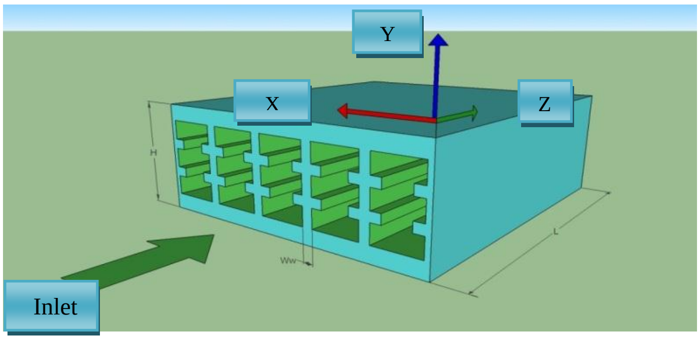
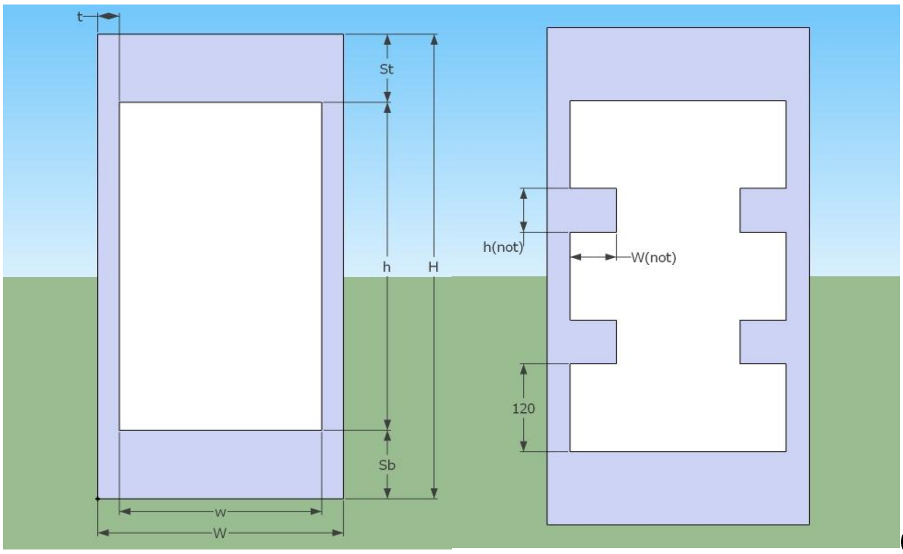
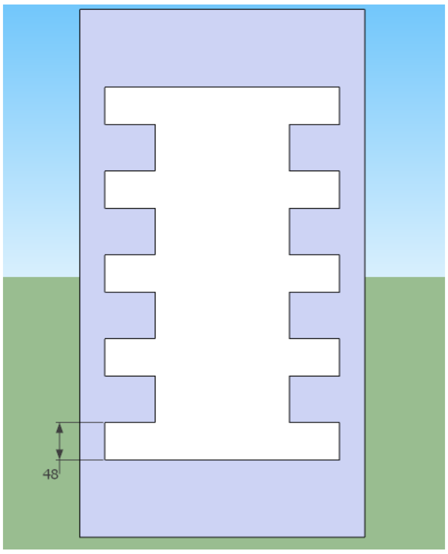


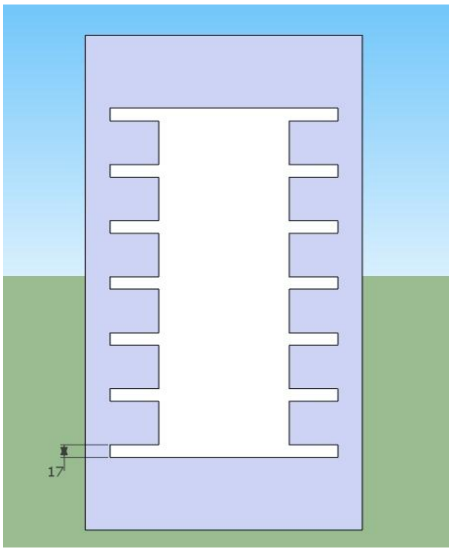
Figure 1: Schematic diagram of the microchannels heat sink



(a)straight



rectangular(b) Microchannel with two-Notches



(c) Microchannel with four-Notches (d) Microchannel with six-Notches

Figure 2: straight and with notches microchannels

Mathematical Formulation and Numerical Methods

In order to solve the Navier–Stokes and energy equations to investigate the effect of channel shape on the MCHS performance, the following assumptions were made: **(i)** both fluid flow and heat transfer are in steady state and three dimensional; **(ii)** fluid is in single phase, incompressible and the flow is laminar; **(iii)** properties of both fluid and heat sink material are temperature-independent; and **(iv)** all the surfaces of heat sink that exposed to the surroundings are assumed tobe insulated except the bottom plate of heat sink where a constant heat flux boundary condition that simulating the heat generation from external sources is specified.

At the channel inlet, the water temperature is equal to a given constant inlet temperature of T_{in}= 293 K. The outflow condition is assumed where there is no gradient in temperature, velocity and pressure gradient is constant.

The governing equations and its boundary conditions in Cartesian coordinates for 3D laminar incompressible flow for the current problem are **[13]**:

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad \text{…..}\ (1)$$

X−Momentum equation:

$$\rho_w\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu_w\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \quad \text{…..}\ (2a)$$

Y−Momentum equation:

$$\rho_w\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu_w\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \quad \text{…..}\ (2b)$$

Z−Momentum equation:

$$\rho_w\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu_w\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \quad \text{…..}\ (2c)$$

Energy equation:

$$\rho C_p\frac{DT}{Dt}=k\nabla^2 T \quad \text{…..}\ (3)$$

In order to facilitate the investigation of channel geometry effects and channel thermal conductivity effect, the numerical simulation was conducted for the entire computational domain, with the solid region being treated as a special liquid. This implies that the computation is of conjugated type **.** The no-slip hydraulic boundary condition of velocity is adopted for the solid wall, the inlet distribution is uniform for velocity at the channel inlet and the outlet boundary condition is considered of local one-way type **[13,14]**:

$$Z = 0,\ w = W_{in},\ u = \upsilon = 0 \quad \text{….}\ (4)$$

Z = L, the influence coefficient of the downstream equals zero. ….. (5)

$$u = 0,\ \upsilon = 0,\ w = 0,\ \text{at solid walls} \quad \text{…..}\ (6)$$

The velocities in the solid region are everywhere zero which is automatically guaranteed by numerical solution algorithm for conjugated problem **[13,14]**.

The thermal boundary conditions are given as follows. The left and right surfaces are the symmetry planes, and the boundary conditions are adiabatic:

$$X = 0,\quad \frac{\partial T}{\partial x}=0 \quad \text{…..}\ (7)$$

$$X = W_c + W_w,\quad \frac{\partial T}{\partial x}=0 \quad \text{…..}\ (8)$$

At the bottom position, the heat flux is given:

$$Y = 0,\quad -k\frac{\partial T}{\partial x}=qw \quad \text{…..}\ (9)$$

The top surface is assumed to be adiabatic:

$$Y = H_b + H_c,\quad \frac{\partial T}{\partial y}=0 \quad \text{…..}\ (10)$$

At the inlet position, the inlet temperature of liquid isgiven to be constant, and the outlet boundary is consideredof local one-way type **[14,15]**:

$$Z = 0,\ T = T_{in} \quad \text{…..}\ (11)$$

Z = L, the influence coefficient of the downstream equals zero ….. (12)

A computational fluid dynamic code is used to calculate flow velocity, pressure, and temperature in the channels of a MCHS. Finitevolume method (FVM), FLUENT was used to convert the governing equations to algebraic equations accomplished using hybrid differencing scheme **[13]**. The SIMPLE algorithm was used to enforce mass conservation and to obtain pressure field **[14]**. This is an iterative solution procedure where the computation is initialized by guessing the pressure field. Then, the momentum equation is solved to determine the velocity components. The pressure is updated usingthe continuity equation. Even though the continuity equation doesnot contain any pressure, it can be transformed easily into a pressurecorrection equation **[13]**. The segregated solver was used to solve the governing integral equations for the conservation of mass, momentum, and energy. Because of the assumption of constant fluid thermo-physicalproperties and negligible buoyancy, themass andmomentumequationsare not coupled to the energy equation. Therefore, the temperature fieldis calculated by solving the energy equation after a converged solutionfor the flow field is obtained by solving the momentum and continuityequations.The iterations were continued until the sum of residualsfor all computational cells became negligible (less than 10^{-7})and velocity components did not change from iteration toiteration.

Results and discussion

For thermal analysis, the fluid used is water with a constantproperties that determined at the mean temperature across the entirelength of the channels. The temperature of the inlet water is taken to be 293 K . The inlet velocity is computed accordingto the value of Reynolds number, hydraulic diameter, and theproperties of fluid. The Reynolds number considered in this work ranged from 100 to 500. The heat flux that applied at the bottom plate ofMCHS was 50,100 and 150 W/m^2.

Temperature Contours

The temperature contour of outlet is shown Figs. (3) & (4) at different Reynolds number and different heat fluxes. For all types of microchannels, it is found that the high temperature region occurs at the edge of the heat sinks since there is no heat dissipation by fluid convection and another reason is due to the low velocity of the flow and maximum temperature of fluid at exit of channel.

Stream Line Contours

Figures (5) & (6) display the Stream Line contours of the fluid at Re=300. For all shapes, the velocity increase and this increase is due to the pressure drop in microchannel, so when the pressure difference between inlet and outlet decreases, the velocity also will decrease.

Temperature distribution

The change in local temperature for all shapes of the microchannel are shown in Figures (7),(8) and (9) respectively ,these temperature are calculated at the bottom wall. All shape of the MCHS, indicates that the highest temperature found in the regions at the edges of the heat sink and the reason for this is that there is no heat dissipation by fluid convection. Lower temperatures occur in areas far from the walls of the microchannel, especially in the central region due to the higher heat transfer coefficientand this is compatible with the results that published **[15,16,17]**. It is clear from Figures (7),(8) and (9) that the distribution of temperatures in the microchannel with notches of 6 be highest among all the shapes that studied in this work and the reason is due to the large contact area with the fluid. Figure also show that the straightmicrochannelhave the highest value of the thermal resistance and gradually decreases with increasing the number of notchesuntil reaches to the least resistance in the microchannel configuration of six notches.

Nusselt number

To analyze the Nusselt number variation, a relationship between average Nusselt number and Reynolds number is explained in Figures (10),(11),(12),(13) and(14). As expected, Nusselt number increases as the Reynolds number increases. The reason is that when the velocity is increased, flow rate of the fluid is increased as well, so its ability of removing heat is larger. When the result for the channels with notches is compared with those corresponding to the straight channel, the channels with notches can significantly increase the heat transfer performance.By comparing the square channel in the present study with the previous studies, it can be concluded that the curves behavior is almost similar, as it is clear in Figure (14) as published by Sui, Y., etal. **[18]**.

Friction factor

The variation of the friction factor with Reynolds number for various channel shapes is shown in Figure (15). The results show that the friction factor decreases with the increase of Reynolds number for all channel shapes. It can be inferred that the changing of Dean vortices (vortices near the edges of the channel) patterns along the flow direction through the channel with notches appears to give an increase in the friction factor compared to step and straight channels for all values of Reynolds number. However, it is found that the friction factor of seq4 channels is still higher than that of the seq2 and seq6 channels. By comparing the present results with previous study of P. Gunnasegaran, etal**[19]** it can be concluded that the curves behavior is almost similar, as it clear from Figure (16)

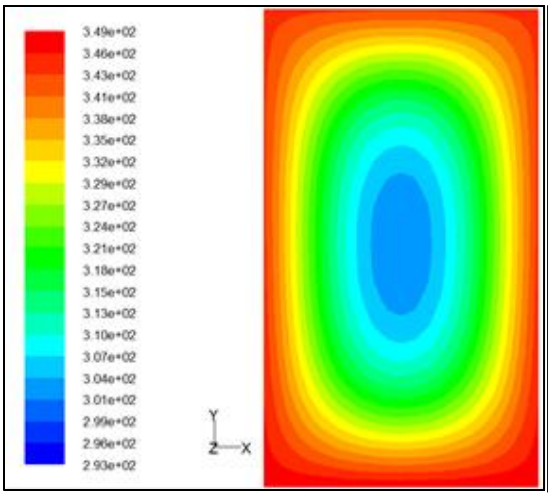


Figure 3: Temperature contour of Outlet at q=100W/cm^2 and Re=100

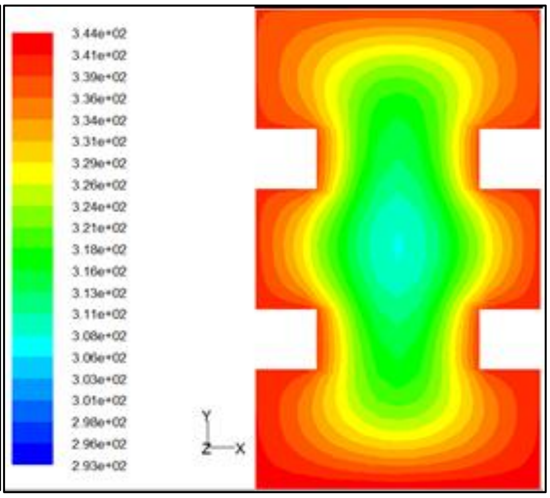


Figure 4: Temperature contour of Outlet at q=100W/cm^2 and Re=100

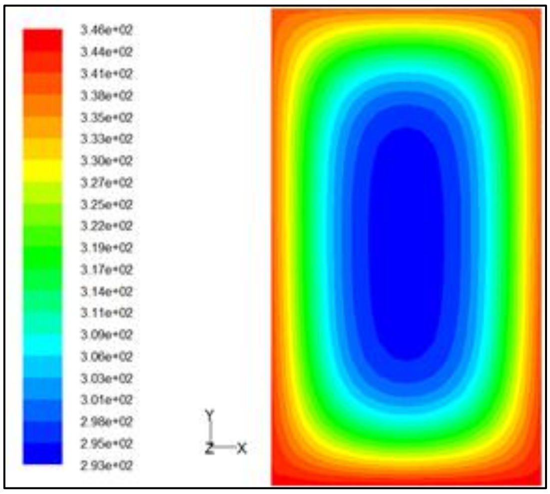


Figure 5: Temperature contour of Outlet at q=150W/cm^2 and Re=300

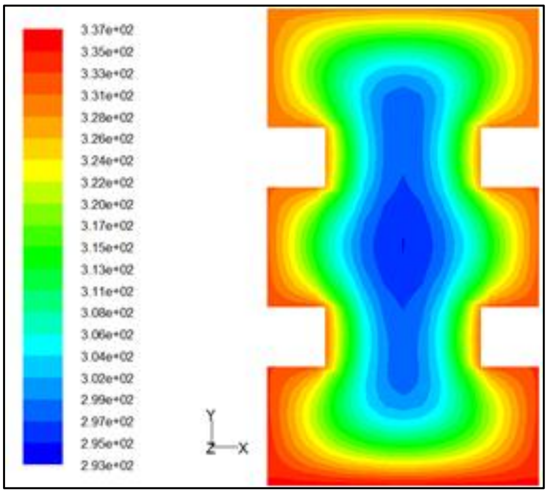


Figure 6: Temperature contour of Outlet at q=150W/cm^2 and Re=300

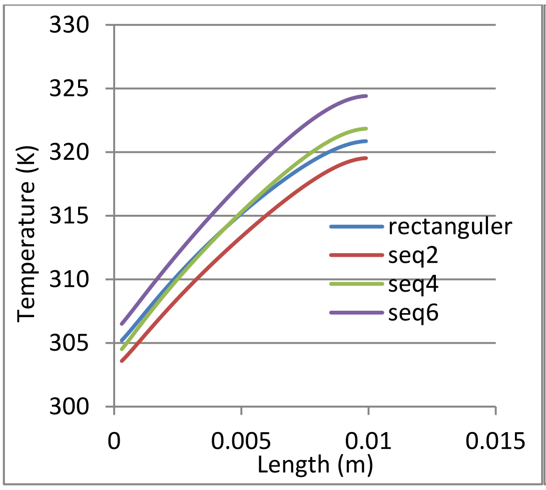


Figure 7: Variation on fluid temperature for pure water, at (Re=100) & (Q=50)W/cm^2 at bottom channel wall

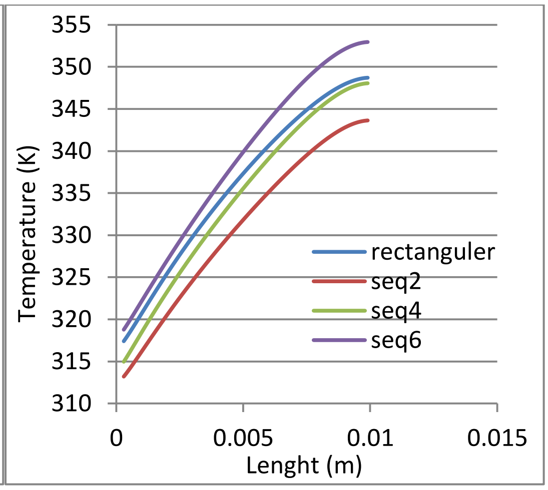


Figure 8: Variation on fluid temperature for pure water, at (Re=100) & (Q=150)W/cm^2 at bottom channel wall

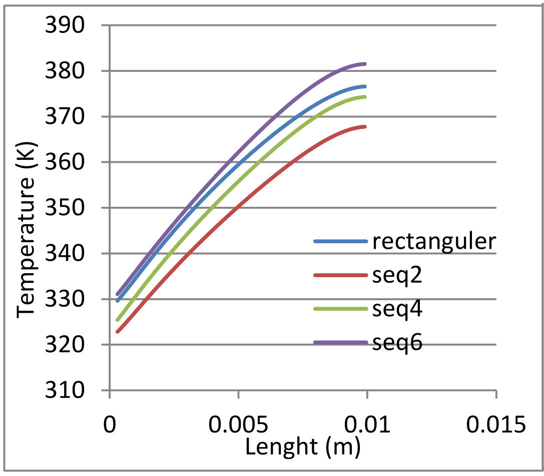


Figure 9: Variation on fluid temperature for pure water, at (Re=100) & (Q =150) W/cm^2 at bottom channel wall

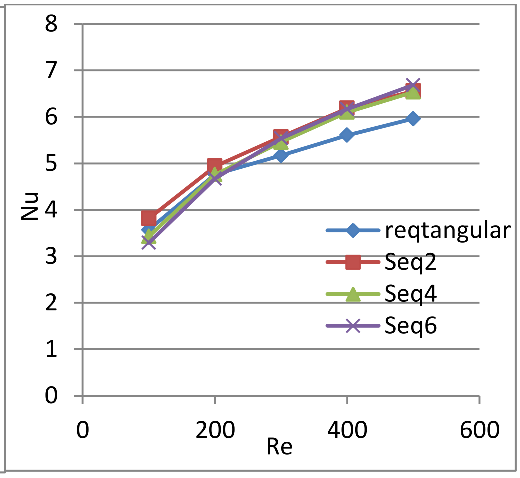


Figure 10: Variation of Nusselt number with Reynolds number for heat flux=50W/cm^2

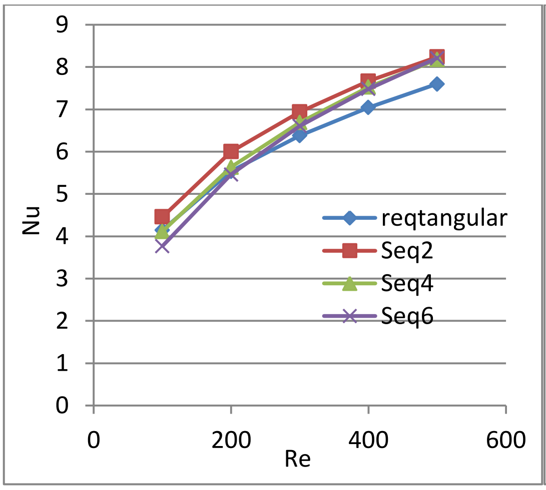


Figure 11: Variation of Nusselt number with Reynolds number for heat

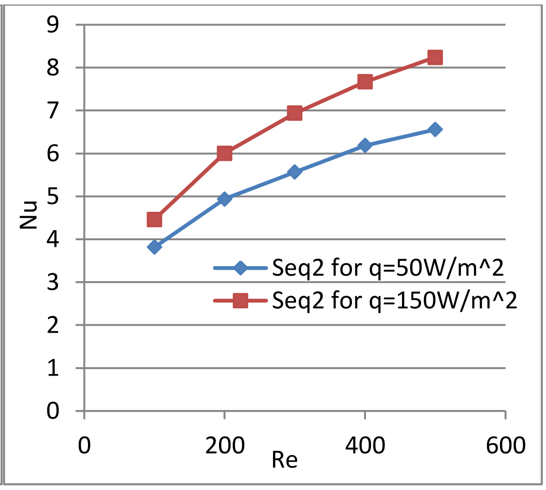


Figure 12: Variation of Nusselt number with Reynolds number for best results

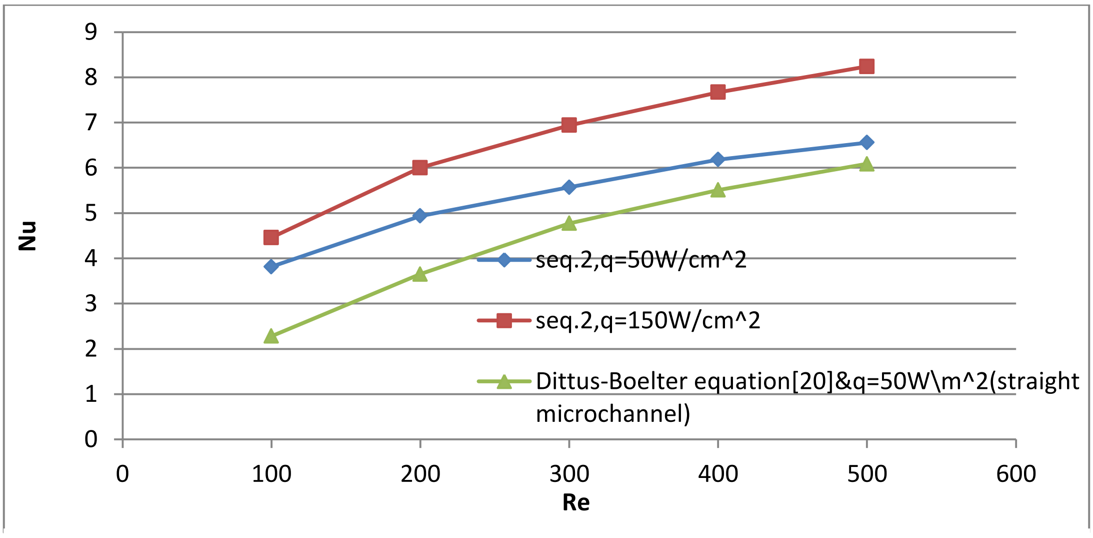


Figure 13: Nusselt number with Reynolds number

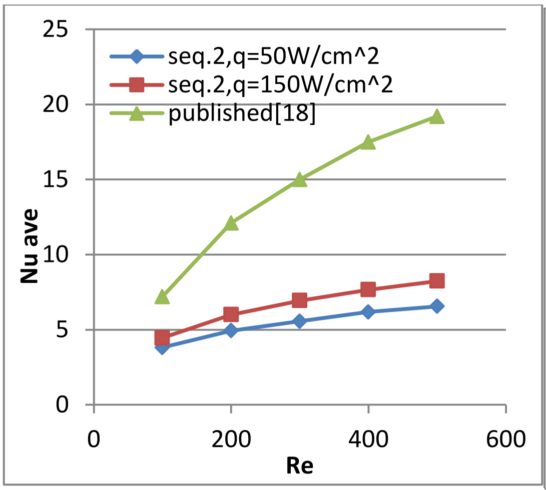


Figure 14: A comparison of Nusselt number result with the other previous numerical studies.

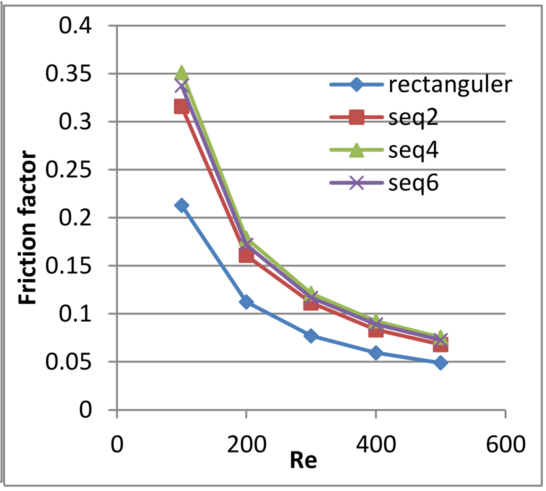


Figure 15: Friction factor of heat sink with Reynolds number for ($Q=100W/cm^2$).

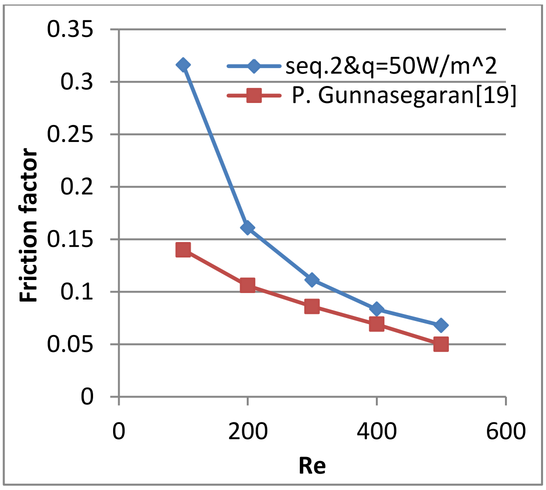


Figure 16: A comparison of variation friction factor result with the other previous numerical studies.

Conclusions

Numerical simulation of the fluid flow and heat transfer characteristics in full scale microchannel heat sinks was performed in this study. The effect of different geometrical such as the differences in hydraulic diameters by add notches for rectangular shape of three different shapes addition to straight rectangular shape of microchannel heat sinks was comprehensively studied. Based on the simulated results, the following conclusions can be made:

1- The highest temperature variation was in shape of notches 6, while the temperature in shape of notches 2 and 4 was smallest then straight rectangular shape.
2- The heat transfer coefficient and Nusselt number increase with the increase of Reynolds number. For microchannels with 2 notches shape, the heat transfer coefficient and Nusselt number are the highest, for microchannels with straight rectangular shape are the lowest, and for microchannels with 4 and 6 notches shape are in between.
3- The friction factor decreases with the increase of Reynolds number from bottom plate of heat sink for the same heat flux.

References

[1] Tuckerman, D.B. and Pease, R.F. 1981. High-performance heat sinking for VLSI. IEEE Electronic Devices Letters. EDL-2, 5, 126-129.

[2] Fedorov, A. G. and Raymond, V. 2000. Three-dimensional conjugate heat transfer in the micro channel heat sink for electronic packaging. International

Journal of Heat and Mass Transfer. 43 399-415.

[3] Rahman, M. M. (2000). Measurements of heat transfer in microchannel heat sinks. International Communications in Heat and Mass Transfer, 27(4), 495-506.

[4] Lelea, D., Nishio, S., & Takano, K. (2004). The experimental research on microtube heat transfer and fluid flow of distilled water. International Journal of Heat and Mass Transfer, 47(12), 2817-2830.

[5] Qu, W., &Mudawar, I. (2004). Measurement and correlation of critical heat flux in two-phase micro-channel heat sinks. International Journal of Heat and Mass Transfer, 47(10), 2045-2059.

[6] Kuan W.-K., Kandlikar S.G., Experimental study on saturated flow boiling critical heat flux in microchannels, *Proc. 4th Int. Conf. on Nanochannels, Microchannels and Minichannels*(2006) *Limerik, Ireland*.

[7] Liu, D. and Garimella, S.V. 2004, Investigation of liquid flow in micro channels, AIAA J. Thermo phys. Heat Transfer 18, 65–72.

[8] Xu, J.L. and Song, Y.X. 2008. Numerical simulations of interrupted and conventional micro channel heat sinks. International Journal in Heat and Mass Transfer. 51, 5906– 5917.

[9] Hetsroni, G., Mosyak, A., Pogrebnyak, E. and Yarin, L.P. 2005. Heat transfer in microchannels: Comparison of experiments with theory and numerical results. International Journal of Heat and Mass Transfer. 48, 5580–5601.

[10] Sabbah, R., Farid, M. M. and Al-Hallaj, S. 2008. Micro-channel heat sink with slurry of water with micro-encapsulated phase change material: 3D-numerical study. Int. J. Applied Thermal Engineering. 29, 445–454.

[11] Allen, P. W. 2007. Experimental and Numerical Investigation of Fluid Flow and Heat Transfer in Microchannels. Msc Thesis, Mechanical Engineering Department, Louisiana State University.

[12] Mathew, B. and Hegab, H. 2009. Application of effectiveness-NTU relationship to parallel flow microchannel heat exchangers subjected to external heat transfer. International Journal of Thermal Sciences. 30, 1-10.

[13] S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemisphere, New York,1980.

[14] W.Q. Tao, Numerical Heat Transfer, second ed., Xi'an Jiaotong University Press, Xi'an, China, 2001.

[15] Chein ,R., Chen ,J. (2009) . Numerical study of the inlet/outlet arrangement effect on microchannel heat sink performance, Int. J. Therm. Sci. 48 1627–1638.

[16] Gunnasegaran, P., Mohammed, H. A., Shuaib, N. H., &Saidur, R. (2010). The effect of geometrical parameters on heat transfer characteristics of microchannels heat sink with different shapes. International Communications in Heat and Mass Transfer, 37(8), 1078-1086.

[17] Mohammed, H. A., Gunnasegaran, P., &Shuaib, N. H. (2011). Numerical simulation of heat transfer enhancement in wavy microchannel heat sink. International Communications in Heat and Mass Transfer, 38(1), 63-68.

[18] Sui, Y., Teo, C. J., & Lee, P. S. (2012). Direct numerical simulation of fluid flow and heat transfer in periodic wavy channels with rectangular cross-sections.*International Journal of Heat and Mass Transfer*, *55*(1), 73-88.

[19] P. Gunnasegaran, H. Mohammed*, N. H. Shuaib,(2009). Pressure Drop and Friction Factor for Different Shapes of Microchannels.

[20] Yunus A. Gengel (2003),heat transfer "Aprictical Approach 2nd .McGraw-Hill"

دراسة عددية في انتقال الحرارة وجريان المائع في القنوات الماكيرو المختلفة

عمار عدنان عبيد
قسم الهندسة الميكانيكية
جامعة الانبار / كلية الهندسة

وسام هاشم خليل
قسم الهندسة الميكانيكية
جامعة الانبار / كلية الهندسة

الخلاصة

القنوات متناهية الصغر في الوقت الحالي هي محط اهتمام الكثير في مجال المبادلات الحرارية , والمفاعلات الدقيقة ,والانظمة الكبيرة والتي تتطلب كفاءة عالية في انتقال الحرارة. الأنظمة الالكترونية مهمة جدا في الحياه اليومية في الوقت الحاضر لذلك من الضروري جدا ايجاد وسائل تبريد عالية الكفاءة لهذه الأنظمة والقنوات متناهية الصغر هي أفضل وسيلة لانتقال الحرارة والتي يكون لديها معامل انتقال عالي بالحمل .هذا البحث يتناول دراسة عددية لانتقال الحرارة وجريان المائع من خلال قنوات مختلفة المقطع وهذا الاختلاف يتم عن طريق اضافه نتوءات مربعة الشكل ومختلفة العدد على طول المجرى مستطيل المقطع. وسيكون الماء هو المائع المستخدم في هذا البحث.ديناميك الموائع الاحتسابي(CFD) وكذلك رسم الأشكال ثلاثية الابعاد 3D باستخدام برنامج ال(FLUENT) لحساب كفاءة انتقال الحرارة وجريان المائع وتحليل النتائج وقد تمت الحسابات على فرض ان الجريان مستقر وطباقي باستخدام المعادلات الحاكمة.النتائج أظهرت ان انتقال الحرارة ورقم نسلت قد ازدادت للأشكال التي تم اضافه النتوءات اليها مقارنة مع الشكل الاصلي الخالي من النتوءات. النتائج تم مقارنتها مع بحوث سابقة وقد أظهرت توافق جيد معها.