International Journal of Engineering Technology and Applied Science
(ISSN: 2395 3853), Vol. 2 Issue 2 February 2016

Experimental and Numerical Analysis of Single Phase Flow Micro-Channel Heat Sink

Sujeet Kumar [1], Nitin Dubey[2]
Mechanical Engg. RGPV Bhopal[1], Mechanical Engg. RGPV Bhopal[2]
sujeet.me32@gmail.com [1], nitindubey20@yahoo.com[2]

ABSTRACT:- The reliability of electronic components is affected critically by the temperature at which the junction operates. As operating powers and speed increase, and as designers are forced to reduce overall system dimensions, the problems of extracting heat and controlling temperature can become crucial. The continuing increase of power densities in electronics packages and the simultaneous drive to reduce the size and weight of electronic products have led to an increased importance on thermal management issues in this industry. Micro Channel Heat sinks are commonly used devices for enhancing heat transfer in electronics components. A theoretical study of single phase micro channel heat exchanger has been carried out. The computational fluid dynamics (CFD) model equations have been solved to predict the hydrodynamic and thermal behavior of the exchanger. The geometry of the problem and meshing of it have been made in ICEM CFD. The models have been solved by CFX solver. Numerical results have been compared also with experimental result. A closed loop closing system has been set up for the dissipation of heat from the micro channel.

KEYWORDS:- computational fluid dynamics, micro- channel, heat sink, heat transfer, meshed geometry, fabrication technology, micro-chip, nano-size

I. INTRODUCTION

While performance of microprocessors and electronic chips becomes advanced, their size gets smaller and they will be more densely integrated in the near future. Besides, a lot of electromechanical devices and systems have been miniaturized by micro fabrication techniques. But such miniaturization and integration accompany greater heat generation per unit area.

Currently, air-cooled heat sinks are the most viable solutions for cooling electronic devices, primarily because of their low cost and high reliability. A large amount of research work on air-cooled heat sinks has been published in the past several decades, and significant improvements of heat sink designs have been achieved based on CFD analyses and experimental investigations.

For high-heat-releasing electronics, air cooling methods have been found to be insufficient in recent years. To improve the heat dissipation flux per unit area in a heat sink, liquid cooling is becoming more and more popular. Liquids usually have a higher heat capacity and thermal conductivity than air, and therefore can significantly improve the heat transfer rate, and lower the maximum temperature level on a heat sink. Some liquids can also be managed so as to take advantage of phase change heat transfer, which can dramatically improve the cooling capability of heat sinks.

In recent years, maturation of computational fluid dynamics (CFD) software codes tailored for applications in the electronics industry and the availability of powerful low cost workstations have made possible the simultaneous solution of both the heat transfer and fluid dynamic problems in undertaking thermal design of electronic devices. With the advancement of fabrication technologies, micro channel heat exchangers have been developed in the last two decades. Micro-channel heat exchangers enable liquid to flow through channels of a hydraulic diameter of 100 - 1000 mm and the heat transfer surface area can be dramatically increased.

II. EXPERIMENTAL SETUP

Procedure

a) The micro-channel heat sink of 1mm x 0.5mm is drilled for holes of 3mm diameter at the top surface and lateral surface for the inlet and outlet of water into and out of the sink.
b) A submersible pump for continuous flow head is used for a flow through a pipe.
c) A flow control valve is used to regulate the flow through change in pressure across it.
d) Pioes are connected to make the set up a closed loop system.
e) Three holes of 0.5mm diameter are drilled to insert thermocouple probes to the top surface of the heat channel.
f) Three thermocouples (12-T-1000-118-0,5-21-3P6M) are taken and used to measure the temperature at three positions in the micro- channel along the flow at 10 mm, 20mm and 30mm from the inlet face of the micro-channel.

g) To acquire the temperature data we have used NI DAQ system.
h) Cartridge heaters were used to heat the copper plate.

<u>Heaters Configuration:</u>

- Square Type Made of Brass.
- 8 x 8 x 40mm L x 75W / 220V.

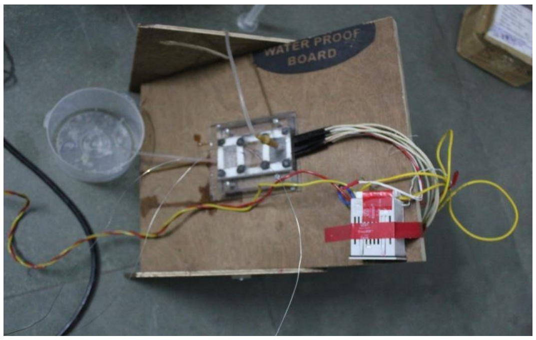
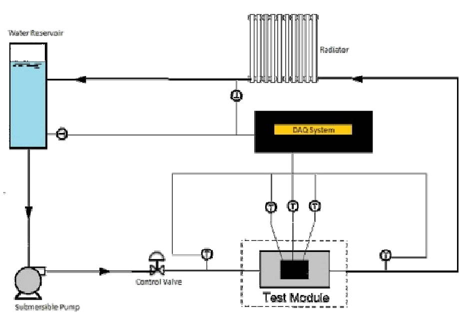


Fig 1: Final Set-up With Rotary Stand

The final test setup is shown in fig. .The three thermocouples were fixed in holes drilled at 10mm, 20mm and 30mm from inlet. Three other thermocouples were used to gauge the temperature at the inlet of radiator (outlet of heat sink), outlet of the radiator and temperature of the reservoir. The other side of thermocouple through a connector was connected to the input port of the NI DAQ system. Lab view software was used to acquire the readings during experiment. Flow was measured using measuring flask at the beginning of the experiment. We have acquired data (temperature) at steady state of the micro-channel heat sink for different flow rates and different angles of inclination (0^0,45^o,90^o).



III. FIGURES

A. Figures

The results of the experiments conducted have been plotted and have been compared with those obtained with the numerical simulation technique using CFX. The flow velocities in the micro-channel for which the tests were conducted are 0.22 m/s, 0.155 m/s and 0.117 m/s.

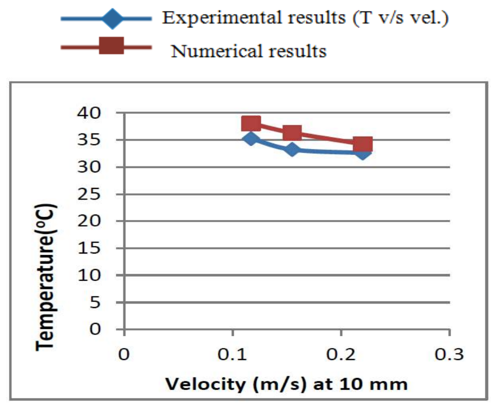


Fig 1.1: Graph between temperature & velocity measured at 10 mm from inlet at an angle of 0^0

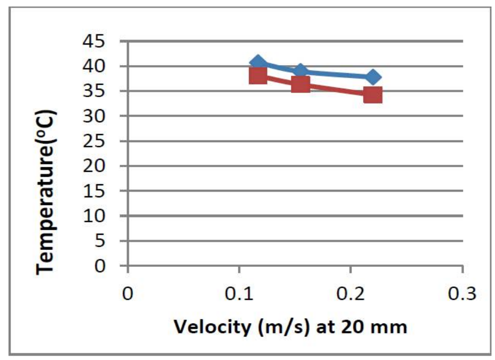


Fig 1.2: Graph between temperature and velocity measured at 20 mm from inlet at an angle of 0^0.

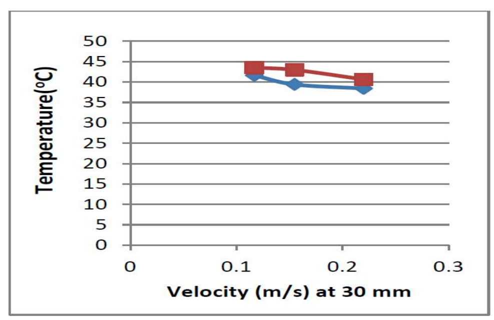


Fig: 1.3 Graph between temperature and velocity measured at 30 mm from inlet at an angle of 90^0

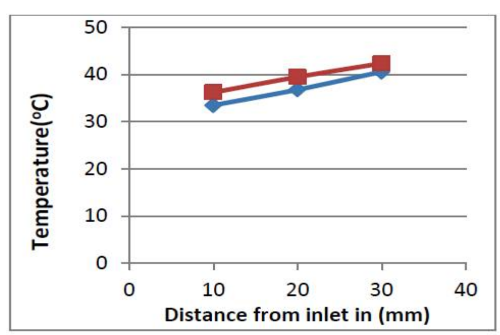


Fig.1.4: Graph between Temperature and distance for inlet velocity V_{in} = 0.22m/s at an angle of 45^0.

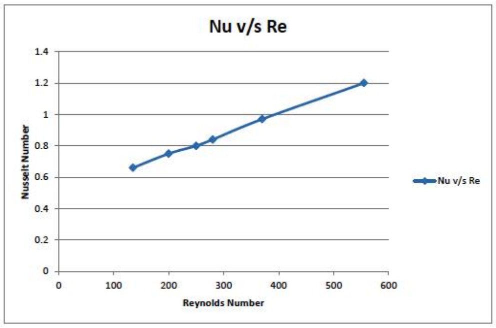


Fig.1.5: Graph between Nusselt Number and Reynolds Number

B. Equations

They should be numbered consecutively throughout the text. Equation numbers should be enclosed in parentheses and flushed right. Equations should be referred to as Eq. (X) in the text where X is the equation number. In multiple-line equations, the number should be given on the last line.

$$Nu = \frac{q_{conv}}{q_{cond}} = \frac{h\delta}{k} \quad \text{………. (1)}$$

$$h = \frac{-k\frac{dT}{dy}}{T_s - T_\infty} \text{ at } y = 0 \text{ so the factor } \frac{dT}{dy} \text{ is maximum for high velocities.}$$

….. (2)

$$Re = \frac{\rho v l}{\mu} \quad \text{…….. (3)}$$

IV. CONCLUSION

The experimental study of the heat transfer is a complex problem because of the small sizes of such channels. It makes difficult direct diagnostic of temperature field in fluid flow and in solid wall. The three-dimensional fluid flow and heat transfer processes in a rectangular copper micro-channel heat sink were analyzed numerically, and a detailed description of the i.e. temperature, heat flux, and Nusselt number, was obtained

a) Laminar, incompressible, fluid flow in micro-channels of rectangular geometry was studied numerically and experimentally.
b) It is possible to reveal the effects of: the Reynolds number, axial conduction, energy dissipation, heat losses to the environment, etc. on the heat transfer.
c) The dimension of rectangular micro channel are 1x0.5x41 mm and the flow regime was laminar with Reynolds number Re = 135-555.
d) Numerical analysis was also carried out by solving a conjugate heat transfer problem involving simultaneous determination of the temperature field in both the solid and fluid regions.
e) The temperature rise along the flow direction in the solid and fluid regions of the micro-channel heat sink can be approximated as linear. The highest temperature point is located at the heated base surface of the heat sink, which is immediately above the channel outlet. The temperature along the transverse y-direction at a given longitudinal distance x is nearly constant.
f) These findings demonstrate that conventional Navier–Stokes and energy equations can adequately predict the fluid flow and heat transfer characteristics of micro-channel heat sinks.
g) We have successfully conducted the experiments and have come to a conclusion that heat dissipation through micro channel heat sink depends largely on flow rate. We also conducted test for different angles of inclination of the micro channel and found that gravity effects on the overall performance of the heat sink is negligible. And largest variation that we found through our experiment was less than 10 percent.
h) Nusselt Number increase almost linearly with Reynolds Number [3].

i) Considerable amount of heat is dissipated in the micro channel heat sink for the close loop system and highest heat dissipation efficiency was found to be 52.96% at Re =555.
j) Increasing Reynolds number increases the length of the developing region. Fully developed flow may not be achieved inside the heat sink for high Reynolds numbers. This results in enhanced heat transfer, alas at the expense of a higher pressure drop.

REFERENCES

[1]. M.B. Dogruoz, M. Urdaneta, A. Ortega, Experiments and modeling of the hydraulic resistance and heat transfer of in-line square pin fin heat sinks with top by-pass flow, Int. J. Heat Mass Transfer 48 (2005) 5058e5071.
[2]. Y.T. Yang, H.S. Peng, Numerical study of pin-fin heat sink with un-uniform fin height design, Int. J. Heat Mass Transfer 51 (2008) 4788e4796.
[3]. Peng, X. F., Wang, B. X., Peterson, G. P., and Ma, H. B. 1995. Experimental

investigation of heat transfer in flat plates with rectangular micro channels. International Journal of Heat and Mass Transfer. 38,127-137.

[4]. E. Walsh, R. Grimes, Low profile fan and heat sink thermal management solution for portable applications, Int. J. Thermal Sci. 46 (2007) 1182e1190.

[5]. G. Ledezma, A. Bejan, Heat sinks with sloped plate fins in natural and forced convection, Int. J. Heat Mass Transfer 39 (9) (1996) 1773e1783.

[6]. D.C. Wadsworth, I. Mudawar, Cooling of a multi chip electronic module by means of confined two dimensional jets of dielectric liquid, J. Heat Transfer112 (1990) 891e897.

[7]. S.C. Tzeng, Spatial thermal regulation of aluminum foam heat sink using a sintered porous conductive pipe, Int. J. Heat Mass Transfer 50 (2007) 117e126.

[8]. G. Hetsroni, M. Gurevich, R. Rozenblit, Sintered porous medium heat sink for cooling of high-power mini-devices, Int. J. Heat Fluid Flow 27 (2006) 259e266.

[9]. M.K. Sung, I. Mudawar, Single-phase and two-phase hybrid cooling schemes for high-heat-flux thermal management of defense electronics, J. Electronic Packaging 131 (2009) 021013.

[10]. K.C. Nayak, S.K. Saha, K. Srinivasan, P. Dutta, A numerical model for heat sinks with phase change materials and thermal conductivity enhancers, Int. J. Heat Mass Transfer 49 (2006) 1833e1844.

[11]. Wang, B.X., Zhou, L.P., Peng, X.F., 2003. A fractal model for predicting the effective thermal conductivity of liquid with suspension of nanoparticles. International Journal of Heat and Mass Transfer 46, 2665–2672.

[12]. Jiang, P., Fan, M., Si, G. and Ren, Z. 2001. Thermal–hydraulic performance of small scale micro-channel and porous-media heat-exchangers. International Journal of Heat and Mass Transfer, 44, 1039-1051.