Journal of Advances in Science and Technology
Vol. 13, Issue No. 1, (Special Issue) March-2017, ISSN 2230-9659

Experimental and Numerical Investigation of Conventional Micro-Channel for Electronic Cooling

Abhinandan Kumar[1]*, S. S. Mohite[2], V. P. Gaikawad[3]

[1]PG Scholar (Department of Mechanical Engineering) GCOE, Karad

[2]Professor Govt. College of Engineering Karad

[3]PG Scholar (Department of Mechanical Engineering) GCOE Karad

Abstract – In this paper, performance of straight micro-channel for two heat fluxes 48.3 & 64.3 W/cm^2 are studied both experimentally and numerically. Rise in surface temperature and fluid temperature are checked for four different flow rates 300 ml/min, 400 ml/min, 500 ml/min and 600 ml/min. Experimentally It is observed that average rise in surface temperature is below 60^{o}C. A maximum temperature of 71^{o}C is recorded for one case with heat flux 64.3 W/cm^2 and flow rate 300ml/min that shows fitness of straight micro-channels for electronic cooling for high heat fluxes. The study also shows that Nusselt Number (Nu) and Reynolds Number (Re) varies linearly and it is found that numerical results are in very good agreement with experimental result with an error of below 3%.

Keywords— Micro-channel, Heat Sink, Electronic Cooling, Conventional micro-channel, Forced Convection, Liquid Cooling

---------------------------♦-----------------------------

INTRODUCTION

In electronic industry, one of the peculiar design aspect is to have an efficient cooling device, which is also called as heat sink. Since, according to joules law of heating, the heat generated 'H' in "joule", in any electronic component, is directly proportional to the square of electronic current 'I' (amp), directly proportional to resistance 'R' (Ω) of the conductor and directly proportional the time 't' of electric current passing through it. Due to the technological advancement, not only the heat generated by electronic component is high in the range of few watt to thousand watt but also with the other devices like laser, microfluidics, bio-medical appliances, micro pumps etc. The heat generated by these devices must be rejected to increase its life- span by reducing thermo-mechanical stress generated, due to gigantic rise in temperature and, to increase its functionality by keeping the device temperature at optimum level.

The first significant work in this field was done by Tuckerman and Pease [1] in 1981. First time, they checked the feasibility of liquid cooling and suggested the feasibility of ultra high speed electronic devices. Experiments were performed on silicon wafer consisting parallel channel on it, through which coolant water was passed. The thermal contact resistance recorded was 0.09^{o}C/W. Since then, lots of developments have taken place. The major trend in this field is found, either the study is performed experimentally or numerically or both to check the performance, optimization, feasibility, pumping power or pressure loss, capacity, uniform temperature gradient, flow pattern etc. few of them are discussed below.

Tuckerman and Pease [1] in their further work studied about the packaging of micro-channel heat sink for electronic cooling. Owing to the difficulty associated with study of two phase flow, majority of research focussed on single phase flow. However, phase change material and other coolant like R-134a and water Nano-fluid combination is also tried. Lee et al. [2] studied numerically the flow and heat transfer enhancement in sectional oblique fins. They have suggested that oblique fins facilitate uniform temperature distribution, enhanced heat transfer than the conventional straight channel and lesser pressure drop than continuous straight channel due redevelopment of boundary layer in oblique fin flow. Xu et al. [5] numerically studied the various configuration of tree shaped microchannel with loop network and also with without loop network. They suggested the advantages of using micro-channel with loop, particularly, during blockages of any channel by maintaining the continuous flow rate. Chen and Cheng [6] experimentally studied the thermal efficiency of fractal tree like micro-channel net and found that the higher thermal efficiency (defined as heat transfer rate per unit power required) than the conventional straight channel for the same boundary conditions. G. Hetsorni et al. [9] done a comparative study of heat transfer data for different shaped microchannel viz. circular, triangular, rectangular and trapezoidal micro-channels with hydraulic diameter ranging from 60μm to 2000μm to compare the experimental data obtained by different researchers.

From literature, it is concluded that researchers have focussed on different flow pattern for example, “leaf pattern” and preached their superiority over conventional straight channel but still there is a dire need to pay keen attention to the straight micro channel by changing the design of test setup or by changing the material of the work piece or by using other coolant, as “it is easy and simple to manufacture the straight micro channel”. Such an attempt is made in this work. In the next section, experimental setup is described.

EXPERIMENTAL SETUP

Straight micro-channel of depth 1.5 mm, width 0.5 mm and length 25 mm were engraved on a rectangular block of copper material. In the block two holes were drilled of depth 50 mm and 51 mm and diameter 13.94 mm to insert two cylindrical cartridge heater of capacity 350 W each, in it. The details dimensions of the test piece are shown in the table1. Dimensions were measured through CMM.

Table1. Dimensions of Test Piece

S. No	Part	Dimension (mm)
1.	Hole diameter	14.10
3.	Depth of hole 1	50.00
4.	Depth of hole 2	52.00
5.	Well depth	2.30
6.	Well width	6.30
7.	Well width	6.10
8.	Cartridge heater dia.	13.95
9.	Channel length	25.00
10.	Channel width	0.50
11.	Channel depth	1.50
12.	Test piece length	19.00
13.	Test piece depth	56.00
14.	Hydraulic diameter	0.75

The image of experimental setup is shown in figure1 below:

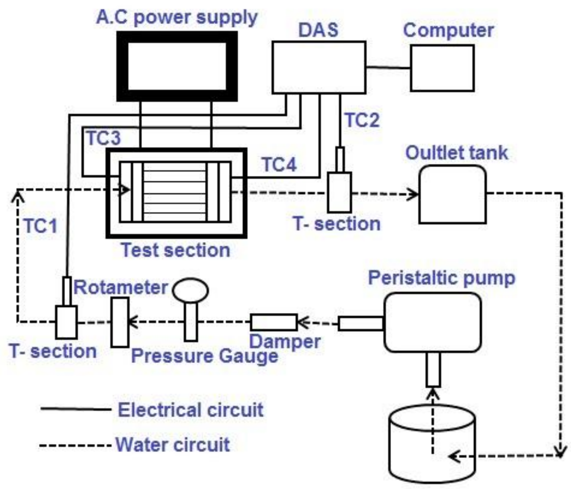

Fig.1 Schematic of experimental setup (flow loop)

The experimental setup consists of a Verderflex peristaltic pump, a pressure gauge, a rotameter, AC power supply, Dimmerstat, data accusation system(DAS), K-type thermocouple, inlet tank and outlet tank.

Deionised water from inlet tank is pumped by the peristaltic pump to inlet of micro-channel. Use of peristaltic pump helps in eliminating the contamination of DI water as the pumping action is done by squeezing action on tube. Though, the flow coming from the peristaltic pump is pulsating in nature, it is made continuous by using a damper at outlet of the pump. The pumped water passes through pressure gauge, Rotameter into the microchannel heat sink. After absorbing heat from MCHS, the water is collected in outlet tank which is again directed back to the inlet tank.

Fig.2 Experimental Setup

1. *Experimental Procedure*

Heat input is set via dimmerstat by changing the voltage and current is noted for each voltage and product of these two (V x I) gives heat input. For this work, a constant heat supply of magnitude, 300 W, and 400 W is provided to the cartridge heaters and the readings are noted for various flow rates. Temperature readings were recorded through Atomberg data acquisition system for temperature to get the inlet T_{fi}, outlet fluid temperature (T_{fo}) and two surface temperatures (T_{si} and T_{so}) near inlet and outlet respectively. Pressure readings were collected after each five minutes till steady state reached.

2 *Experimental Results*

The heat carried by the fluid is given by the formula q = $\dot{m}C_p\Delta T$ (1) and q is equivalent to VI the heat input. More than 70% heat is carried by the fluid as sensible heat gain by the coolant D.I water. Where, $\dot{m}=\rho Q$ (2)

The fluid properties were evaluated by the following formulae:

Density:

$$\rho(T) = \frac{a_0 + a_1T + a_2T^2 + a_3T^3 + a_4T^4 + a_5T^5}{1+bT} \quad (3)$$

Value of constant coefficients are given in table 2 and T in degree Celsius (^{o}C).

Table 2. Value of constant coefficients

a_0	= 999.8396,
a_1	= 18.22494
a_2	= -7.92221×10^{-3}
a_3	= $-5.544485 \times 10^{-10}$
a_4	= 1.49756×10^{-10}
a_5	= -3.93295×10^{-10}
b	= -1.81597×10^{-2}

Specific heat capacity

$$C_p(T) = 8958.9 - 40.54T + 0.11T^2 - 1.01\times10^{-3} \quad (4)$$

Thermal Conductivity (K),

$$K(T) = -0.58 + 6.36\times10^{-3}T - 7.96\times10^{-6}T^2 \quad (5)$$

Dynamic Viscosity (μ),

$$\mu(T) = 2.414\times10^{-5}\times10^{\frac{247.8}{T-140}} \quad (6)$$

For equation (4), (5) and (6) 'T' should in Kelvin.

For friction factor calculation following formula is used.

$$f = \frac{\Delta P_{ch} D_h}{2\rho V^2_{avg} L} \quad (7)$$

Figure 3 plots the Nusselt number versus Reynolds number for 300 W and 400 W heat supplied to the micro channel heat sink. The Nusselt number increases with increasing Reynolds number as the boundary layer thickness decreases. However, the rate of increase of Nusselt number is higher for 400 W conditions. At lower Reynolds number the Nusselt number is nearly same, but for higher Reynolds number the difference is nearly 22 percent.

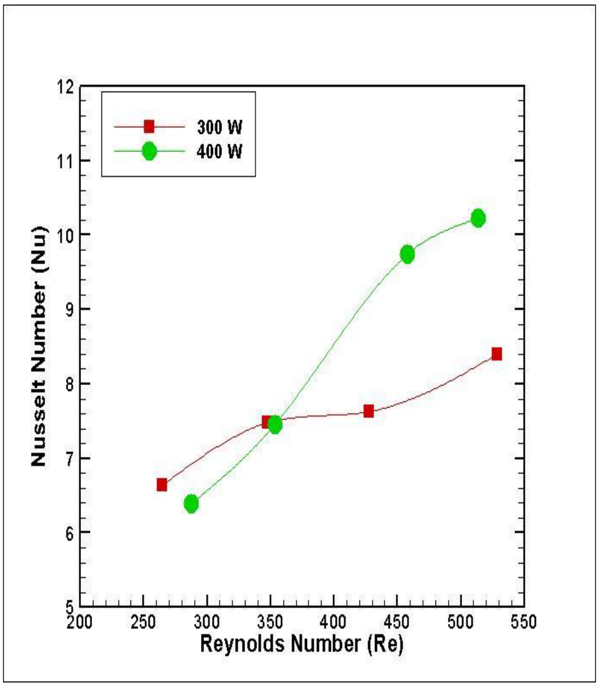

Fig.3 Comparison of Nusselt number for various heats supplied

Abhinandan Kumar[1]*, S. S. Mohite[2], V. P. Gaikawad[3]

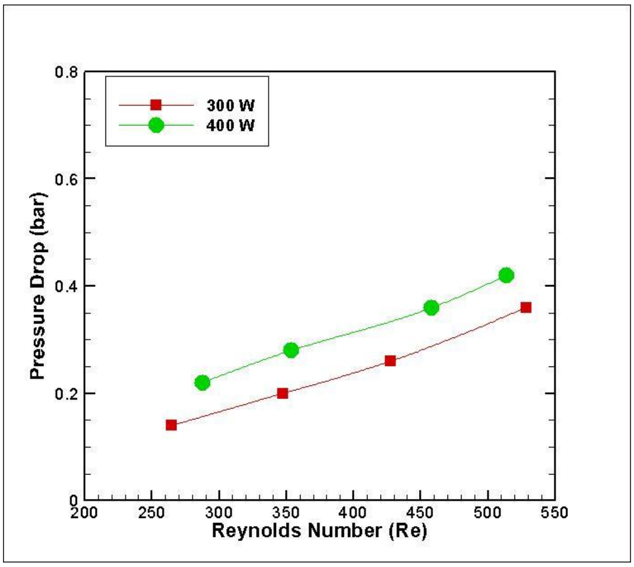

Fig.4 Comparison of Pressure drop for various heat supplied

The experimental pressure drop across microchannel heat sink for 300 W and 400 W of heat supplied are as shown in figure 4. In both cases the pressure drop increase with increase in Reynolds number, though the pressure drop is slightly more in case of 400 W heat supply.

The effect of flow rate on the surface temperature is also an important parameter to judge the thermal performance of MCHS. Figure 5 shows the rise in temperature of microchannel above the water inlet temperature. For low flow rates the surface temperature rise is nearly 50°C while at higher flow rates the rise is 31°C.

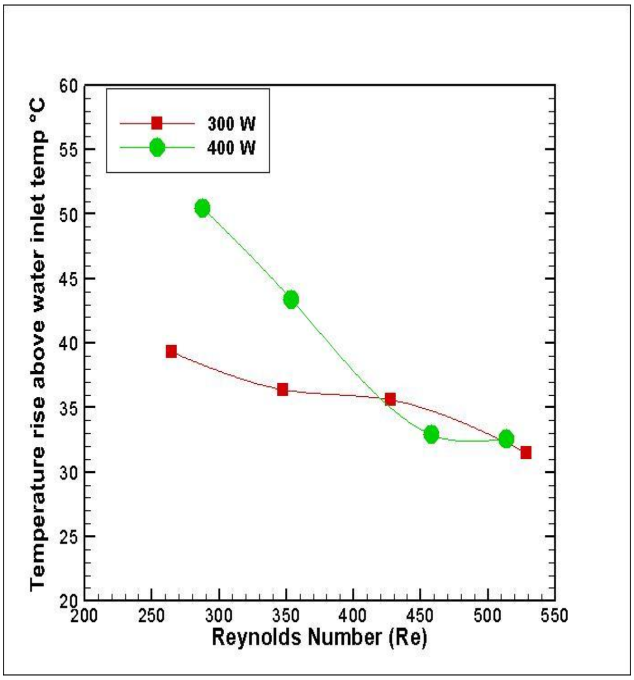

Fig.5 Comparison of temperature rise above water inlet temperature various heat supplied

NUMERICAL SIMULATION

Due to symmetry of micro channel a single channel is extruded in ANSYS-FLUENT version 18.0 design modeller. The length 'L' of the channel was kept 25mm and width 'W' is 0.25mm and depth 'H' was 1.5mm. First sweep method is applied at the bottom then, edge sizing is applied at top and side edge of the channel with face element size 0.000005mm. The final mesh count is 23 lakh cells. Grid independence is done with staring cell count 6 lakhs and with face element size 0.0005mm. For entire simulation domain hex mesh is selected with very good quality (Aspect ratio 1, skewness below .1 etc.).

After modelling and meshing, 3D double precision pressure based solver is selected in FLUENT setup. Viscous laminar model is selected with energy equation on.

Boundary conditions applied are velocity inlet, Constant heat flux at bottom and pressure outlet. Material selected is copper for solid and water-liquid for fluid.

For solution method, standard discretization scheme is selected for pressure along with second- order upwind discretization scheme for both momentum and energy equation.

RESULT AND DISCUSSION

In this section the results obtained from simulation is discussed. Temperature and velocity contour and X-Y plot images are included.

Temperature Contours:

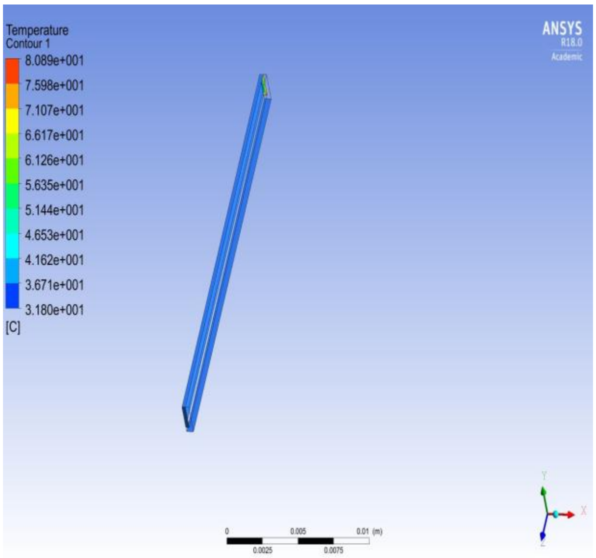

Fig.6 Temperature Contour for Q=300 m/min & q = 400 W

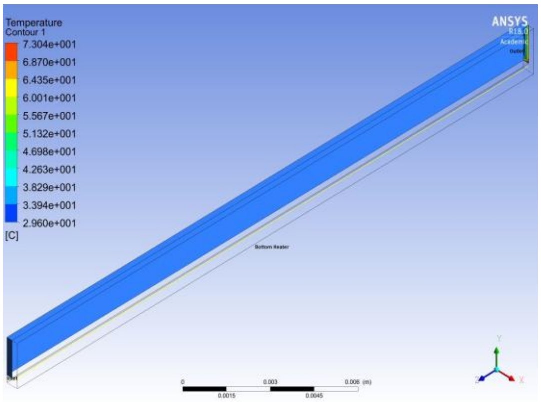

Fig.7 Temperature Contour for Q= 400 ml/min

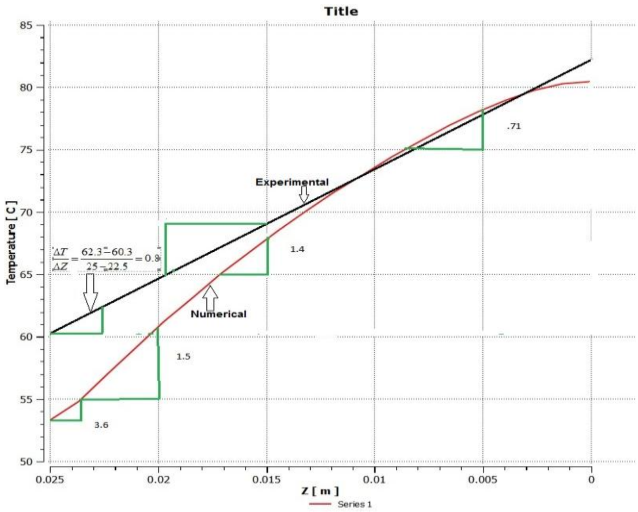

Fig.8 Exp. v/s numerical thermal gradient for 400 W &300 ml/in

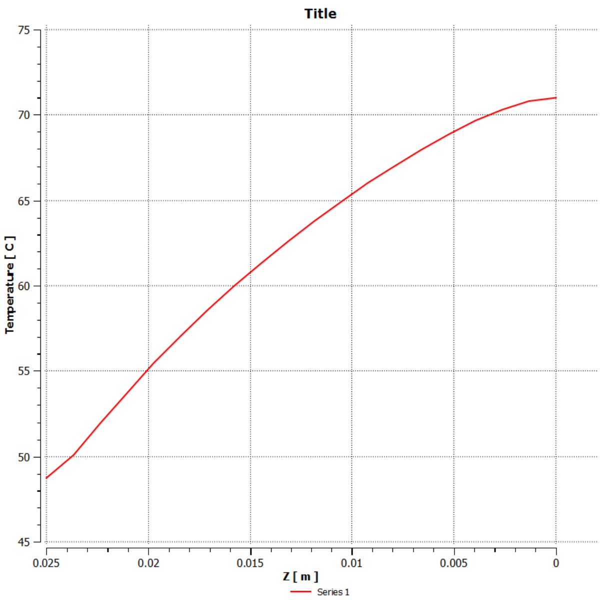

Fig.9 X-Y chart for Q = 500ml/min & q = 400 W

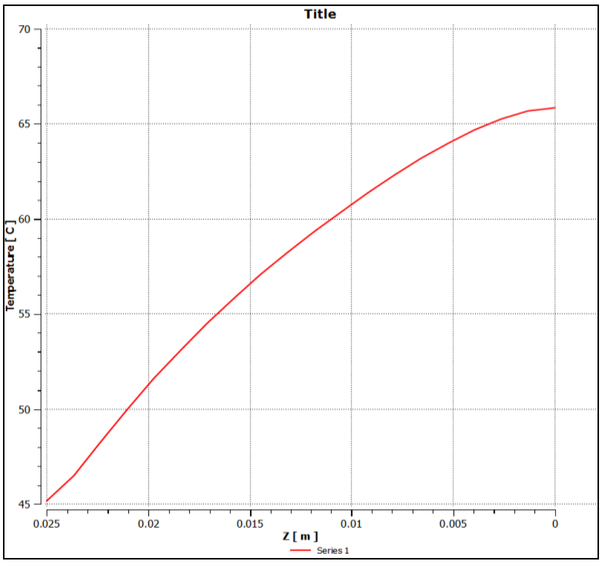

Fig.10 X-Y chart for Q = 600ml/min & q = 400 W.

Velocity Contours for q= 400W

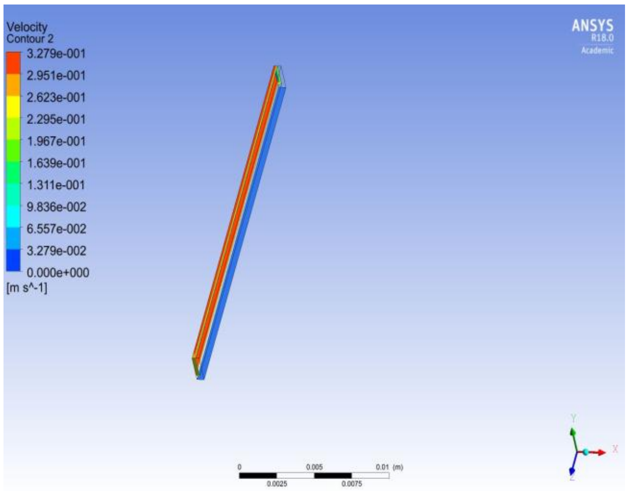

Fig.11 Velocity contour for Q= 300 ml/min

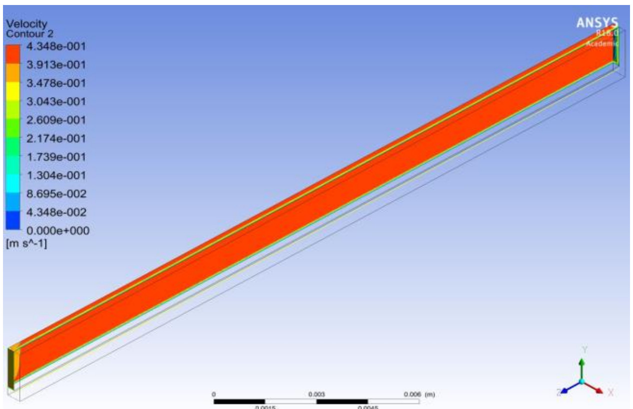

Fig.12 Velocity contour for Q = 400 ml/min

From temperature contour it is observed that straight micro channel benevolently takes heat from bottom heater and maintains the temperature below required temperature for electronic cooling, normally, it is blow below 71°C. But a hot spot is seen at the outlet of the channel, where further attention is required.

CONCLUSIONS

In this work, thermal and flow behavior of conventional micro channel is studied, with an objective of its suitability at higher heat fluxes. Following key conclusions from this work are summarized below-

1. Conventional micro channel is able to cool the electronic devices emitting heat flux of as high as 64.25W/cm^2.
2. This micro-channel can also be used for other applications like laser, ultrahigh electronic devices etc.
3. However, it represents non uniform thermal gradient, a low value at inlet with a temp-hot spot at outlet.
4. Results of this work can also be used for comparison with other enhanced flow pattern which is our further work.

ACKNOWLEDGMENT

This research work is supported by PG Heat Power Project through TEQIP-II, R & D grant of Govt. College of Engineering Karad, Maharashtra, India.

Nomenclature

Re = Reynolds number

Nu = Nusselt number

Pr = Prandtl number

T = Temperature, °C

T_{fi} = Inlet fluid temperature, °C

T_{fo} = oultlet fluid temperature, °C

Tsavg = avg. suface temperature, °C

C_p = specific heat capacity

Q = flow rate, ml/min

q = heat input, W

q'' = heat flux W/cm^2

K = thermal conductivity, W/mK

v = velocity m/s

ΔP_{ch} = pressure drop (mbar)

L = length of Channel, mm

w = Channel width, mm

D_h = hydraulic diameter, mm

H = depth of microchannel

Greek Symbols

Δ = gradient

μ = dynamic viscocity, Ns/m^2

ρ = mass density, kg/m^3

Subscript

Ch = channel

avg = average

fi = fluid inlet

fo = fluid outlet

si = surface inlet

so = surface outlet

REFERENCES

[1] Tuckerman, D. B., Pease, R. F. “High Performance Heat Sinking for VLSI”, IEEE Electronic Device Letters, EDL- 2 (1981)

[2] Lee Y.J., Lee P. S., Chou S. K.,“Numerical Study of fluid flow and heat transfer in the enhanced microchannel with oblique fins”, ASME Journal of Heat Transfer Vol. 135, (2013)

[3] Xiang-Qi Wang et al., “Flow and thermal characteristics of offset branching network” IJTS, 49 (2010) 272-280

[4] D.A Kamble and B.S Gawali, “Experimental and numerical investigation of forced Convection Heat Transfer in Rectangular Microchannels” IJMNST, Vol 5 Nov 2014

[5] Peng Xu, X.Q Wang, A.S. Majumdar, C.Yap, B.M. Yu, “Thermal characteristics of tree

shaped microchannel nets with/without loops, IJTS ,48 (22009) 2139-2147

[6] Yongping Chen and Ping Cheng, “An experimental investigation on thermal efficiency of fractal tree-like microchannel nets”, ICHMT 32 (2205)931-938

[7] Richard J. PHILIPIS, “ Microchannel Heat Sinks”, The licoln laboratory Journal, Volume 1, Number 1 (1988)

[8] Yunus. A. Cenjel , “Heat Transfer: A Practical Approach” 3rd edition

[9] G. Hetsroni, A. Mosyak, E. Pogrebnyak, L.P. Yarin, “Heat transfer in micro-channels: Comparision of experiment with theory and numerical results” IJHMT, 48(2005) 5580-5601

Corresponding Author

Abhinandan Kumar*

PG Scholar (Department of Mechanical Engineering) GCOE, Karad

E-Mail – er.akanand12@gmail.com