CFD Simulation of Compact Type Mini Channel Heat Exchanger by Using of Water

P.J.Pan[1] N.S.Patel[2]
[1]Student [2]Guide
[1,2]Government Engineering College, Dahod, Gujarat Technological University

Abstract—A heat exchanger is a device that is used to transfer thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. In the conventional heat exchangers, pipes are larger in size which makes heat exchanger bulky. But in some typical applications such as closed loop gas turbine heat exchangers, cryogenic applications, heat exchangers used in PWR power plants, nuclear submarines, etc., size and weight are critical design constraints on the heat exchanger. Also for high pressure applications tubes are subjected to high bending stresses. To overcome these difficulties, compact heat exchangers can be employed. Mini channels heat exchanger is a type of compact heat exchanger in which mini channels are machined on metal plates and then such plates are bonded together. Such an arrangement provides high strength so that it can be used for high pressure applications. It has been observed that in mini channels, convective heat transfer coefficient is more than the tubes used in conventional heat exchangers. Hence, the length for same heat transfer is greatly reduced which results in reduced overall size and weight of the heat exchanger. Electronic device are in heavy demand for computer processor applications and generate large amount of heat. These high power device can be cooled off very effectively by either liquid or gas coolant flowing through micro or mini channels. Continuous research work is ongoing for developing high speed processor which generate high amount of heat. Cooling of such particular systems requires high amount of mass flow rate and compactness is also required. In present work, a mini channel heat exchanger is designed with assuming inlet and outlet of hot temperature, inlet of cold water temperature and also the mass flow rates of cold and hot water. This compact heat exchanger can be used for cooling purpose of electronics device like silicon chip which would be used for microprocessor. In order to cool down silicon chip, it is kept in place of hot fluid plate. Cooling of silicon chip is required to prevent from damage and subsequently failure. The heat exchanger is designed for obtaining 10°C of hot fluid.

I. INTRODUCTION

Micro channels and Mini channels are found in many biological systems providing very high heat and mass transfer rates in organs such as the brain, lung, liver and kidney. Many high flux cooling applications are effectively utilizing their high heat transfer capabilities of these channels. Small channel diameters are at the heart of all biological systems. Fluid flow and mass transfer in the human body, for example, utilize the high heat and mass transfer coefficients associated with micro channels. The potential of micro channels in high heat flux removal application was first brought to our attention by the pioneering work of Tuckerman and Pease (1982). A broad historical perspective of micro channel and mini channel development was presented by Kandlikar and Grande . [1]

Channel Classification Based on Channel Hydraulic Diameter:

- Conventional Channels: $D_h > 3$ mm
- Mini channels: 3 mm $\geq D_h >$ 200 µm
- Micro channels: 200 µm $\geq D_h >$ 10 µm
- Transitional Channels:10 µm $\geq D_h >$ 0.1 µm
- Transitional Micro channels:10 µm $\geq D_h >$ 1 µm
- Transitional Nano channels:1 µm $\geq D_h >$ 0.1 µm
- Molecular Nano channels: 0.1 µm $\geq D_h$

Paisarn Naphon and Osod Khonseur performed experiments to investigate the heat transfer characteristics and pressure drop in the micro-channel heat sinks under constant heat flux conditions. The experiments were performed for the Reynolds number and heat flux in the ranges of 200–1000 and 1.80–5.40 KW/m^2, respectively. The micro-channel heat sink with two different channel heights and two different channel widths are accomplished by wire electrical discharge machine. Higher heat transfer rate as air mass flow rate increases. However, increase of heat transfer rate is less than that of air mass flow rate. Therefore, outlet air temperature tends to decrease as air mass flow rate increases. Higher surface area and surface roughness result in increase heat transfer rate, therefore, the outlet air temperatures of channel with of w=0.2 mm are higher than those of w=0.3 mm.The heat transfer rate depends on the cooling capacity rate of air. Therefore, the average heat sink temperature decreases as air Reynolds number increases. Due to higher surface area and surface roughness, the heat transfer rate from the heat sink surface to the cooling air increases. Therefore, the increase channel height results in lower heat sink temperatures. The heat transfer coefficient increases with increasing heat flux. Due to higher temperature difference between inlet air temperature and heat sink temperature, higher heat flux gives heat transfer coefficient higher than those lower ones. The heat transfer coefficients significant increase with increasing air Reynolds number. With the research and development of the miniaturized technologies, mini and micro-channel cooling systems have been widely used in the electronic devices [2]

Thanhtrung Dang et al reported that their study indicated that the heat transfer rate obtained from micro channel exchanger was higher than those obtained from the mini channel heat exchangers; however, the pressure drops obtained from the micro channel heat exchanger were also higher than those obtained from the mini channel heat exchangers. As a result, the micro channel heat exchanger should be selected for the systems where high heat transfer rates are needed. In addition, at the same average velocity of water in the channels used in this study, the effectiveness obtained from the micro channel heat exchanger was 1.2–

1.53 times of that obtained from the mini channel heat exchanger. Micro channel and mini channel heat exchangers have been used in various fields of engineering and scientific applications. Heat transfer rates in these micro channels and mini channels are higher than those in conventional channels; as a result, for a given heat transfer capacity, mini channel heat exchangers are more compact and lighter in weight. The experimental system consists of three major components: the test section (the micro channel or mini channel heat exchanger), syringe system, and overall testing loop, as shown in Figure 2.2. One micro channel and two mini channel heat exchangers were designed and tested in this study. The three heat exchangers have the same total cross-sectional area of 1 mm^2 for all channels involved. That implies that the average velocity in the channels is the same. These heat exchangers can be used to cool electronic devices or for other cooling applications. The heat transfer process of these devices is carried out between two liquids which are hot water and cold water; the hot and cold fluids are flowing in the opposite directions. Figure 2.3 shows the dimensions of the test samples. The material for the heat exchanger is aluminium, used as a substrate with thermal conductivity of 237 W/mK, density of 2,700 kg/m^3, and specific heat at constant pressure of 904 J/kg K. The thickness of these substrates is 2 mm.[3]

X. L. Xie el al reported that with the rapid development of the information technology (IT) industry, the heat flux in integrated circuit (IC) chips cooled by air has almost reached its limit of about 100 W/cm2. The micro channel flow geometry offers large surface area of heat transfer and a high convective heat transfer coefficient. However, it has been hard to implement because of its very high pressure head required to pump the coolant fluid though the channels. A normal channel could not give high heat flux although the pressure drop is very small. A mini channel can be used in heat sink with a quite high heat flux and a mild pressure loss. A mini channel heat sink with bottom size of 20 mm x 20 mm is analysed numerically for the single-phase laminar flow of water as coolant through small hydraulic diameters and a constant heat flux boundary condition is assumed. The effects of channel dimensions, channel wall thickness, bottom thickness and inlet velocity on the pressure drop, thermal resistance and the maximum allowable heat flux are presented. The results indicate that a narrow and deep channel with thin bottom thickness and relatively thin channel wall thickness results in improved heat transfer performance with a relatively high but acceptable pressure drop. A nearly-optimized configuration of heat sink is found which can cool a chip with heat flux of 256 W/cm2 at the pumping power of 0.205 W. They concluded that heat transfer, a narrow and deep channel is better than that of a wide and shallow channel, in spite of the high pressure drop penalty. A nearly-optimized configuration is obtained for the heat sink with bottom size of 20 mm x 20 mm. For this heat sink the maximum heat flux reaches about 256W/cm2 under the constraint of temperature difference 50 K with inlet velocity of 1.5 m/s, the corresponding thermal resistance is 0.0488 K/W and pumping power is 0.205 W. Even at lower inlet velocity of 0.1 m/s, the thermal performance is also quite good with thermal resistance of 0.091 K/W, pumping power of 5.3 $\times 10^{-4}$ W and the maximum heat flux of 137 W/cm^2. The orthogonal analysis has verified the nearly-optimized configuration to be the best one in the research ranges of parameters [4]

Shung et al reported that the analytical results show that the average temperature of the hot and cold side flow significantly affects the heat transfer rate and the pressure drop at the same effectiveness. Different effectiveness has a great influence upon the heat transfer rate and pressure drop. When the micro heat exchanger material is changed from silicon to copper, the thermal conductivity changes from 148 to 400 W/m K. The heat exchanger efficiency is also similar. Therefore, the (110) orientation silicon based micro heat exchanger made using the MEMS fabrication process is feasible and efficient. Furthermore, the dimensions effect has a great influence upon the relationship between the heat transfer rate and pressure drop. Under the same effectiveness, the heat transfer rate increases with rising working fluid temperature in the hot and the cold flow side. The pressure drop decreases because of the temperature influence, especially on the cold flow side. And the higher average temperature situation has the larger heat transfer rate. Under the different effectiveness, the heat transfer rate and pressure drop decrease with the increase in effectiveness. Contrasting the increasing magnitude of the pressure drop, the cold flow side is larger than the hot flow side. At this time, the better heat transfer rate is in the low effectiveness situation. Although the thermal conductivity of the copper k = 400 W/m K exceeds the silicon k = 148 W/m k by two times, the influence is very small for a micro heat exchanger. This is because the fin thickness between the hot and the cold flow channels is very thin, and the thermal resistance of the fin is very small. Therefore, it reduces the influence of the material thermal resistance in the micro heat exchanger. With the dimensions enlarged two times and the outer dimensions enlarged two times, there are advantages and disadvantages in the pressure drop and the heat transfer rate. The designer can choose the appropriate plan by their requirement [5]

M. Thirumarimurugan, T.Kannadasan and E.Ramasamy have investigated heat transfer study on a solvent and solution by using Shell and Tube Heat Exchanger. In which Steam is taken as the hot fluid and Water and acetic acid-Water miscible solution taken as cold fluid. A series of runs were made between steam and water, steam and Acetic acid solution. Experimental results such as exchanger effectiveness, overall heat transfer coefficients were calculated. [6]

Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan,Kanwar Naveed Ahmad, Sarfaraz Khani**,** focused on the applications of Computational (Fluid Dynamics (CFD) in the field of heat exchangers. It has been found that CFD employed for the fluid flow mal-distribution, fouling, pressure drop and thermal analysis in the design and optimization phase. Different turbulence models such as standard, realizable and RNG, k – ε, RSM, and SST k - ε with velocity-pressure coupling schemes such as SIMPLE, SIMPLEC, PISO and etc. have been adopted to carry out the simulations. Conventional methods used for the design and development of Heat Exchangers are expensive. CFD provides cost effective alternative, speedy solution and eliminate the need of prototype, it is limited to Plate, Shell and Tube, Vertical

Mantle, Compact and Printed Circuit Board Exchangers but also flexible enough to predict the fluid flow behavior to complete heat exchanger design and optimization involving a wide range of turbulence models and Integrating schemes the k - ε turbulence model is most widely employed design and optimization .The simulations results ranging from 2% to 10% with the experimental studies. In some exceptional cases, it varies to 36%. [7]

In this paper Sharad Kumar *et al.* presents the effect of arc shaped geometry on heat transfer co-efficient, friction factor and performance enhancement were investigated covering the range of roughness parameter (from 0.0299 to 0.0426) and working parameter (Re from 6000 to 18000).Different turbulent models have been used for the analysis and their results were compared. Re normalization group (RNG) κ- ε model based results have been found in good agreement and accordingly this model was used to predict heat transfer and friction factor in the duct [8]

Kadir bilen *et al.* Experimentally investigated the surface heat transfer and friction characteristics of a fully developed turbulent air flow in different grooved tubes. Tests were performed for Reynolds number range of 10000 to 38000 and for different geometric grooved shape (circular, trapezoidal and rectangular) tubes [9]

II. DESIGN PROCEDURE

ε - NTU approach is used for designing of this counter-flow heat exchanger. The purpose of this heat exchanger is to reduce the temperature of incoming hot fluid and deliver it at a lower temperature. So inlet and outlet temperature of hot fluid and inlet temperature of cold fluid is taken as reference. Moreover since cold water is available in abundance Cold-Hot-Cold configuration is adopted so that hot water gets cooled faster. Cross-section dimensions of each channel are taken 3mm ×1mm. Other design parameters are given in Table I

Table I
Design Parameter

	Hot fluid	Cold fluid
Mass flow rate (Kg/s)	0.15	0.1
Inlet temperature (°C)	80	25
Outlet temperature (°C)	70	-

Consider a small element of heat exchanger of length 1 mm from the entry of the cold fluid. For one module of C-H-C configuration length of hot and cold fluid channels is 1 mm only so properties of fluid are considered to be constant for this length. These constant properties are calculated at known temperature of both fluids (i.e. for cold fluid, temperature of fluid entering in the segment and for hot fluid, temperature of fluid leaving the segment).ε-NTU approach is used to calculate cold fluid outlet temperature and hot fluid inlet temperature for first segment The outlet temperature of cold fluid for first segment is the inlet temperature for the next cold segment and inlet temperature of hot fluid for first segment is the outlet temperature for the next hot segment. This iteration goes on till 80˚C temperature of hot water at inlet is obtained A program in C to perform these iterative calculations. The program terminates when desired hot inlet temperature is achieved. Thus the total length of the heat exchanger is obtained. By the output of C program, the length of Heat Exchanger will be obtained as 394mm.By using design parameter, the plate is modeled by using pro-E wildfire 5.0 as shown in below

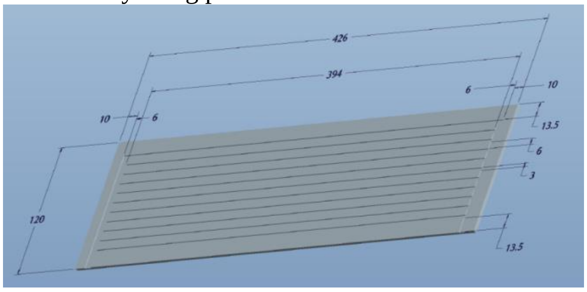

Fig. 1: 3D model of cold plate and hot plate

In Establishing of assembly model of Heat exchanger three cold plates, six hot plates and one flat plate will be used. The Cold plates and Hot plates are arranged into opposition orientation with respect to each other to build up counter flow of Cold and Hot fluid within Heat exchanger. This is shown in figure below

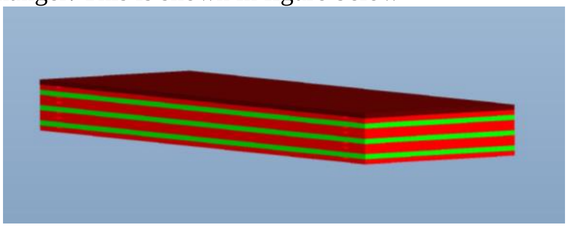

Fig. 2: 3D assembly model of heat exchanger

Fig. 3: Experimental set up

A. *Procedure*

Fill both tanks with water. Using temperature controller set temperature to 85˚C, wait till 80 ˚C temperatures is attained at hot water inlet temperature indicator. This will take some time since there are losses in the system before the temperature sensor. There is no need of priming hot water pump as it is self-priming. Start hot water pump and using valve set the mass flow rate to 600 lph. i.e. 0.16 kg/keep the valve of cold water supply closed and prime the pump as cold water pump is not self-priming. Start cold water pump and using valve set the mass flow rate to 400 lph. i.e. 0.11 kg/measure pressure and temperature of both hot water and cold water at inlet and outlet. During this measurement, we did not fairly concentrate on pressure measurement; we just concentrated on temperature measurement. During performance near designed conditions leakages started at some points in the pipes and on the lateral surfaces of the heat exchanger. So the experiment had to be stopped and the leakages had to be sealed at those points. Since leakages are more at higher mass flow rates and high temperatures, the heat exchanger was operated at off design conditions, i.e. at lower mass flow rate upto 300 lph at temperature 50 °C.

B. *Off-Design Performance*

Set the hot water temperature at 55°C to get 50°C temperature at hot water inlet.. Start the hot water pump and using valve set hot water mass flow rate to 200 lph. Now start cold water pump after priming and set cold water mass flow rate to 200, 250, 300 for same mass flow rate of hot water as 200 lph Measure temperature of both hot water and cold water at inlet and outlet. Repeat the same procedure for hot flow rate 250, 300, 350 lph.

Table Ii
Experimental Reading

The concerns were created during taking reading at design condition. So reading performance was taken at off – design

SR	Hot fluid				cold fluid			
	m˚ in lph	.Thi In°C	Tho In°C	ΔT	m˚In lph	Tci In °C	Tco In °C	ΔT
1	200	50	42	8	200	31	39	8
2	200	50	41	9	250	31	39	8
3	200	50	41	9	300	31	38	7
4	250	50	41	9	200	31	41	10
5	250	50	40	10	250	31	41	10
6	250	50	40	10	300	31	40	9
7	300	50	40	10	200	31	43	12
8	300	50	39	11	250	31	42	11
9	300	50	39	11	300	31	42	11

condition. Here we can show that hot temperature is equal to nearby 20°C instead of 10°C. This was happing because of setting different performance parameter from the design parameters like as mass flow rate of hot fluid and cold fluid (200,250,300lph), inlet temperature of hot fluid and cold fluid as 60°C and 31°C respectively In order to verify that this reading is validated for predetermined design of each hot and cold plate and subsequently arrangement of all these plates, CFD simulation will be required to do base on these performance parameters. If CFD results and experimental readings at off – design condition are being got nearly equal then we can say that this experimental set up is established as per our desires.

Table Iii
Cfd Results

SR	Hot fluid				cold fluid			
	m°	Thi	Tho	ΔT	m°	Tci	Tco	ΔT
1	200	323	313	10	200	305	315	10
2	200	323	313	10	250	305	313	7
3	200	323	313	10	300	305	311	6
4	250	323	315	8	200	305	314	9
5	250	323	313	10	250	305	316	1
6	250	323	314	10	300	305	312	7
7	300	323	316	7	200	305	314	9
8	300	323	315	8	250	305	314	9
9	300	323	313	10	300	305	316	10

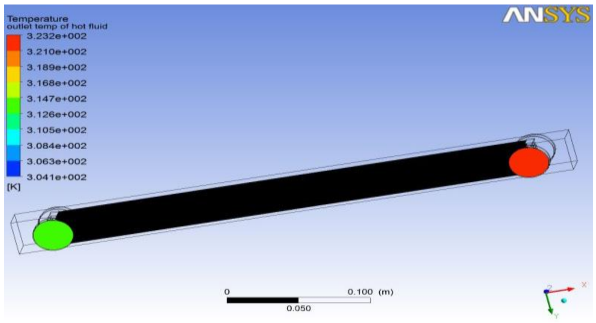

Fig. 3: Inlet and outlet temperature contour of hot fluid

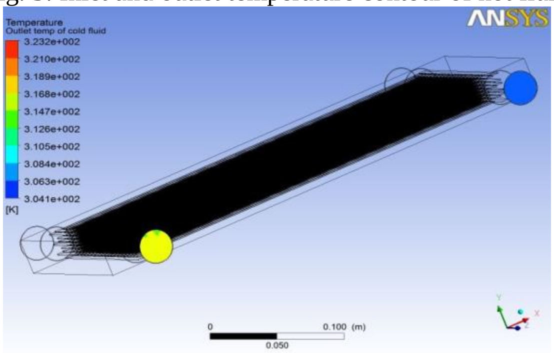

Fig. 4: Inlet and outlet temperature contour of cold fluid

Table Iv
Comparion Between Experime NTAL And CFD

SR	Hot fluid				Cold fluid			
	m°	ΔT Exp.	ΔT CFD	Error In %	m°	ΔT Exp.	ΔT CFD	Error In %
1	200	323	313	10	200	305	315	10
2	200	323	313	10	250	305	313	7
3	200	323	313	10	300	305	311	6
4	250	323	315	8	200	305	314	9
5	250	323	313	10	250	305	316	1
6	250	323	314	10	300	305	312	7
7	300	323	316	7	200	305	314	9
8	300	323	315	8	250	305	314	9
9	300	323	313	10	300	305	316	10

ACKNOLEGMENT

I would like to sincere thank my guide Akash pandeysir and N.S.Patelsir to contribute their efforts to complete my research work. As well as my friends those have been provided directly and indirectly consultant me for my research work

REFERENCE

[1] Kandlikar S.G. and Mark E. Steinke "Examples of micro channels mass transfer processes in Biological

system", first international conference on micro channels and mini channels April 24 -25 , 2003.
[2] Paisam Naphon and osod Khonseur "study on the convective Heat transfer and pressure drop in the micro channel heat sink", International communication in Heat and mass transfer.
[3] Thanhtrung Dang and Juhn – Tong Teng " The effects of configurations on the performance of Micro channel counter flow Heat exchanger – An experimental study"
[4] X.L.Xie.el.at. "Numerical study of laminar heat transfer and pressure drop characteristics in A water – cooled mini channel heat sink ", science direct applied thermal engineering 29(2009)
[5] Shung Wen Kang et. Al,"Analysis of effectiveness and pressure drop in micro cross flow heat Exchanger "science direct applied thermal engineering27(2007).
[6] CFD Analysis of shell and tube heat exchanger – A review, International journal of engineering Science and innovative technology , volume 2 , issue 2, March 2013.
[7] .Muhammad Mahmood , Aslam Bhutta , " Numerical Analysis if fin tube plate heat exchanger By using CFD Technique " ARPN journal of engineering and Applied Science Volume 6 , No7 July 2011 , ISSN 1819 – 6608
[8] Kabir Bilen et. al. , sharad Kumar et.al. "CFD analysis of Heat transfer and friction factor characteristics of a Turbulent flow for Internally grooved tubes ", National Institute of Technology , Tiruchirapalli - 620
[9] Mark A Pierson, Srinath V Ekkad, Robert Hendricks, "Development of a Mini Channel Compact Primary Heat Exchanger for a molten salt reactor" , April 28, 2011.

