Experimental and Numerical Investigation of Forced Convection Heat Transfer in an Optimized Microchannel Heat Sink

V.B. Swami*, D.A. Kamble and Dr. B.S. Gawali*****
**Assistant Professor Mechanical Engineering Dept. WCE Sangli
***Associate Professor and Head Mechanical Engineering Dept. PDEA'S COE Manjari Pune
***Professor and Head Mechanical Engineering Dept. WCE Sangli

ABSTRACT
In this study, the experimental and numerical investigation of forced convective heat transfer in an optimized rectangular micro-channel heat sink (MCHS) is analyzed using water as the working fluid. The optimization of MCHS has carried out by using CVM (Constant Volume Method) and VVM (Variable Volume Method). From the optimization, it is clear that aspect ratio three which gives the better heat transfer coefficient and lower pressure drop values as comparing with adjacent aspect ratios. A numerical investigation of MCHS based on the finite volume method. The numerical results are validated by comparing the predictions with analytical solutions and experimental data. High heat flux and high coefficient of heat transfer obtained by reducing the core volume of the channel and changing operating conditions. The Optimized micro-channels geometry has a width of 700 μm and a depth of 2100 μm, and is separated by a 350 μm wall. The microchannels are made on cooper plate by using a wire Electro Discharge Machining. The pressure drop and heat transfer investigation is carried out for various Reynolds Number. The obtained results are also compared with the correlation available in literature and presented in the paper.

Keywords: Microchannel heat sink, optimization, CVM, VVM.

NOMENCLATURE

W_c	Width of the microchannel
H_c	Height of the microchannel
S	Fin thickness of the microchannel
α	Aspect ratio = H_c/W_c
ΔP_{thero}	Theoretical pressure drop
ρ	Density of water,
f	Friction factor
L	Length of the microchannel heat sink
V	Mean velocity.
D_h	Hydraulic diameter = 4Area/Perimeter
Re	Reynolds number
Nu	Nussult number.

1. INTRODUCTION

Microchannel heat-sinks are appealing solutions to the increasing heat dissipation demands, especially in electronic devices. Multiple microchannels are machined on the back of the substrates of electronic components in integrated circuits removes heat at faster rate. The heat generated by the electronic component is transferred to the coolant by forced convection. Therefore, heat sinks with larger extended surfaces, highly conductive materials and more coolant flow are keys to reduce the temperature. High heat transfer rate is the need of many systems in today's world in order to improve their performance by maintaining their operating temperature below acceptable levels as in case of high performance computer chips is below 100°C, microelectronic equipments. This can be achieved by both direct geometry advantage of "higher heat transfer area" and "higher heat transfer coefficient" (Shah and Landon). As the computer industry continues to make rapid advances in processor speed, thermal management plays a more important role. The first concept for using microchannels as a heat exchanger device for VLSI circuits was conceived by Tuckerman and Pease (1981) [1]. It was found that the microchannels are very effective in heat dissipation. The reason for very high heat removal was associated with the fact that microchannels have very small hydraulic diameters ranging upto a few hundred microns. This was in conjunction with the significant area enhancement due to the finned surface. In this work, 20 microchannels in parallel are machined on copper plate having width (W_c) and depth (H_c) of 700 μm and 2100 μm respectively, fluid under study is water. The copper plate is heated electrically. As the channel hydraulic diameter is reduced the corresponding pressure drop will be very high. The channel dimensions W_c and H_c, the fin thickness (S), and the coolant flow rate are the parameters of interest in designing a microchannel heat sink.

2. DESIGN AND OPTIMIZATION FOR PRESSURE DROP ANALYSIS OF MICROCHANNEL HEAT SINK GEOMETRY

2.1. Introduction

The design and optimization of microchannel passages in a direct heat sink is important from an operational standpoint. Pressure drop considerations will further determine the pumping power required and the operating pressure to which the chips will be subjected. The heat transfer problem in case of small channel is much more dependent on the flow conditions. i.e., developed velocity profile or developing velocity profile as well as on the wall thermal boundary conditions. Therefore it is always necessary to ensure the flow conditions in the channels. For this purpose the experimental setup is designed to measure pressure drop through the channels as well as across the whole geometry to understand the effect of manifolds. This will give a direct estimate of the flow conditions [2]. There are some basic types of flow which play important role in designing the microchannel heat sink with optimal geometrical parameters. Liquid coolant is usually used, and the flow regime is usually single-phase and laminar. The optimal geometric parameters of the channels of a Microchannel Heat Sink (MCHS) are determined using the analytical approach of Kwasi et al [3]. Kwasi reduced the analysis of the microchannel flow problem to a quasi two-dimensional differential equation and presented exact solutions to analytically determine the optimal dimensions of microchannels under given constraints. Based on the given constraints such as pumping power and space limitation the variables to be optimized are the channel width, aspect ratio and channel spacing. The optimal aspect ratio of the MCHS channels, subject to the constraints imposed. The operating geometrical parameters of the channels of microchannel heat sink are determined using the two different design scenarios, such as (1) The allowable volume of the heat sink is fixed based on design constraint (i.e. constant volume method) and (2) No limit is placed on the volume (i.e. variable volume method) of microchannel heat sink core, however, the dimensions of microchannel heat sink are within the limits.

The following results are obtained by analysis of these two methods. By applying the constraints, such as lower wall temperature of channel base below 100 ºC with minimal pumping power through channels and space limitations. The variables to be selected for getting good thermal performance of microchannel heat sink are channel width, aspect ratio and channel spacing [3].

2.2. Optimization by constant volume method

In this case, operating height equal to 2.1 mm, operating width equal to 0.7 mm, AR or α equal to 3.

It results in 5.9% increase in pumping power, 5.6% reduction in ratio of heat removed to pumping power and 19% reduction in wall temperature of channel base and 2% increase in heat transfer coefficient when comparing its parameters with previous aspect ratio equal to 2 parameters. In the

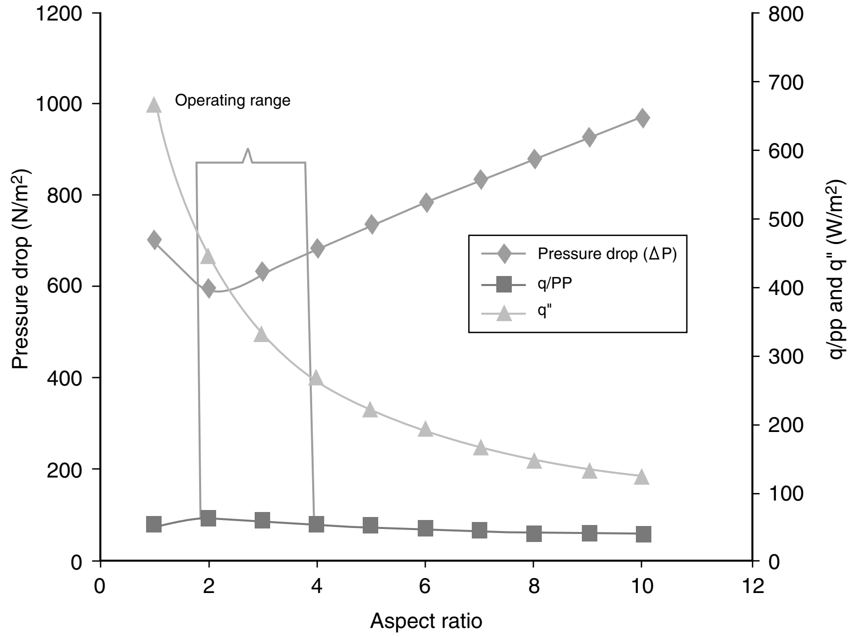

Figure 2.2.1. Effect of Aspect Ratio on Pressure drop, q/pp and q by Constant Volume Method (CVM).

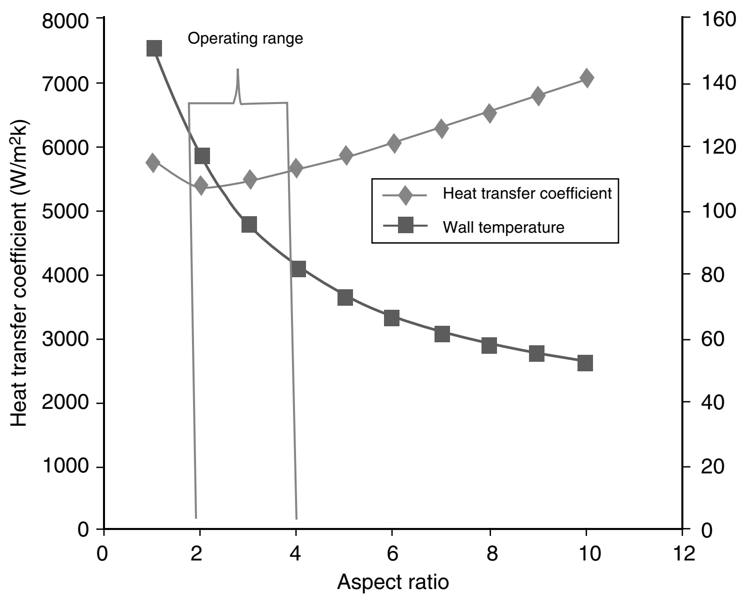

Figure 2.2.1. Effect of Aspect Ratio on Heat Transfer Coefficient and Wall Temperature by Constant Volume Method (CVM).

another case, operating height equal to 2.45 mm, operating width equal to 0.6125 mm, AR or α equal to 4. It results in 8% increase in pumping power, 7.5% reduction in ratio of heat removed to pumping power and 14.3% reduction in wall temperature of channel base and 3.3% increase in heat transfer coefficient when comparing its parameters with previous aspect ratio equal to 3 parameters.

2.3. Optimization by variable volume method

In the first case, operating height equal to 2.1 mm, operating width equal to 0.7 mm, AR or α equal to 3. It results in 4.2% reduction in pumping power, 7.4% gain in ratio of heat removed to pumping power and 11% reduction in wall temperature of channel base when comparing its parameters with previous aspect ratio equal to 2 parameters.

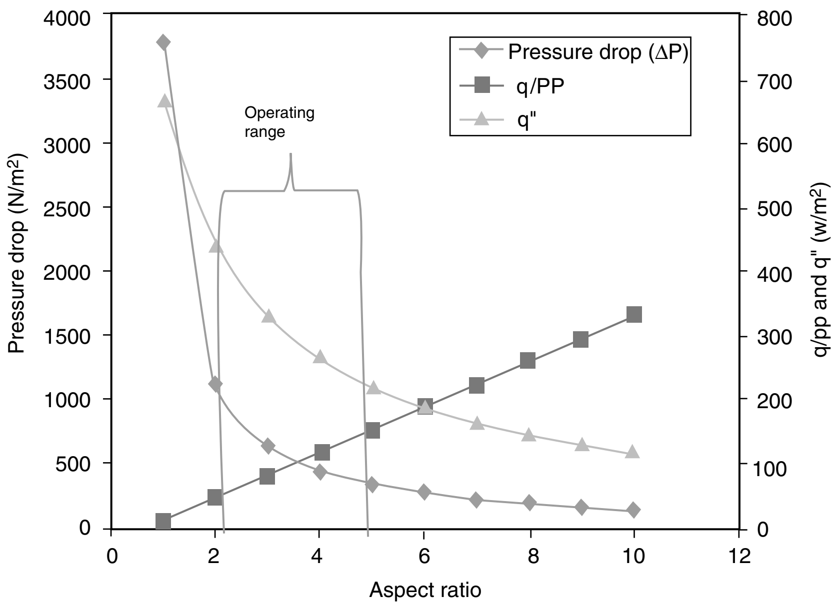

Figure 2.3.1. Effect of Aspect Ratio on Pressure drop, q/pp and q by Variable Volume Method (VVM).

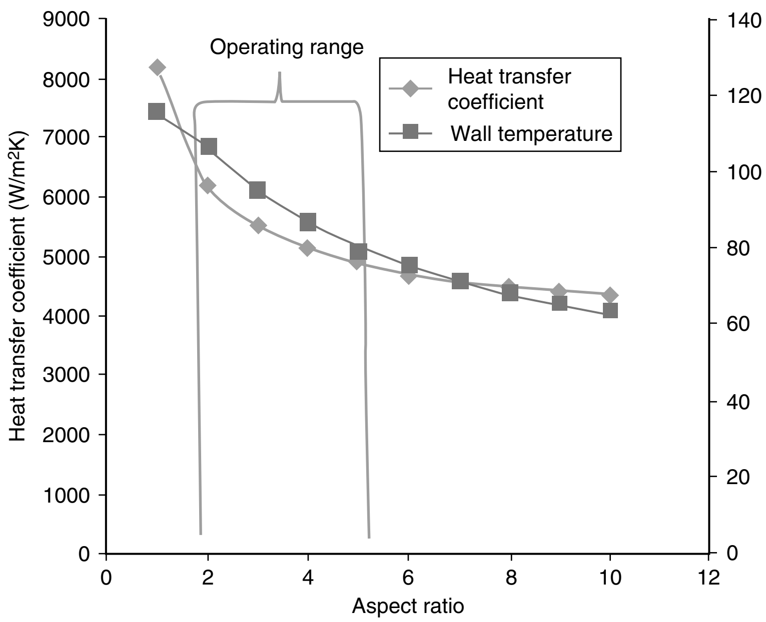

Figure 2.3.2. Effect of Aspect Ratio on Heat Transfer Coefficient and Wall Temperature by Variable Volume Method (VVM).

In the second case, operating height equal to 2.8 mm, operating width equal to 0.7 mm, AR or α equal to 4. It results in 3.0% reduction in pumping power, 4.2% gain in ratio of heat removed to pumping power and 9% reduction in wall temperature of channel base when comparing its parameters with previous aspect ratio equal to 3 parameters.

3. EXPERIMENTAL INVESTIGATION OF FRICTIONAL PRESSURE DROP AND FORCED CONVECTIVE HEAT TRANSFER

In this method experimental setup is designed to measure static pressure drop across the whole geometry between two points located at known distance from each other to understand the fluid flow conditions inside the manifold which is made for microchannel heat sink.

$$\Delta P_{\text{thero.}} = \frac{2\rho f L V^2}{D_h}$$

Input flow rates – 100 ml/min to 2000 ml/min i.e. *Re* equal to 59.5238 to 1190.4762. For measurement of total pressure drop across the whole geometry and also, we can provide two T- arrangements at the inlet and outlet manifold of the geometry and the pressure drop which is measured by the digital inclined manometer.

It is clear from the figure 3.2 that the contribution of manifold design to the total pressures drop across the whole geometry of MCHS is very less. At lower Reynolds number ranges i.e. from 450–690 is varying from 1.05–1.12 times of channel frictional pressure drop across the geometry. These effects are due to the low velocity of fluid at lower Reynolds number. This total pressure drop increases linearly with increase in Reynolds number because the entrance and exit are not considered into these results.

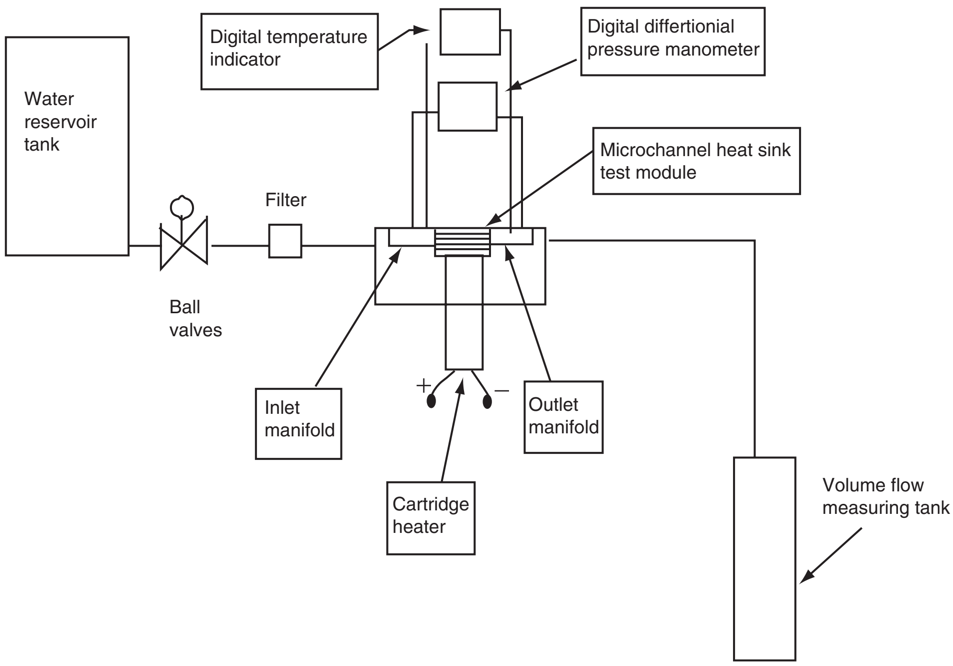

Figure 3.1. Schematic Block Diagram of an Experimental Setup for Pressure Drop and Heat Transfer Analysis.

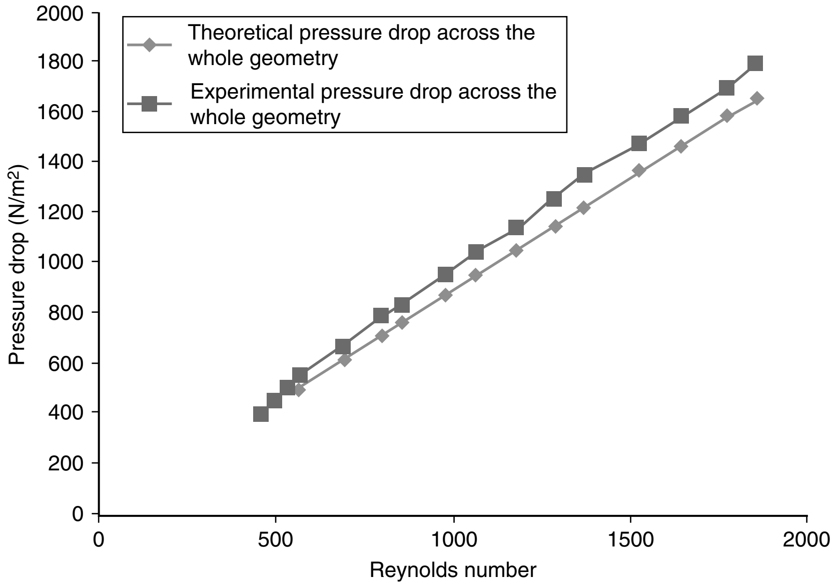

Figure 3.2. Effect of the Reynolds No on the Theoretical Pressure Drop and Experimental Pressure Drop.

4. NUMERICAL INVESTIGATION OF AN OPTIMIZED MICROCHANNEL HEAT SINK GEOMETRY

4.1. Introduction

The CFD calculation reported here is conducted using a commercial general purpose CFD software package Fluent 6.3.26, which solves the three-dimensional form of mass, momentum, and energy equations.

The problem under consideration is a conjugate heat transfer problem which involves heat transfer in both solid and liquid. The governing equations are discretized on a Cartesian grid using the finite volume method. Figure 4.1 shows the computational domain and the boundary conditions.

4.2. Generating solutions with fluent

The steps involved in FLUENT are:

a) Defining the domain / geometry.
b) Meshing/Gridding.
c) Governing equations.
d) Locate Boundary conditions.
e) Select Solution method.
f) Post processing.

The CFD calculation of the entire heat sink with 20 channels is very intensive and time consuming. Since all the channels are geometrically identical and receive the same flow rate and heat flux the calculation domain can be restricted to only one channel. Note that the end effects can be ignored since the heat transfer from the side walls of the heat sink only affects a few channels in each side of the heat sink.

Also, due to the symmetry in the flow and heat transfer across a channel the computational domain can be limited to only single channel and two half of the fin adjacent to the channel. Table 1 summarizes the boundary conditions. The number of control volumes used for all calculations presented here is $100 \times 20 \times 40$ in the x, y, and z.

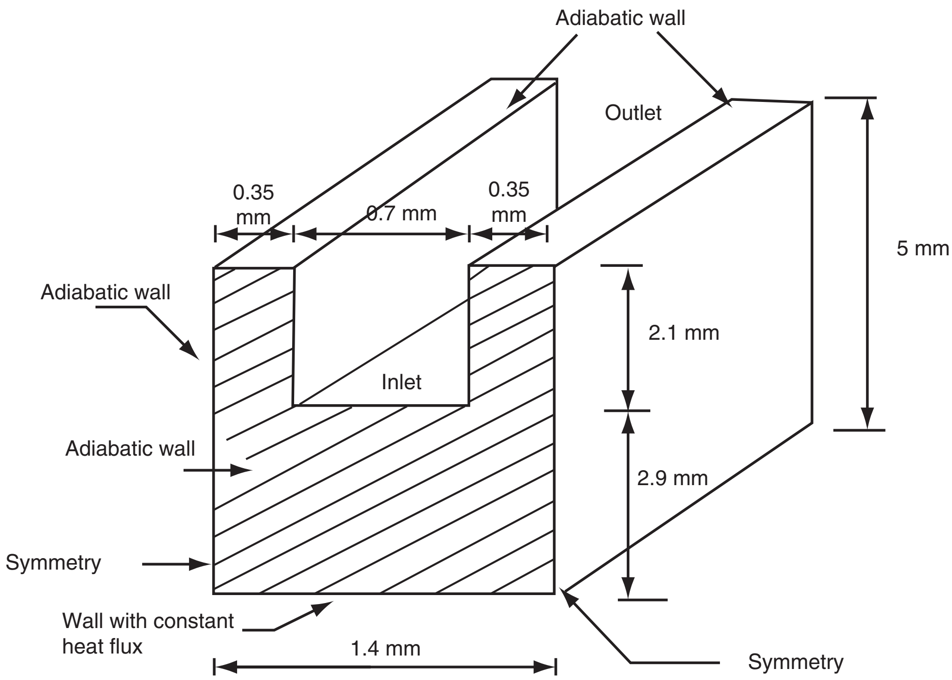

Figure 4.1. Schematic of the Computational Domain and Boundary Conditions for q Boundary Condition.

Table 4.2.1. Flow and thermal boundary conditions.

Boundary	**Flow boundary condition**	**Thermal boundary condition**
Front Inlet	Inlet	Mass flow inlet
Front wall	Wall	Adiabatic
Front outlet	Outlet	Outflow
Back wall	Wall	Adiabatic
Left wall	Wall	Adiabatic
Right wall	Wall	Adiabatic
Bottom wall	Wall	Constant Heat flux
Top wall	Wall	Coupled
Inner wall-Left	Wall	Coupled
Inner wall-Right	Wall	Coupled
Inner wall-Bottom	Wall	Coupled

The CFD simulations are carried out to find the profile of average heat transfer coefficient for all three different flow rates those are 350 ml/min, 550 ml/min and 750 ml/min in thermally developing region and the values are then compared with the experimental results obtained. Figure 4.2.1 shows that as the heat transfer coefficient increases Nusselt number also increases. Simultaneously, it shows comparisons of experimental results and CFD simulation results for three wall heating case that with the result of Harms et al. (1999) [5], for average heat transfer coefficient.

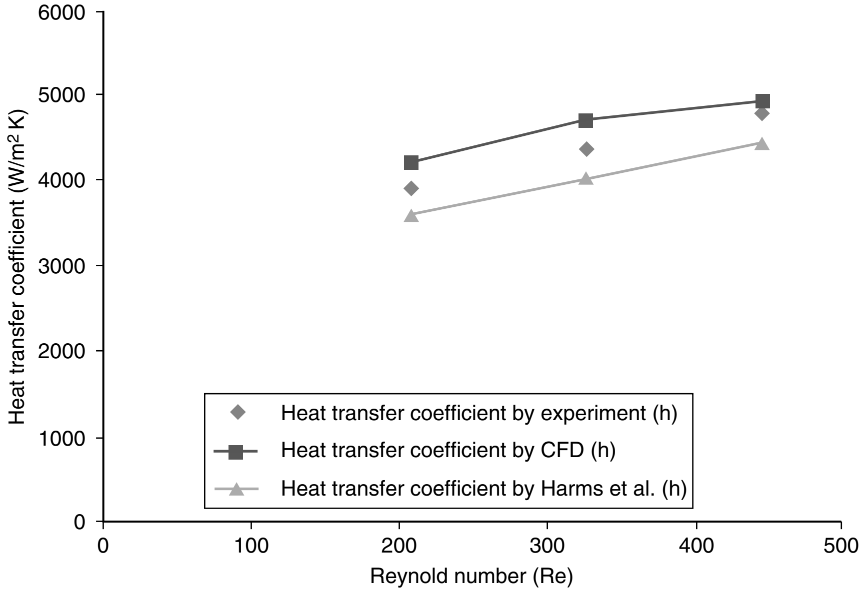

Figure 4.2.1. Comparison Comparisons of Results for Average Heat Transfer Coefficient.

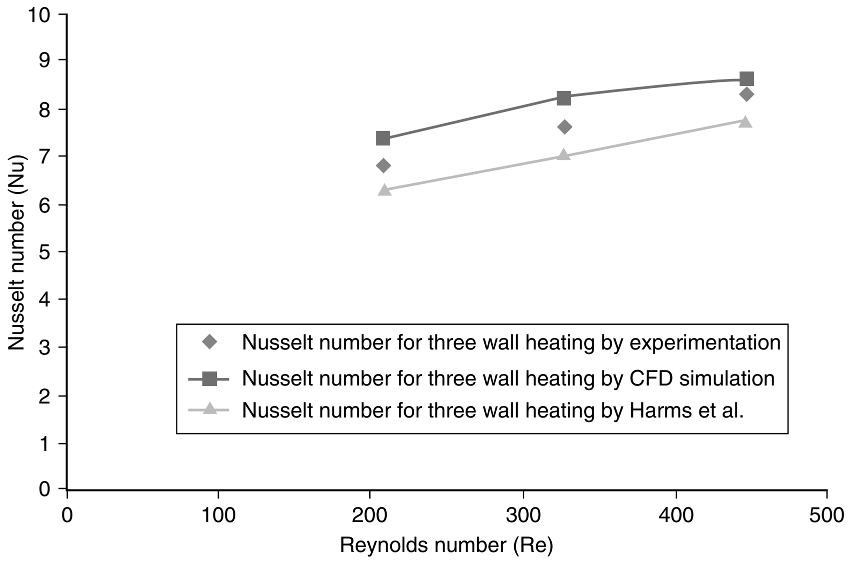

Figure 4.2.2. Comparison Comparisons of Results for Average Nusselt Number.

Looking at the Figure 4.2.2 for the case of thermally developing and hydrodynamically developed flow, as expected the experimental, CFD and analytical Nusselt number increases with the Reynolds number. Clearly, the length of the thermal entrance region increases with an increase in Re, the values in the entrance region are larger than those in the fully developed region, which highlights the critical importance of the entrance region in determining the heat transfer characteristics in microchannel devices.

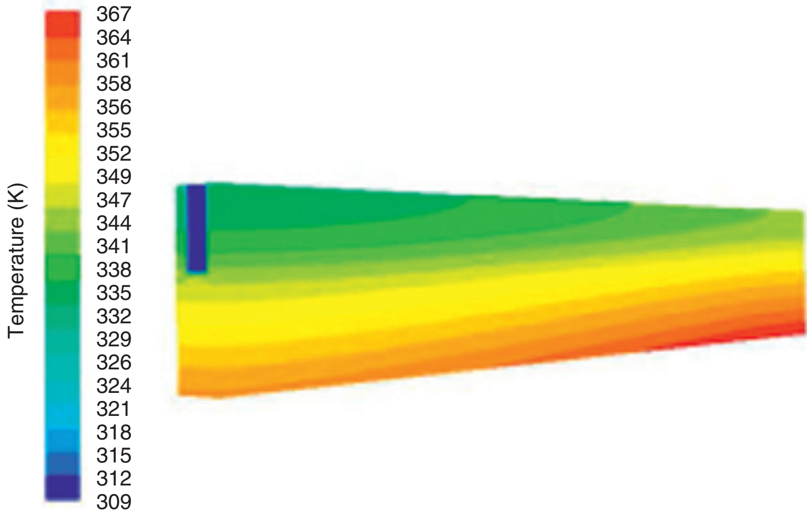

Figure 4.2.3. Computational Fluid Dynamics (CFD) analysis of Microchannel Heat Sink at Re = 446.43.

5. CONCLUSIONS

The geometry obtained from with this method gives high heat flux and lower pressure drop than those obtained with the other approaches. High heat flux and high coefficient of heat transfer obtained by reducing the core volume of the channel and by changing operating conditions. This analysis gives better design, development and manufacturing techniques for microchannel heat sink which gives better results in consideration of the pressure drop analysis and heat transfer analysis. The experimental results have good agreements with the CFD analysis.

6. REFERENCES

[1] Tuckerman, D.B., Pease, 1981, R.F.W., “High performance heat sinking for VLSI”, IEEE, Electron Dev. Lett, 2, 126–129.

[2] Shah and kakac, 2003 “Single Phase Convective Heat Transfer” John Wiley & Sons, New York.

[3] Kwasi Foli, Tatsuya Okabe, Markus Olhofer, Yaochu Jin, Bernhard Sendhoff, 2005 “Optimization of micro heat exchanger: CFD, analytical approach and multi-objective evolutionary algorithms” International Journal of Heat and Mass Transfer 49, 1090–1099.

[4] Shah, R.K., London, A.L., 1978, “Laminar Flow Forced Convection in Ducts”, Academic Press, New York, S.V.

[5] Harms, T. M., Kazmierczak, M.J., Gerner, F.M., 1999, “Developing convective heat transfer in deep rectangular microchannels”, Int. Journal of Heat Fluid Flow, 20,149–157.

[6] Lee, P.S., Garimella, S.V., Dong, L., 2005, “Investigation of heat transfer in rectangular microchannels”, Int. Journal of Heat and Mass Transfer, 48, 1688–1704.

[7] Qu, W., Mudawar, I., 2002, “Experimental and numerical study of pressure drop and heat transfer single-phase microchannel heat sink”, Int. Journal of Heat Mass Transfer, 45, 2549–2565.