MCHS Design Using Fluid Distribution Analysis in Parallel Microchannels

M. N. Mazlan, Y. Husaini
Faculty of Electrical Engineering, Universiti Teknologi Mara (UiTM), 40450 Shah Alam, Selangor, Malaysia

Abstract — The aim of this paper is to address the issue with effective cooling system in microchannel heat sink (MCHS). In conjunction to achieve it, a uniform fluid distribution in parallel channels is required as to achieve the heat distribution optimization. Analysis in computational fluid dynamics (CFD) can be used to check flow distribution in rectangle-shaped parallel microchannels which resulting in good thermal distribution. We manipulate a novel multichannel design model by changing the distance between channels to compare fluid flow uniformity. This study focuses on the laminar flow in rectangular shape channels. It has been found that fluid flow is most uniform when the distance between microchannels is 5.1mm at 0.1 mm/s speed. The fluid flow distribution should be uniform to achieve an optimum cooling performance in the MCHS.

Keywords — rectangular channel, uniform flow distribution, microchannels, velocity distribution.

I. INTRODUCTION

Microchannel heat sink has turn into an essential cooling devices system to very-large-scale integrated circuit and Microelectromechanical system [1]. The interest for quicker circuits and expanded limit, in any case, has prompted to an expansion in power densities what’s more, a requirement for ceaseless change in the techniques for heat removal. An experiment carry out using three-dimensional numerical solution of heat transfer performance also pressure loss to analyse the performance of MCHS heat transfer [2]. The concept initially exhibited the utilization of microchannels for cooling integrated circuits [3]. The heat created by the segment is initially exchanged to the channels by heat conduction and evacuated by the cooling liquid which is forced to course through the channels [4].

Hence, thermal management is needed at every point control dispersal and is included in the operation of any framework [5]. The present PC innovation owes to the scaling down of circuits of silicon chip. The interest for quicker circuits and expanded limit, be that as it may, has prompted to an expansion in power densities and a requirement for persistent change in the strategies for heat evacuation. Microchannel heat sink is known for a phenomenal cooling limit because of the high surface to volume proportion that upgrades the heat evacuation. The temperature appropriation in the dynamic area utilizing Transmission-LineMatrix procedure uncovers that the utilization of microchannels to cool down microchip enhanced thermal resistance conduct and diminished dynamic district temperature in steady state.

The miniature of cooling system in micro sizing is accomplished by integrating parallel microchannels [6]. Thus, achievement on efficient reaction from heat can be done, but must consider care action to ensure that undue fluidic resistance not produce by parallelization, which allow the flow to be maintained. It is also to get optimum heating can be achieve uniformly by each channel. The significant reason that affecting the performance of laminated structured micro-devices in heat and mass transfer efficiency which is the flow uniformity in parallel microchannels [7].

The previous research on heat sink application [8] stated that parallel channel can produce a uniform flow distribution with a triangular inlet plenum and a trapezoidal outlet plenum. Then, using the electrical circuit analogy prediction which is resistance network that equivalent to fluid flow and simulating the impact of distance between channel to the mass flow and temperature distributions in parallel microchannels using numerical methods.

The objective of the present study is to investigate the geometry of MCHS to achieve uniformity fluid flow in the parallel channel by using the parameters different distance between channels and different inlet velocity fluid flow.

II. METHODOLOGY

For the MCHS design, our model based of previous research[9] . The model’s geometry use the non-circular shape which is rectangular shape where the pressure different is relatively small [10]. By using the Hagen-Poiseuille’s law, the rectangular microfluidic channels for liquid fluid flow can be predicted [11]. In this study, four parallel microchannels selected using Stephan’s design [12]. The sizing parallel channel are 0.1 mm deep and 0.5 mm wide.

Next, the distance between these parallel channels being changed to be stimulated in CFD which is from 5.1 mm, 4.1 mm, 3.1 mm and 2.1 mm for these case study. In previous research, the MCHS use water nanofluid as flow coolant for good heat transfer performance [6]. For our MCHS study, by using the fluid water as the fluid flow along the body (density as 997 kg/m3) at 25 °C and inlet velocity selected 10mm/s, 1 mm/s and 0.1 mm/s. In order to get a good uniformity fluid flow in MCHS, the fluid flow must be laminar flow for smooth fluid transfer [13]. For the laminar
flow in rectangular shape, the Reynolds number based on the hydraulic diameter, $D_h$.

$$D_h = \frac{4A_c}{p}$$  \hspace{1cm} (1)

$$Re = \frac{\rho v D_h}{\mu}$$  \hspace{1cm} (2)

where $D_h$ is the hydraulic diameter for rectangular shape, $A_c$ is the cross-sectional area of the pipe and $p$ is wetted perimeter. This value than substitute in the equation 2 to get the Reynolds number to determine whether the flow is laminar or turbulent. For the inlet flow, CFD is used to solve the Navier-Stokes equation for coupled continuity and momentum.

$$\nabla \cdot u = 0$$  \hspace{1cm} (3)

$$\rho \left( \frac{du}{dt} \right) = -\nabla p + \mu \nabla^2 u$$  \hspace{1cm} (4)

where $u$ is to indicate the velocity vector. For assumption, it is use steady, incompressible, laminar flow with constant parameter. The gravity is negligible. Microchannel heat sink performance in term of heat distribution, flow velocity in the channel can be analyse by using CFD technique by using computer software which is ANSYS tools, so we can create a model to simulate the flow performance and heat distribution along the parallel channel based on the microchannel heat sink design.

CFD analysis process includes problem statement which is information about the flow, mathematical model, mesh generation for nodes/cells time instants of the objects and other process. By using the commercial CFD solver ANSYS CFX, laminar flow, rectangular and AIN-based MCHS with water coolant has been designed and upgraded [14]. All this process can perform on MCHS. A researcher from Honda Research Institute enhanced microchannels by CFD (computational fluid dynamics) [15]. In CFD calculations, there are three main steps which is pre-processing, solver execution and post-processing.

In figure 1, at pre-processing phase, the geometry model using [16] as reference. The main objective of study is changing the value of distance between channel at $X_1$, $X_2$ and $X_3$ to analyse the output of flow uniformity as shown in figure 2. Next, the boundary conditions were set in solver setting and the parameters set in table 1.

TABLE I. PARAMETERS SET FOR BOUNDARY CONDITIONS

<table>
<thead>
<tr>
<th>Boundary conditions</th>
<th>Parameters set</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet velocity</td>
<td>10, 1 and 0.1 mm/s</td>
</tr>
<tr>
<td>Outlet pressure</td>
<td>0 Pa</td>
</tr>
<tr>
<td>Wall</td>
<td>No slip wall</td>
</tr>
<tr>
<td>Fluid material</td>
<td>Water</td>
</tr>
<tr>
<td>Flow</td>
<td>Laminar flow at 1 atm</td>
</tr>
</tbody>
</table>

Fig 1   Steps for CFD modelling flowchart.

Fig 2   MCHS geometry model.
The mesh independent study performed to make a comparison at which value of element produces same average velocity. The elements were set for three, five, seven and nine in every parallel channel as shown in figure 3. So, the next element we can assume the result simulation is same. For this case of study, number of element seven selected, for further analysis. The mesh independent study shown on figure 4.

III. RESULT AND DISCUSSION

The uniformity performance of fluid flow represented in term of $\alpha$ ($U_c/U_m$) which is unitless. In figure 3, the graph shown that by decreasing the inlet velocity parameters value from 10 mm/s to 1 mm/s and 0.1 mm/s, the uniformity fluid flow has worsened at 10 and 1 mm/s inlet velocity. But the uniformity performance improves using inlet velocity 0.1 mm/s. The uniformity fluid flow in parallel channel changes drastically based on decreasing the length between channels from 5.1 mm to 4.1 mm, 3.1 mm and 2.1 mm at channel 2 which is high velocity through it compared to channel 1, 3 and 4. This condition is not showing a good uniformity fluid flow trend.

By referring this graph, we can conclude that the best uniformity fluid flow occur at 5.1 mm distance between channels which is the $\alpha$ value is maintained and not exceeding value of one for every inlet velocity changes from 10 mm/s to 1 mm/s and 0.1 mm/s. Based on previous research [9], by using rectangular channel model and considered power law manifold shape design for uniform flow control, the result produced on power law manifold design is equivalent to our model that using variable 5.1 mm distance between channel which is the uniform fluid flow is optimal performance. Other variable that we use which is 4.1 mm until 2.1 mm is not produced a good uniformity fluid flow. For this result, the graph trend produces not exactly follow the expected result which is uniformity fluid flow
performance should increase by decreasing the distance between channels from 5.1 mm until 2.1 mm.

IV. CONCLUSION

The uniformity fluid flow distribution performance is important to produce an efficient cooling mechanism for electronic devices like piping system, heat sink for engineering devices and other cooling application.

The objective of achieving uniform fluid flow distribution in parallel channel by changing the distance between channels is reached. The study shows that the changes of inlet velocity influence the uniformity of parallel channel with different distance between one channel to another. Ideal uniformity flow performance occurs at distance between channel 5.1 mm.

ACKNOWLEDGMENT

The author would like to thank Faculty of Electrical Engineering UiTM to enable this project as a final year project.

REFERENCES


