INVESTIGATION OF HEAT TRANSFER IN A MICROCHANNEL HEAT SINK
USING WATER AND Al₂O₃ NANOFIUIDS

AZLIZUL AIZAT BIN RAZALI

A project report submitted in partial fulfillment of
the requirement for the award of the
Degree of Master of Mechanical Engineering

Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia

JULY 2017
ACKNOWLEDGEMENTS

I would like to express my sincere gratitude to Dr. Azmahani binti Sadikin for her benevolent guidance and worthy suggestion during the course of this thesis. The valuable research experienced gained during this thesis would not have been possible without her encouragement and support. I would also like to thank my co-supervisor, Dr. Siti Aida binti Ibrahim for her support and guidance. The technical discussions with them were always been very insightful and I will always be indebted to their knowledge shared with me.

I would like to thank the faculty and staffs of Thermodynamics Laboratory, Materials Science Laboratory also with Microelectronics and Nanotechnology Research Centre (MiNT-SRC) for helping me in develop inclination towards research. I would also like to thank my colleagues for extending their help and sharing knowledge in this field of study. Last but not least, I express my sincere gratitude to my family for all their support and presence at time when I needed it the most.
ABSTRACT

Microchannel heat sink is a small device that creates an innovative cooling technology to remove large amount of heat from small area. The dimensions ranging are from 10 μm to 1000 μm and commonly, it has been used for electronic cooling. However, there was limit confined by manufacturing technology and on conduction inside solid material to created small dimension of channel. Therefore present study investigates heat transfer in a microchannel using water and Al₂O₃ nanofluids as working fluids in a square microchannel. The investigation has been carried out by experimental and CFD simulation. The microchannel was designed and fabricated in square cross section with 0.5 mm height and width with length of 28.0 mm whereas made from copper. During the experiment, a constant heat input of 325 W was set up at bottom of the heat sink. The combination of microchannel and nanofluids has provided both highly conducting fluids and large heat transfer area. This investigation found that heat transfer was achieved by using 2.5 wt. % Al₂O₃ nanofluids. The CFD simulation was performed in three dimensional and solving the conjugate heat transfer problem according to the experiment. In addition, present heat transfer performance was simulating on 5.0 wt. % concentration and found that it is yields the same result as use water. In order to understand more on the heat transfer performance in microchannel, this study has analyzed Nusselt number between the presented experimental and literature conventional correlations. It was clearly shown that, the experimental result have similarity pattern to the conventional correlations proposed by Boelter and Zerradi. After 10 hours of application of Al₂O₃ nanofluids, it was confirmed by XRD method that the heating process in the microchannel has not changed the structure of Al₂O₃ nanofluids. According to the XRD patterns show that the diffraction peaks are sharper after increase the temperature. Besides that, the morphological study found the heating process inside the microchannel has increased the grain size of Al₂O₃ nanofluids. Therefore, this investigation concluded that Al₂O₃ nanofluids improve the performances of heat transfer in the microchannel heat sink. tTe structure of Al₂O₃ nanofluids have no changed after 10 hours in the heating side but the grain size increased because of the particles started to agglomerate.
ABSTRAK

Penenggelam haba saluran mikro adalah jenis alat yang menghasilkan teknologi penyejukan inovatif untuk mengeluarkan sejumlah besar haba daripada permukaan yang kecil. Alat ini mempunyai saiz alur antara 10 µm hingga 1000 µm. Kebiasaannya, ia telah digunakan untuk penyejukan alat elektronik. Walau bagaimanapun, terdapat kekangan oleh untuk mereka bentuk dan menghasilkan saluran bersaiz mikro pada bahan logam. Kajian ini dilaksanakan untuk mengkaji perubahan haba dalam saluran mikro yang menggunakan air dan cecair nano Al2O3 yang mengalir dalam saluran mikro. Analisis kajian ini telah dilaksanakan secara ujikaji dan simulasi CFD. Alat ujikaji dihasilkan dengan tangki cecair, pam, meter aliran, blok tembaga dan Instrunet DAQ. Saluran mikro pula telah direka dan dibina dalam keratan rentas persegi dengan 0.5 mm tinggi dan lebar dengan panjang 28.0 mm pada bahan tembaga. Semasa ujikaji dijalankan, pemanas tetap 325 W dihasilakn pada bahagian bawah penenggelam haba. Simulasi CFD dijalankan pada model tiga dimensi dan telah menyelidik masalah konjugat pemindahan haba mengikut kaedah ujikaji. Gabungan saluran mikro dan cecair nano telah menemuihasil pemindahan haba yang lebih baik. Penyiasatan ini juga mendapati pemindahan haba maksimum terhasil dengan menggunakan 2.5 wt. % cecair nano Al2O3. Ia telah diterjemahkan oleh simulasi CFD bahawa prestasi pemindahan haba untuk 5.0 wt. % konsentrasi menghasilkan keputusan yang sama seperti menggunakan air. Perbandingan telah dilaksanakan untuk analisis nombor Nusselt antara kajian eksperimen dengan terbitan korelasi konvensional. Hasilnya jelas menunjukkan bahawa, korelasi konvensional yang dicadangkan oleh Boelter dan Zerradi mempunyai persamaan dengan kaedah ujikaji. Ia telah disahkan oleh kaedah XRD bahawa proses pemanasan di saluran mikro itu tidak mengubah struktur cecair nano Al2O3. Corak XRD hadir menunjukkan bahawa puncak pembelauan adalah lebih tajam selepas kenaikan suhu. Selain daripada itu, kajian morfologi mendapati proses pemanasan dalam saluran mikro telah meningkat saiz bijian daripada cecair nano Al2O3. Oleh itu, penyiasatan ini membuat kesimpulan bahawa cecair nano Al2O3 membantu untuk mempunyai prestasi yang lebih baik pemindahan haba dalam sink mikrosaluran haba. Selepas 10 jam, struktur cecair nano Al2O3 telah tidak berubah nanopartikel tetapi saiz bijirin meningkat kerana zarah mula menggumpal.
LIST OF TABLES

3.1 Average dimension of microchannel heat sink 49
3.2 Inlet velocity based on experimental volumetric flow rate 56
3.3 Crystallographic parameters 67
4.1 Specification of zone type in ANSYS workbench 74
4.2 Relaxation factors 75
4.3 Materials properties of working fluids 79
5.1 Crystallite size of 1.0 wt. % Al₂O₃ nanofluids before and after used in the microchannel 110
5.2 Crystallite size of 2.5 wt. % Al₂O₃ nanofluids before and after used in the microchannel 111
LIST OF FIGURES

2.1 Schematic diagram of (a) direct cooling and; (b) indirect cooling 7
2.2 Example of machining process using micro end milling 8
2.3 Example of microchannel heat sink (a) Copper based (b) Aluminum based 9
2.4 Computational domain to find heat transfer in microchannel 14
2.5 Example of meshed model for microchannel 15
2.6 Comparison Nusselt number by Dittus-Boelter, experiment and CFD simulation method 18
2.7 Comparison of Dittus-Boelter adjusted with experimental and CFD simulation 19
2.8 Comparison of pressure drop by experiment and numerical (CFD) simulation 23
2.9 Comparison pressure drop by increasing Al₂O₃ nanoparticles with pure water 24
2.10 Effect of preparation techniques on thermal conductivity of Al₂O₃ nanofluids 27
2.11 Comparison the heat transfer coefficient variation with Reynolds number for water, 1.0 wt. % Al₂O₃ nanoparticles concentration and 2.0 wt. % Al₂O₃ nanoparticles concentration for each of the four axial locations along the microchannels 28
2.12 Bragg’s Law theory for XRD method 32
2.13 X-ray diffraction pattern of water based magnetic nanofluids

2.14 XRD pattern results of materials according to samples;
   (a) Comprises desired amount of copper powder is added and
   stirred into Al(NO₃)₃, (b) involving add the desired amount of
   copper powder into solution of Al(NO₃)₃

2.15 Example of FESEM image

2.16 FESEM images of aluminum powder at different laser energies

3.1 Experimental flow chart

3.2 Schematic of experimental apparatus

3.3 Fluid temperature based on running time

3.4 Experimental test rig

3.5 Liquid tank

3.6 Pump

3.7 Metal pipe stainless steel

3.8 Rubber tube

3.9 Heater

3.10 Voltage regulator

3.11 Top view drawing of microchannel

3.12 Front view drawing of microchannel

3.13 Schematic of microchannel heat sink test section (a) Top view;
   (b) Side view of the test section with location of thermocouples

3.14 Flow meter

3.15 K-type thermocouple
<table>
<thead>
<tr>
<th>Section</th>
<th>Content</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.16</td>
<td>InstruNet DAQ</td>
<td>54</td>
</tr>
<tr>
<td>3.17</td>
<td>Temperature measured data</td>
<td>55</td>
</tr>
<tr>
<td>3.18</td>
<td>U-tube manometer</td>
<td>60</td>
</tr>
<tr>
<td>3.19</td>
<td>Al$_2$O$_3$ nanofluids in 20 wt. % concentration</td>
<td>62</td>
</tr>
<tr>
<td>3.20</td>
<td>Dilution process for Al$_2$O$_3$ nanofluids</td>
<td>63</td>
</tr>
<tr>
<td>3.21</td>
<td>Al$_2$O$_3$ nanofluids after reducing volume concentration</td>
<td>64</td>
</tr>
<tr>
<td>3.22</td>
<td>Al$_2$O$_3$ nanofluids kept in glass container</td>
<td>64</td>
</tr>
<tr>
<td>3.23</td>
<td>Image of Al$_2$O$_3$ nanoparticles</td>
<td>68</td>
</tr>
<tr>
<td>4.1</td>
<td>Schematic diagram of computational domain</td>
<td>70</td>
</tr>
<tr>
<td>4.2</td>
<td>Three dimensional model</td>
<td>71</td>
</tr>
<tr>
<td>4.3</td>
<td>Structured mesh for three-dimensional geometry of square shape microchannel (a) outlet plenum; (b) channel inside view close up; (c) model side view</td>
<td>76</td>
</tr>
<tr>
<td>4.4</td>
<td>Grid independence test</td>
<td>77</td>
</tr>
<tr>
<td>4.5</td>
<td>Heat transfer coefficient at different mesh grid size</td>
<td>78</td>
</tr>
<tr>
<td>5.1</td>
<td>Heat transfer coefficient of water in microchannel heat sink</td>
<td>81</td>
</tr>
<tr>
<td>5.2</td>
<td>Heat transfer coefficient of 1.0 wt. % Al$_2$O$_3$ nanofluids in microchannel heat sink</td>
<td>82</td>
</tr>
<tr>
<td>5.3</td>
<td>Heat transfer coefficient of 2.5 wt. % Al$_2$O$_3$ nanofluids in microchannel heat sink</td>
<td>83</td>
</tr>
<tr>
<td>5.4</td>
<td>Nusselt number versus Reynolds based on experimental result</td>
<td>85</td>
</tr>
<tr>
<td>5.5</td>
<td>Temperature at outlet plenum of microchannel</td>
<td>86</td>
</tr>
</tbody>
</table>
5.6 Temperature different of working fluids in microchannel

5.7 Comparison of present predicted Nusselt number with published results

5.8 Variation of wall temperature in the microchannel heat sink

5.9 Contour plot of temperature of water (a) along the microchannel and (b) outlet microchannel

5.10 Contour plot of temperature of 2.5 wt. % Al₂O₃ nanofluids (a) along the microchannel and (b) outlet microchannel

5.11 Contour plot of temperature of 5.0 wt. % Al₂O₃ nanofluids (a) along the microchannel and (b) outlet microchannel

5.12 Contour plot of temperature of 1.0 wt. % Al₂O₃ nanofluids (a) along the microchannel and (b) outlet microchannel

5.13 Comparison of Nusselt number based on CFD simulation and experimental result

5.14 Comparison experimental Nusselt number with proposed correlation

5.15 Comparison experimental Nusselt number with conventional correlation

5.16 Pressure drop in a square microchannel

5.17 Contour plot of pressure drop in the microchannel heat sink

5.18 Friction factor in a square microchannel

5.19 Heat transfer coefficient ratio of Al₂O₃ nanofluids versus Reynolds number

5.20 Heat transfer coefficient ratio versus thermal conductivity ratio
5.21 XRD pattern of 1.0 wt. % Al$_2$O$_3$ nanofluids used as working fluid in the microchannel before (Sample A1) and after 10 hours (Sample B1) used in the microchannel heat sink.  

5.22 XRD pattern of 2.5 wt. % Al$_2$O$_3$ nanofluids before (Sample A2) and after 10 hours (Sample B2) applied as working fluid in the microchannel heat sink  

5.23 FESEM images of 1.0 wt. % Al$_2$O$_3$ nanofluids (a) before and (b) after 10 hours used in the heat sink  

5.24 FESEM images of 2.5 wt. % Al$_2$O$_3$ nanofluids (a) before and (b) after 10 hours used in the microchannel.
**LIST OF SYMBOLS AND ABBREVIATIONS**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D</td>
<td>Three dimensional</td>
</tr>
<tr>
<td>Al₂O₃</td>
<td>Aluminum Oxide</td>
</tr>
<tr>
<td>EDM</td>
<td>Electric Discharge Machine</td>
</tr>
<tr>
<td>FESEM</td>
<td>Field Emission Scanning Electron Microscope</td>
</tr>
<tr>
<td>XRD</td>
<td>X-ray Diffraction</td>
</tr>
<tr>
<td>ϕ</td>
<td>Volumetric concentration (wt. %)</td>
</tr>
<tr>
<td>µₙf</td>
<td>Viscosity nanofluid (kg/ms)</td>
</tr>
<tr>
<td>µₙbf</td>
<td>Viscosity base fluid (kg/ms)</td>
</tr>
<tr>
<td>ρₙf</td>
<td>Density base fluid (kg/m³)</td>
</tr>
<tr>
<td>ρₚ</td>
<td>Density particle (kg/m³)</td>
</tr>
<tr>
<td>a</td>
<td>Dimensionless perimeter</td>
</tr>
<tr>
<td>b</td>
<td>width (mm)</td>
</tr>
<tr>
<td>cₚ</td>
<td>Specific heat capacity (kJ/kg K)</td>
</tr>
<tr>
<td>cₚₙbf</td>
<td>Specific heat capacity base fluid (kJ/kg K)</td>
</tr>
<tr>
<td>cₚₚ</td>
<td>Specific heat capacity particle (kJ/kg K)</td>
</tr>
<tr>
<td>D</td>
<td>Diameter (mm)</td>
</tr>
<tr>
<td>Dₕ</td>
<td>Hydraulic diameter (µm)</td>
</tr>
<tr>
<td>h</td>
<td>Heat transfer (W/m²K)</td>
</tr>
<tr>
<td>Kₙcu</td>
<td>Thermal conductivity copper (W/mK)</td>
</tr>
<tr>
<td>kₙf</td>
<td>Thermal conductivity nanofluid (W/mK)</td>
</tr>
<tr>
<td>kₚ</td>
<td>Thermal conductivity particle (W/mK)</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------</td>
</tr>
<tr>
<td>$k_{bf}$</td>
<td>Thermal conductivity base fluid</td>
</tr>
<tr>
<td>$k_f$</td>
<td>Thermal conductivity fluid</td>
</tr>
<tr>
<td>$L$</td>
<td>Length</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$P$</td>
<td>Perimeter</td>
</tr>
<tr>
<td>$W$</td>
<td>Width</td>
</tr>
<tr>
<td>$q_{gain}$</td>
<td>Heat gain</td>
</tr>
<tr>
<td>$T_w$</td>
<td>Wall temperature</td>
</tr>
<tr>
<td>$T_m$</td>
<td>Mean temperature</td>
</tr>
<tr>
<td>$\Delta T$</td>
<td>Temperature different</td>
</tr>
<tr>
<td>$u_{in}$</td>
<td>Velocity inlet</td>
</tr>
<tr>
<td>$\dot{V}$</td>
<td>Volumetric flow rate</td>
</tr>
<tr>
<td>$\Delta p$</td>
<td>Pressure drop</td>
</tr>
</tbody>
</table>
CONTENTS

TITLE i
STUDENT’S DECLARATION ii
ACKNOWLEDGMENTS iii
ABSTRACT iv
LIST OF TABLES vii
LIST OF FIGURES viii
LIST OF SYMBOLS AND ABBREVIATIONS xiii

CHAPTER 1 INTRODUCTION 1

1.1 Introduction to microchannel heat sink 1
1.2 Project background 2
1.3 Problem statement 3
1.4 Objectives 4
1.5 Scope of study 4

CHAPTER 2 LITERATURE REVIEW 5

2.1 Introduction 5
2.2 Overview of microchannel heat sink 5
2.3 Heat transfer in microchannel heat sink 10

2.3.1 Measurement heat transfer in microchannel heat sink by experiment 11
2.3.2 Prediction heat transfer in microchannel by CFD simulation 13
2.3.3 Heat transfer analysis using correlation method 17
4.5 Meshing of the computational domain
4.6 Grid independence test
4.7 Thermal properties of working fluids

CHAPTER 5 RESULT AND DISCUSSION

5.1 Introduction
5.2 Heat transfer coefficient in microchannel heat sink
5.3 Analysis Nusselt number by experiment
5.4 Fluid temperature in microchannel heat sink
5.5 Heat transfer in microchannel by CFD simulation
5.5.1 CFD simulation validation data
5.5.2 Predicted wall temperature of microchannel
5.5.3 Analysis Nusselt number by CFD simulation
5.6 Analysis Nusselt number with available conventional correlation
5.7 Pressure drop and friction factor in microchannel heat sink
5.8 Ratio of heat transfer coefficient using Al₂O₃ nanofluids
5.9 Structural analysis and morphological of Al₂O₃ nanofluids
5.9.1 XRD analysis
5.9.2 Morphological study of Al₂O₃ nanofluids

CHAPTER 6 CONCLUSION AND RECOMMENDATION

6.1 Conclusion
6.2 Recommendation

BIBLIOGRAPHY

APPENDIX
CHAPTER 1

INTRODUCTION

1.1 Introduction to microchannel heat sink

Microchannel heat sink is a device that has very fine channel of the width of normal human hair widely used in electronic cooling. The dimensions ranging is from 10 to 1000 µm and serve as flow passages for cooling fluid. The microchannel has very potential of wide application in cooling high power density microchips in the central processing unit (CPU) system and micro power system. It is also can be used to transport biological materials or chemical samples such as i-STAT blood sample analysis cartridge.

Normally, the microchannel heat sink is stacked together in order to increase the total contact surface area for heat transfer and reduce pressure drop. The microchannel heat sink combines the characteristics of very high surface area to volume ratio, large convective heat transfer coefficient, small mass and volume with small coolant inventory (Qu & Mudawar, 2002). Currently, the microchannel is fabricated either by precision machining or micro fabrication technology made from high thermal conductivity material such as silicon, aluminum and copper based. Therefore, the microchannel creates innovative cooling technology to remove large amount of heat from small area.
1.2 Project background

Heat transfer in micro scale is a very complex issue due to challenges in microchannel fabrication as well as in characterization of performance. The determination of heat transfer parameters in microchannel need to consider of physical size and concern of fluid interaction to surface.

Inspired by the past study, new design and modeling approaches of high performance cooling device have been proposed in a square shape due to the limitation of create deeper height of channel. The earlier studies found that by using water as working fluid in the microchannel heat sink can eliminate a maximum of 790 W/cm² of heat. Recently, nanotechnology gain interest to explore the microchannel cooling benefits of water containing small concentration of nanoparticles as working fluid. It has been found that nanoparticles dispersed with water have increased the heat transfer coefficient in the microchannel. It should be noted that limited studies are available on nanofluids flow and heat transfer characteristics in experiment.

This present investigation of heat transfer in a square microchannel heat sink to find heat transfer of water, 1.0 wt. % Al₂O₃ nanofluids and 2.5 wt. % Al₂O₃ nanofluids was carried out in experimental. Futhermore, it is also deals by add with 3D model CFD simulation of laminar flow of square shape microchannel with volume range from 1.0 wt. % to 5.0 wt. %. The CFD simulation used to illustrate the heat transfer in the microchannel and finds the optimum volume concentration of Al₂O₃ nanofluids that cannot be achieved by the experiment. The results of interests such as temperature distribution, heat transfer coefficient, pressure drop, friction factor and effect of the Al₂O₃ nanofluids volume concentration on microchannel performance. In addition, the structure of particles used and morphological studies to find effect of heating process by using Al₂O₃ nanofluids after applied as working fluids in the microchannel heat sink. There are two methods that very useful to investigate the effect in the structure of particles which are X-ray diffraction method (XRD) while Field Emission Scanning Electron Microscope (FESEM) is useful for morphological study.
1.3 Problem statement

Microchannel heat sink is well known as has potential of wide applications in large scale thermal systems requiring effective of cooling capacity. The magic behind the microchannel heat sink is their ability to achieve high heat transfer in small area. However, there was limit confined by manufacturing technology and on conduction inside solid material. This is because of difficulty to create large aspect ratio channels with uniform wall thickness in the micro-scale. The heat dissipation ability of liquid cooled heat sink is determined by heat conduction in solid material and heat convection in the fluid.

Nowadays, there is no doubt of needed to saving more energy and the using of traditional fluids such as water is not an effective way of improving the performances of microchannel heat sink. Recently, the developments in nanotechnology and related manufacturing techniques have made possible the production of nano-sized particles. There were number of studies about improving heat transfer in the microchannel using Al₂O₃ nanofluids. Since the solid meal has larger thermal conductivity than traditional working fluid (water), suspending metallic solid fine particles into water is expected to improve the thermal conductivity of that fluid. Most of previous study explores the heat transfer of nanofluids in range of 1.0 wt. % to 2.0 wt. % volume concentration. There was also less of study about the effect of the nanoparticles structure after been used as working fluid in the microchannel heat sink that have contacted with constant heat input.

Therefore, before the investigation started, it was assumed that the best heat transfer performance in square shape microchannel by using Al₂O₃ nanofluids. The structure of Al₂O₃ nanofluids will not have any changes after use in the microchannel. However, the agglomerate will occur that may cause from the heating process.
1.4 Objectives

The objectives of this research are stated below:

i. To measure heat transfer in microchannel by comparing workings fluids between water and Al$_2$O$_3$ nanofluids.

ii. To predict optimum heat transfer in microchannel using CFD simulation according to volume concentration of Al$_2$O$_3$ nanofluids.

iii. To determine the pressure drop and friction factor of working fluids in microchannel.

iv. To investigate structures and morphology of Al$_2$O$_3$ nanofluids before and after use in microchannel.

1.5 Scope of study

The scopes of study for this research are as below:

i. Measure and predict heat transfer in microchannel in range of Reynolds number of 633 to 1300 with constant heat input of 325 W.

ii. Compare heat transfer performance of water, 1.0 wt. % Al$_2$O$_3$ nanofluids and 2.5 wt. % Al$_2$O$_3$ nanofluids in a square shape microchannel heat sink while CFD simulation was run in range of 1.0 wt. % to 5.0 wt. % Al$_2$O$_3$ nanofluids.

iii. Analyze particles structure of Al$_2$O$_3$ nanofluids use X-ray Diffraction (XRD) and morphology using Field Emission Scanning Electron Microscope (FESEM) after 10 hours used in microchannel.
CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter explains details the investigation of heat transfer in microchannel heat sink based on available literature. There were three methods of study that have been used to explore the heat transfer in the microchannel. Most of past researchers have investigates by experimental measurement, CFD simulation and predict heat transfer by correlation. Recently, nanofluids have been found to increase the heat transfer performance rather than water by increasing the volume concentration. In advanced, this investigation explores more on the analysis structure of the nanofluids by X-ray diffraction method and morphological study.

2.2 Microchannel heat sink

Microchannel heat sink constitutes an innovative cooling technology for removal large amount of heat in small area. The heat sink usually made from a high thermal
conductivity material such as copper, silicon or aluminum. The dimensions ranging is from 10 to 1000 micron (Acharya, 2015).

Commonly, the microchannel has been used for electronic cooling. In a microchannel heat sink, the microchannel is stacked together in order to increase the total contact surface area for heat transfer enhancement and reduce pressure drop. As coolant, fluid flow through the microchannel, their large surface enables them to take large amounts of energy per unit time per unit area while maintaining a considerably low device temperature. The heat fluxes as high as 1000 W/m$^2$ can be dissipated at relatively low surfaces. There are two main configurations for its application which are direct cooling and indirect cooling. The direct cooling requires direct contact between surface of microchannel to be cooled and the coolant fluid. Figure 2.1 shows schematic of direct cooling and indirect cooling. The direct cooling requires contact between the surface to be cooled and coolant fluid as seen in the Figure 2.1 (a). It is reduces thermal resistance between the surface and coolant. This scheme causes the cooling performance become very effective. However, electrical and chemical compatibility between the coolant and device need to be ensured for this system work. An alternative from the configuration is use of metallic heat sink to conduct the heat away from the device to the coolant. The Figure 2.1 (b) shows the configuration allows for a greater flexibility in coolant selection at the cost of increased thermal resistance between the device and the heat sink due to the heat of diffusion.
The design of experimental facility is depend on dimension of the microchannel. The hydraulic diameter and cross-section area are restricted by the available fabrication technique. Based on the micro mechanization technique, it is reasonable mechanization quality for square cross sectional area for channel of at least 100 micron in height. It is important to remark the influence that the micro-fabrication method and its mechanical characteristics have the frictional behavior of liquid micro flows. The structure and flow configuration of the microchannel heat sink need to consider.

The selection of micro machining techniques for the fabrication of microchannel depends on the size of channel, aspect ratio, surface roughness etc. An electrical discharge machining (EDM) process is one of technique used to remove metal through the action of an electrical discharge of short duration and high current density between the tool and the work piece. Recently, developments in the field of EDM have progressed.

Figure 2.1: Schematic diagram of; (a) direct cooling and; (b) indirect cooling (Gunnasegaran et al., 2010)
due to the growing application of *EDM* process and the challenges being faced by the modern manufacturing industries, from the development of new materials that are hard and difficult to machine such as tool steels, composites, stainless steel and etc. (Shamsudin *et al.*, 2008).

Among the micro machining techniques, the micro end milling has shown great potentials in the fabrication of micro features. The motivation comes from the translation of the knowledge obtained from the conventional process to the micro level. Its specialty includes the ability of fabricating micro features from wide varieties of materials with complex three dimensional geometries (Ali, 2009). Micro Electrical Discharge milling is a promising technique for machining microchannels on metallic materials with almost no burrs as shown in Figure 2.2 below. This process is usually used for fabricating master micro mold which will be used for replication of microfluidic channels on plastic mainly by hot embossing.

![Figure 2.2: Example of machining process using micro end milling (Shamsudin *et al.*, 2008)]
Material of microchannel also plays an important thing during heat transfer process. Figure 2.3 shows an overview of the molded copper and aluminum block containing arrays of microchannel. Working fluids will flow through the microchannel and absorb heat in the heat sink. Previous study by (Parida, 2007) investigated heat transfer is higher at copper based microchannel compared to aluminum. It is speculated that the presence of significant surface roughness within the microchannel resulted the increasing of flow mixing. The thermal performance for metal based microchannel heat sink devices has enormous potential in many heat transfer problem primarily in electronic cooling. The heat transfer in microchannel is not only significantly reduces the weight load but also increase the capability to remove much greater amount of heat than any of large scale of cooling systems. There are many design of microchannel from past researchers based on their purpose and objectives.

Figure 2.3: Example of microchannel heat sink; (a) Copper based (b) Aluminum based (Parida, 2007)

The microchannel heat sink has very potential of wide applications in cooling high power density microchips in central processing unit (CPU) system, micropower systems and other large scale thermal systems requiring effective cooling capacity. Moreover, the growing interest of industrial in micro devices and their wide applications
especially in cooling high-heat-flux has motivated many researchers to investigate flow phenomena in microchannel.

2.3 Heat transfer in microchannel heat sink

There are number of significant advantages of heat transfer in microchannel. The main advantages of microchannel is their smaller dimension of channel that allowing for faster reactions, better product quality and smaller amounts of costly reactants also easy parallelization of analytical processes. Nowadays electronic components are required to perform tasks at faster rate and so high-powered integrated circuits have been produced. The high-speed circuits are expected to generate heat fluxes that will cause the circuit to exceed its allowable temperature. It is also found that, the heat input low the frictional losses and viscosity leading to increasing the fluid temperature especially at lower Reynolds number.

According to the previous investigation, when the size of the channel reduces to micron size, the heat transfer coefficient can increased thousand times from the original value (Liu et. al, 2007). In addition, it has been stated that heat transfer coefficient increase based on cross-section shape of the channel, Reynolds number and thermal conductivity of working fluid. Usually, the rectangular shape have been favorite choice because of using deeper channel, it will increase the convective heat transfer and reduce the pressure drop of the microchannel heat sink. However, there is limit confined by manufacturing technology of solid material. This is because of difficulty to create large aspect ratio channels with uniform wall thickness in the micron scale.

There are different published results of heat transfer in microchannel heat sink based on methods used by previous researchers. They have investigated by experimental, simulation and conventional correlation methods. It has been investigated by (Lee et al., 2005), shown that heat transfer increase with decreasing the size of channel and flow rate. The wide disparities reveled that mismatch in the boundary and inlet conditions between experimental and the conventional correlations precluded their use for predictions. Then,
the numerical methods were found to be in good agreement with experimental data. Since the experimental taken long time to get the results, numerical simulation can be used to carry out the investigation in many type of fluids. The computational fluid dynamic (CFD) packages could be used to simulate various types of coolants and help to predict which one provides better cooling capabilities at affordable cost. Then, the experiments may be performed later to validate the simulation’s results (Hassan et al., 2004). Effect of working fluid types should be investigated more thoroughly. The liquid seem to provide superior cooling properties compared to gases, since they offer lower thermal resistance. Water has been chose as working fluid for most experiments because it is readily available, cheap, and high specific heat capacity.

2.3.1 Measure heat transfer in microchannel heat sink

Measure heat transfer is carried out by experimental work. There is a test rig need to be fabricated include with measurement of instrument such as flow meter and thermocouple. Concept used by (Parida, 2007) and (Irwansyah et al., 2014) for their experiment of heat transfer in microchannel heat sink are simple, easy to understand and low cost. Both studies have using similar method and concept whereas the researcher consist the apparatus in three sections which is pressuring section, test section and data acquisition section. During the experiments, the microchannel heat sink was used as cooling device with water as working fluids. It has been considered a single-phase flow for all heat transfer experiment. Recently, Al$_2$O$_3$ nanofluids was used as working fluids and the circulating flow was maintained using thermostatic.

A K-type thermocouple is used and placed at inlet and outlet plenum of the microchannel to measure fluid temperature at both sides. In order to produce heat in the microchannel, cartridge heater is inserted. During the experiment, the fluid is entering to the microchannel with constant fluid flow rate using flow meter. The heat transfer is determined by calculating heat gain produced in microchannel based on temperature reading, density, specific heat of fluid and flow rate. The area of convective heat transfer
is needed to be considered. Therefore, heat gain computed from this equation represents a conservative calculation of water power gain.

Usually, the investigation of heat transfer has been carried out for a straight microchannel by selecting the criteria of temperature distribution, pressure drop and heat transfer coefficient as cooling performance. It has been concluded that the straight microchannel give better result at Reynolds number in range of 400 to 450 with flow rate in between 320.0 ml/min to 350.0 ml/min (Gawali et. al, 2014). The experimental results in heat transfer indicated that forced convection in the microchannel exhibited excellent cooling performances, especially in the phase change regime. It was applied as heat removal and temperature control devices in high power electronic components. When the critical nucleate heat flux condition appeared, the flow mechanism changed into fully developed nucleate boiling and accompanied with wall temperature decreased rapidly while pressured drop increased sharply (Chen et al., 2004).

There were sources of error in the investigation of heat transfer in the microchannel by experimental study. This is because the utilize conduits with micro scale dimensions have a number of inherent problems that can lead to undetectable errors. These experimental difficulties can be categorized into three distinct areas. A first source of error is measurement of dimensions. The errors in dimensional may be a result of the measurement technique or due to changes in dimensions that occur under test condition. The second error is measurements of pressure and flow rate. The errors usually occur primarily due to measurement uncertainty of the instruments and failure to account for entrance and exits effects. In many cases, the pressure is measured in manifolds outside the microchannel and the entrance and exits effects have either been estimated or neglected. The third error is surface effects. According to laminar theory, friction factor is independent of wall surface roughness. However, on the microchannel, molecular interaction with the walls increases relative to intermolecular interaction when compared to traditional scale flows (Papautsky & Frazier, 2001).
2.3.2 Prediction heat transfer in microchannel by CFD simulation

A computational fluid dynamics (CFD) simulation is performed usually to reduce cost and time in the investigation. The CFD can be used in parallel with experimental setup in an effort to predict the flow and heat transfer characteristics of given surface modification under specified control parameters and boundary conditions. There are different analyses using CFD which is a model was formulated to solve for the three dimensional conjugate heat transfers in the microchannel by accounting for both convection in the channel and conduction in the substrate. The simulation was performed for specifics test geometries. A uniform heat flux usually applied at bottom surface which is simulating the heat flow from the cartridge heaters. At the inlet, a fully developed velocity profile was specified for thermally developing flow simulations while uniform inlet velocity used for simultaneously developing flow simulations (Pandey, 2011).

The CFD simulation of heat transfer in microchannel can be performed in four approaches. First, is two dimension (2D) model included inlet and outlet plenums with constant heat flux is applied at bottom surface. Second approach is three dimensional (3D) thin wall model (3D thin wall model with one side heated). In this model, the wall thickness is zero and constant heat flux is applied at the bottom surface only while other walls were considered as adiabatic. Third approach is three dimensional thin wall model with 3 sides heated (3D thin wall model). This model is similar with the previous model approach. The different is a constant heat flux is applied at the bottom and two side walls while the top wall was keep as adiabatic. The last approach is three dimensional full conjugated model (3D full conjugate). In this model, the full copper block which similar to the experimental is simulated. A constant heat flux boundary condition is applied at the location of the cartridge heater. The inlet and outlet plenums are not included in this model. According to (Mahmoud et al., 2014), the 3D thin wall models predicted the experimental values with excellent agreement compared to 3D full conjugate model. The deviation between the experimental values and conjugate model could be due to fact that conjugate effects are not taken into consideration in the experimental data reduction process. Figure 2.4 shows example of computational domain to find heat transfer in
microchannel. A uniform heat flux, $q''$ was applied at bottom surface of microchannel while the temperature and velocity is constant at the inlet channel. It has been considered a steady 3D flow in the microchannel with heating from below and adiabatic conditions at the other boundaries. The symmetry of the heat sink yields as domain comprised of unit cell with single microchannel.

![Computational domain to find heat transfer in microchannel](image)

Figure 2.4: Computational domain to find heat transfer in microchannel (J. Li et al., 2004)

One of the important things in order to proper resolve the velocity and viscous shear layer is by choosing fine grid mesh for y and z directions. In addition, to have more accurate, define the conjugate heat transfer at the surface of the channel, thereby to improve the temperature resolution. The comparison with standard theoretical or simulation results, indicates the fine of the mesh size, the higher simulation result accuracy. Therefore, in order to ensure the accuracy and reliability of simulation results, it is necessary to carry out a mesh independency test by varying the cell density on a model for same set of boundary conditions. Figure 2.5 represents three dimensional geometry of microchannel with structured mesh.
In order to achieve solutions for the flow and the temperature fields, boundary conditions are specified to the model. Usually, there is no slip boundary condition assigned to the surfaces, where both phases enter the channel at the inlet with the same uniform velocity that is specified according to the Reynolds number. At the channel outlet, it is specified as pressure where the flow reach atmospheric pressure. The heat sink surfaces are subject to adiabatic condition for all position except the bottom sink. The heat flux is assigned as effective to channel bottom, channel left and channel right while for channel top is subject as adiabatic condition (Pandey, 2011). In order to focus on the effect of using nanofluids with different volume concentration, the following assumptions are made: (i) both fluid flow and heat transfer are in steady-state and 3D; (ii) the fluid is in single phase, incompressible and in laminar flow; (iii) the properties for both fluid and heat sink material are temperature-independent; and (iv) all the surfaces of the heat sink exposed to the surroundings are assumed to be adiabatic except for the bottom where constant heat flux is specified (Mohammed et al., 2010). The single phase model equations including continuity, momentum, and energy equation (ANSYS Fluent) for fully developed 3D flow heat transfer are;
Conservation of mass (continuity)
\[ \nabla (\rho \vec{V}) = 0 \]  
\[(2.1)\]

Conservation of momentum
\[ \vec{V} \cdot \nabla (\rho \vec{V}) = -\nabla p + \nabla \cdot (\mu \nabla \vec{V}) \]  
\[(2.2)\]

Conservation of energy for fluid
\[ \vec{V} \cdot \nabla (\rho c_p T_f) = \nabla \cdot (k_f \nabla T_f) \]  
\[(2.3)\]

Conservation of energy for solid
\[ \nabla \cdot (k_w \nabla T_w) = 0 \]  
\[(2.4)\]

The CFD simulation method able to investigate in range of low Reynolds number and different value of heat flux provided. Previously, they have evaluated the performances of microchanel in terms of temperature profile, heat transfer, velocity profile, and pressure drop and friction factor. The results reveal that both heat transfer and pressure drop increase by increasing the Reynolds number (Khafeef & Albdoor, 2014). Normally, the simulation results matched closely with the experimental and calculation from correlation method. It is important to use temperature dependent material properties in computational method as the temperature dependent material properties influence the accuracy of the results. The heat transfer does not vary for the simulation using constant material properties. The difference is less pronounced when comparing the pressure drop. The heat transfer was found to increase by 5.0 % for every 10.0 degree rise in fluid temperature (S.Subramaniam et al., 2014).

The analysis of predicted heat transfer performance in the microchannel can be done based on Nusselt number. It has been found that the Nusselt number increase as the fluid enters into the inlet. This could be anticipated as result of the development of thermal entry region at the channel and the values of the Nusselt number tend to stabilize
after fully develop region has been achieved. The energy equation for the entry length is complicated. This is because of simple solution to explain the thermal entry length problem is based on assuming that thermal condition develop in the presence of fully develop profile. It can be seen that, when Reynolds number increase, the value of Nusselt number also increase (Abubakar et al., 2015).

When the operating power and speed increases, the designers are forced to reduce overall systems dimensions, the problems of extracting heat and controlling temperature becomes crucial. The seemingly simplified and quicker modelling process sometimes may have hidden weaknesses that the user may not be aware about that. Previously investigated by (Deepak et al., 2014) stands to the challenges posed by increasing chip heat flux, smaller enclosures and stricter performance and reliability standard. It is utilizes the CFD to identify a cooling solution for a desktop computer, which uses a 5 W CPU. The design is able to cool the chassis with heat sink attached to the CPU is adequate to cool the whole system. Nowadays, there were many interesting results suggesting potential of using nanofluids in microchannel to enhance the thermal performance. It is shown that increasing the thermal conductivity of working fluid is enahce the heat transfer performance of the microchannel (Abubakar et al., 2015).

2.3.3 Heat transfer analysis using correlation method

Another method to measure heat transfer in the microchannel heat sink is used a conventional correlation method. Usually the correlations were proposed according to experimental result. The earlier studies focus on single phase heat transfer of moving fluid in a smooth tube to develop correlation schemes. In fact, comparison of experimental data with heat fluxes value has clearly present the increasing heat flux will strongly increase nucleate pool boiling heat transfer coefficient.

In order to estimate the heat transfer coefficient accurately, conventional correlation has been developed on the basis of Stephan and Preußer correlation (Sarafraz et al., 2012). It is apply to model the heat transfer coefficient in other geometries. The
most widely used correlation in pipe is well known Dittus-Boelter (1930) equation as given follow

\[ Nu = 0.023 \text{Re}^{0.8} \text{Pr}^{0.4} \] (2.5)

where \( Re \) is the Reynolds number, \( Pr \) represents the Prandtl number, and \( Nu \) is the number. The model presents an ideal that flow boiling can be separated into a nucleate boiling terms as characterized by still fluid being boiled by heat applied directly to nucleation sights. A convective boiling term to characterize heat transferred to moving fluid is the microchannel (Donowski & Kandlikar, 2009). The Dittus-Boelter is well known and most correlation been used to analyze heat transfer in circular, rectangular and square shape microchannel. Figure 2.6 shown comparison Nusselt number by Dittus-Boelter correlation with experiment and CFD simulation. The Dittus-Boelter correlation is range of comparability with both experiment and CFD data.

![Figure 2.6: Comparison Nusselt number by Dittus-Boelter, experiment and CFD simulation method (Phillips, 2008)](image-url)
However, new equations have been developed in order to fit more with the experimental and CFD simulation data. Figure 2.7 shown the results of adjustment Dittus-Boelter compare with experimental and CFD simulation is more close to each other. The good agreement between the experimental data and the adjusted Dittus-Boelter equations indicates that it is in fact worthwhile and beneficial to account for both surface roughness and thermal entry length effects when comparing experimental result with Nusselt number correlations.

Figure 2.7: Comparison of Dittus-Boelter adjusted with experimental and CFD simulation (Phillips, 2008)

There was many development of new number based on Dittus-Boelter to fit with other shape of microchannel as expressed by Owhaib & Palm (2004) in equation 2.9. The equation has focus on laminar flow in circular pipe.

\[ Nu = 0.000972 \ Re^{1.17} \ Pr^{\frac{1}{3}} \]  

(2.6)
It is important to stress that deviations from experimental trends are not necessarily related to weakness in the correlations themselves. Operating conditions of water-cooled microchannel failing outside the recommended application range for most correlation (Qu & Mudawar, 2003). However, the correlation been stated is not suitable to find heat transfer using nanofluids. It is difficult to establish any formulated theory that can predict the flow of nanofluids. It is expected that the Nusselt number depends on number of factors such as flow pattern, viscosity of base fluid, shape of particle, volume fraction, thermal conductivity and specific heat capacity of base fluid. Therefore, the general form of Nusselt number for nanofluids is expressed as in equation 2.12.

\[
\text{Nu}_{nf} = f \left( \text{Re}_{nf}, \text{Pr}_{nf}, \frac{k_d}{k_f}, \frac{\rho c_p}{\rho c_p}_d, \phi \right)
\]

where Re_{nf} is the Reynolds number of the nanofluids, Pr_{nf} is the Prandlt number of the nanofluids, k_d is the thermal conductivity of the nanoparticles, k_f is the thermal conductivity of base fluid, \((\rho c_p)_d\) is the heat capacity of the nanoparticles, \((\rho c_p)_f\) is the heat capacity of the base fluid and \(\phi\) is the volume fraction of nanoparticles. Consequently, the Nusselt number must be a function of the nanoparticle concentration of the nanofluids. According to this argument, a new Nusselt number correlation is developed as function of Re_m, Pr and \(\phi\).

According to the Nusselt number is function of the thermal conductivity and heat transfer coefficient of the nanofluids, it is vary with the particle concentration of nanofluids. Consequently, Nusselt number must be a function of the particle concentration of the nanofluids. The equation 2.8 was developed as the function of Reynolds number, Prandlt number and volume concentration (Sudarmadji, 2015).

\[
\text{Nu} = f \left( \text{Re}_m, \text{Pr}, \phi \right)
\]

where Re_m is represent as modified Nusselt number while f is the base fluid. Based on this equation, a new Nusselt number correlation which contains additional term for particles concentration for nanofluids is given in following equation 2.9.
\[ Nu = Pr^p \left( \alpha \phi + \beta Re^q + \chi \phi Re^q + \delta \right) \]  \hspace{1cm} (2.9)

The data of nanofluids available in the literature for different sizes at different volume fraction and temperature are subjected to nonlinear regression analysis. The constants are obtained for the case of nanofluids containing spherical particles. Thermo physic coefficients remain as constant value. The \( \alpha = 1.0257, \beta = 1.1397, \chi = 0.7884, \delta = 1.207, \) \( p = 0.1039 \) and \( q = 0.205. \) The choice of these spherical nanoparticles was governed by the abundance of experimental data available in the literature (Zerradi et al., 2014).

As mention, the enhanced heat transfer by nanofluid may result from the following aspect which is the suspended particles increase the thermal conductivity of two-phase mixture. The other aspect is that chaotic movement of ultrafine particles accelerates energy exchange process in the fluid.

\[ Nu_{nf} = 0.4238(1.0 + 11.285 \phi^{0.754} Pe_d^{0.218}) Re_{nf}^{0.333} Pr_{nf}^{0.4} \]  \hspace{1cm} (2.10)

Based on the equation (2.14), the calculated results of sample nanofluids indicates that correlation correctly takes into account the main factors that affect heat transfer of the nanofluids and can be used to predict heat transfer coefficient of the nanofluids (Li & Xuan, 2002).

### 2.4 Pressure drop and friction factor in microchannel heat sink

Advances in the micro fabrication make it possible to build microchannel with small characteristic lengths. The microchannel has shown promising potential for being incorporated in a wide variety of unique, compact and efficient cooling applications such as in microelectronic device. Large number of papers has reported pressure drop data for laminar flow of fluid in microchannel with various cross sections. The fully developed flow frictional pressure drops can be measured in experimental and computational methods. In order to predicts the pressure drop of fully developed a compact approximate model is proposed by (Bahrami, et al., 2005). The proposed model was
successfully predicts the pressure drop for a wide variety of shapes with maximum difference on the order of 8.0%.

Pressure drop occurs between two points of fluid carrying network in microchannel heat sink. Usually, the pressure is low at exit of microchannel. There have many ways to measure pressure in microchannel such as experimental, correlation and numerical prediction. Based on previous study, the pressure drop is increase when increasing Reynolds number. Figure 2.8 shows pressure drop measured by previous study (Qu & Mudawar, 2002). The figure shows the pressure drop increase as the increasing Reynolds number. Both measured and predicted pressure drop accounts for the pressure drop along the microchannel. The increasing pressure drop is cause by the pressure losses associated with the abrupt contraction and expansion at the inlet and outlet of microchannel.

There are several reasons for the slope change in the pressure drop characteristics as seen in the figure above. Firstly, as the constant power input and water inlet temperature, the outlet water temperature should decrease with increasing Reynolds number. Secondly, the inlet and outlet pressure losses are proportional to the square of velocity. Therefore, increasing Reynolds number produces more pronounced increase in the inlet and outlet pressure losses.
There is a relationship between pressure drop with friction factor. The previous study has discussed about the relationships of pressure drop with the products of Reynolds number and friction factor. It has been indicated that entrance and exit losses need to be accounted for while presenting overall friction factors losses in the microchannel heat sink. Most of the data that accounted for friction factor loss show good agreement with the theory. According to (Mohammed et al., 2010), the effect of friction factor has been computed by using Darcy equation. They presented the friction factor was similar for all particle volume fractions where it decreases with the increase of Reynolds number. The application of nanofluids to the microchannel appears to give a slight rise in the friction factor. Therefore, the friction factor increase directly proportional to the particle volume fractions.

There was clearly observed that friction factor increase when decrease the flow area and pressure drop (Bahrami et al., 2005). The fluid in laminar flow with smooth or near smooth microchannel with hydraulic dimeter of several hundred micrometers obeys the theory. If there exist such as micro effects as electro-viscous effects for liquid flow, roughness effects, their impacts are negligible in such microchannel or at least cannot be identified with the consideration of the experimental uncertainties (Yue et al., 2004).
Based on Figure 2.9, for each axial location, 2.0 wt. % Al₂O₃ nanoparticles concentration shows the greatest heat transfer enhancement. For 1.0 wt. % Al₂O₃ nanoparticles concentration, the enhancement is obvious for the first two upstream locations, but virtually nonexistent for the two downstream locations where the thermal boundary layer is almost fully developed. In fact, the enhancement effect for both concentrations appears far more prevalent in the entrance region than the downstream fully developed region. It can therefore be concluded that nanoparticles have an appreciable effect on thermal boundary layer development.

Figure 2.9: Comparison pressure drop by increasing nanoparticles with pure fluids (Lee & Mudawar, 2007)

The comparison shown that pressure drop increase using 2.0 wt. % concentration of Al₂O₃ compared to pure fluid. This result proved that using high concentration of nanoparticles will increase pressure drop in microchannels. Usually, pressure is high at the entrance but it become low at outlet of microchannels. The value of pressure drop also different based on materials used for specimen test.
BIBLIOGRAPHY


coefficient of MEG/DEG/water ternary mixture. *Chemical Industry and Chemical Engineering Quarterly, 18*(1), 11–18. doi:10.2298/CICEQ110625041S


