Faculty of Mechanical And Manufacturing Engineering Technology

Numerical Study of Thermal Enhancement in Microchannel Heat Sink with Applying Cavities Technique

Muhammad Zawawi Bin Zulkefly

Bachelor of Mechanical Engineering Technology (Refrigerating & Air Conditioning System) with Honours

2019

DECLARATION

I declare that this thesis entitled "Numerical Study of Thermal Enhancement in Microchannel Heat Sink with Applying Cavities Technique" is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....

Name : Muhammad Zawawi Bin Zulkefly

Date : ....

APPROVAL

I hereby declare that I have read this dissertation/report and in my opinion this dissertation/report is sufficient in terms of scope and quality as a partial fulfillment of a Bachelor of Mechanical Engineering Technology (Refrigerating & Air Conditioning System) with Honours.

Signature :
Supervisor Name : Ts. Qamar Fairuz Bin Zahmani
Date :

ABSTRAK

Teknologi komputer mula berkembang pada tahun 1940-an yang membuat pemindahan dan memproses data pada komputer menjadi lebih cepat. Apabila teknologi mula berkembang, komponen atau peranti elektrik menjana lebih banyak haba. Haba yang dihasilkan oleh komponen elektronik akan merosakkan komponen atau prestasi peranti elektronik akan jatuh. Oleh itu, banyak penyelidik telah mengkaji tentang pembangunan sinki haba mikro seperti tambah rongga untuk kadar pemindahan haba, penggunaan kuasa kurang, kecekapan dan sebagainya. Walaupun terdapat banyak bentuk kaviti yang telah dikaji tetapi masih terdapat banyak bentuk yang belum belum lagi menaikkan kadar pemindahan haba ke dalam rongga. Tujuan kajian ini adalah untuk menghasilkan model baru dengan menggunakan kaedah rongga dan mensimulasikan model-model baru sinki haba mikro dengan rongga. Kemudian, untuk menganalisis peningkatan pemindahan haba dengan kaedah analisis berangka. Selepas itu, buat perbandingan kadar pemindahan haba di antara model penyejuk haba mikro petak dan sinki haba mikro baru dengan model kaviti. Terdapat tiga model yang dijana dengan menggunakan perisian SolidWorks iaitu sinkronik haba mikro segi empat tepat, sinki haba mikrochannel dengan rongga oktagon dan sinki haba mikro dengan rongga bujur. Untuk menilai model ini, perisian dinamik cecair pengkomputeran (CFD) yang digunakan adalah ANSYS Fluent. Analisis ini akan terdiri daripada perbincangan dalam suhu dan halaju. Keputusan kajian ini menunjukkan bahawa suhu maksimum adalah terdapat dalam sinki haba mikro dengan rongga bujur pada 44.85^{o} C. Ia menunjukkan bahawa peratusan prestasi meningkat sebanyak 7.2%. Halaju tertinggi juga mendapat keuntungan dalam sinki haba mikro dengan rongga bujur pada 3.336 m / s. Perbandingan dibuat dan memutuskan bahawa sinki haba mikrochannel dengan rongga bujur adalah reka bentuk terbaik kerana ia memperoleh nilai tertinggi dalam suhu dan halaju. Penemuan mencadangkan bahawa untuk mengarang model untuk menjalankan kajian eksperimen untuk membandingkan hasil antara simulasi dan kajian eksperimen. Pemahaman yang baik mengenai mekanik fluida diperlukan untuk menilai keadaan sempadan awal untuk menjadi sama dengan keadaan sebenar.

ABSTRACT

The technology of computer begins to expand on 1940's that make the transferring and processing data on computer become faster. As the technology begins to expand, the electrical component or device generate more heat. The heat that has been generated by electronic component will damaged the component or the performance of electronic device will be drop. Therefore, many researchers have study about the development of microchannel heat sink such as added cavities to heat transfer rate, less power consumptions, efficiency and etc. Although there are many shapes of cavities has been research with but there are still many shapes has not been discovering yet to improve the rate of heat transfer in the cavities. The purpose of this study is to generate a new model with applying cavities method and simulate the new models of microchannel heat sink with the cavities. Then, to analyse the heat transfer enhancement with numerical analysis method. After that, make a comparison of the rate of heat transfer between the rectangular microchannel heat sink model and the new microchannel heat sink with cavities models. There are three model that be generated by using SolidWorks software that is rectangular microchannel heat sink, microchannel heat sink with octagon cavities and microchannel heat sink with oval cavities. To analysis this model, a computational fluid dynamics (CFD) software that is ANSYS Fluent is being used. The analysis will consist of discussion in temperature and velocity. The results of this study show that the maximum temperature is gain in microchannel heat sink with oval cavities at 44.85^{o} C. It shows that the percentage of performance is increase by 7.2%. The highest velocity also gains in microchannel heat sink with oval cavities at 3.336 m/s. The comparison is be made and decide that microchannel heat sink with oval cavities is the best design due to it gains the highest value in temperature and velocity. The findings suggest that to fabricate the model to run an experimental study to compare the results between simulation and experimental study. A good understanding of fluid mechanic is needed to evaluate the initial boundary condition to be same as the real situation.

DEDICATION

Alhamdulillah, be grateful to Allah S.W.T because keeps me level head up throughout this project. I would like to dedicated this project to my beloved family especially my mother Pn Norazilah Binti Nasarudin, who encourage me mentally, physically and spiritually to keep moving forward throughout this project and inspire me to break border of true limitation and reach the definite of purpose. In additional, I would like to give huge credit to my siblings through financial support and extra push of encouragement. Lastly, I would like to give such a special thanks to my supervisor Ts. Qamar Fairuz Bin Zahmani who is guide and help me from early stage until complete this project. Thank you.

ACKNOWLEDGEMENTS

First and foremost, I would like to take this opportunity to express my sincere acknowledgement to my supervisor Ts. Qamar Fairuz Bin Zahmani from the Faculty of Mechanical and Manufacturing Engineering Technology Universiti Teknikal Malaysia Melaka (UTeM) for his essential supervision, support and encouragement towards the completion of this thesis.

Special thanks to all my peers, my late father, beloved mother and siblings for their moral support in completing this degree. I would like to appreciate to Universiti Teknikal Malaysia Melaka (UTeM) to give the opportunity for me to study in Bachelor of Mechanical Engineering Technology (Refrigeration and Air Conditioning) and bring out my true potential during my studies.

Lastly, thank you to everyone who had been to the crucial parts of realization of this project.

TABLE OF CONTENTS

CONTENTS	PAGE

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

CFD	Computational Fluid Dynamics
MCHS	Microchannel Heat Sink
Re	Reynolds Number
^{o}C	Degree Celcius
ρ	Density
V	Velocity
L	Lenght
μ	Dynamic Viscosity
v	Kinematic Viscosity
Nu	Nusselt Number
h	Heat convection coefficient
D	Hydraulic Diameter
k	Thermal Conductivity
A	Area
P	Perimeter
H	Height

CHAPTER 1

INTRODUCTION

1.0 Introduction

On these days, an electronic device is an important device that needed in modern human life. With the technology of an electronic begins to expand that make an electronic device became faster to transfer and process the data. It makes the work became more fast and easier for human.

The technology of computer begins to expand on 1940's that make the transferring and processing data on computer become faster. As the technology begins to expand, the electrical component or device generate more heat. The heat that has been generated by electronic component will damaged the component or the performance of electronic device will be drop. This problem will make the experience of using the device will be affected. In the beginning, the cooling system that be used for cooling the electronic component was used fan by blow the air to the component to remove the air to the environment. But it required more power and space as it needs a big size of fan to cool the component.

Therefore, the new technology has been discovering by Tuckerman and Pease that be called microchannel heat sink. The microchannel heat sink is able to dissipate more the heat and more efficient. Due to that the electronic device become more cooler compare to not using the microchannel heat sink. When the electronic device such as processor cooler than before, it can work longer time than usual that make the electronic device can generate more production than before.

1.1 Problem Statement

The microchannel heat sink that be discover by Tuckerman and Pease in 1981 has made a major impact to the industry. It allows the electronic device work more efficiency and long time. But the technology will keep more expands than before make the electronic device generate more heat. Therefore, many researchers have study about the development of microchannel heat sink such as added cavities to heat transfer rate, less power consumptions, efficiency and etc. Although there are many shapes of cavities has been research with but there are still many shapes has not been discovering yet to improve the rate of heat transfer in the cavities. So, it a waste to not investigate the effect of applying shape cavity in the microchannel heat sink and maybe can improve and surpass the current results. Therefore, this study will purpose the new shape of design to analysis the behaviour of water in microchannel heat sink.

1.2 Objective

To investigate the 3D models of microchannel heat sink with the cavities that has been upgraded. Then make the comparison of the result between the rectangular and the new microchannel heat sink models. By using the ANSYS FLUENT to simulate the models to take the results of the experiment. The parameter of the results that will be analysis in this experiment is temperature and velocity.

The objective of this study will be:

i. To generate a new model with applying cavities method and simulate the new models of microchannel heat sink with the cavities.
ii. To analyse the heat transfer enhancement with numerical analysis method.
iii. To compare the rate of heat transfer between the rectangular microchannel heat sink model and the new microchannel heat sink with cavities models.

1.3 Scope of Project

This study will be start by generated the new model using the software SOLIDWORK. Then, the model will import to the Computational Fluid Dynamics (CFD) software to meshing the model and insert the boundary conditions. Then, the model is been analysis to study on the behaviour of fluid such as the heat transfer and velocity. The CFD software that will be use is ANSYS FLUENT. The project will not cover any experimental study for microchannel heat sink.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

Electronic device has been a part of human life that will make a work become more easier than working it by manually like a past time. As time passes, the technology of electronic device become more grow that make the work so fast and easier. As the growing of technology of electronic device, it generated more heat that make the performance will drop. So, a solution is need to solve the problem of heat generated on electronic device. Microchannel Heat Sink (MCHS) is one of solution that can increase the cooling rate to dissipated the heat that be generated by electronic device. The advantages for using MCHS is it more compact, lighter and possess more surface area that will impact the cooling rate. Computational Fluid Dynamics (CFD) is a tool that use mathematics, physics and visualize the flow of fluid by the software. The advantage of CFD is flexibility, accuracy and breadth of application. It also acceptable in the industry as many researches use it to their research. Before this, there are various topic of microchannel that has been study. Many researches have already made the experiment on the effect of friction and the heat transfer rate in the microchannel. It can conclude that every journal is to find or increase the cooling rate in the MCHS.

2.1 Microchannel Heat Sink Past Research

The old method to dissipated the heat that generated by electronic device by using the air cooling was not enough to dissipated the heat. So that the new technology of microchannel heat sink was introduced by Tuckerman and Pease. They have fabricated the microchannel by using silicon as the material. The cooling fluid that be used was water to dissipated the heat that generate by electronic chip. The area of $1cm^2$ with the maximum thermal resistance is about 0.1 $^oC/W$. The highest heat flux was tested to 790 W/cm^2. They have confirmed that the flow rate will follow the Poiseuille's equation. They said that the water flow rate will be affected to the maximum thermal resistance because conduction from the front of the wafer to the channel region. (Tuckerman and Pease, 1981)

Gunnasegaran et al. was focus on studying about heat transfer characteristic of microchannel with different shape. The shape that they was study is rectangular, trapezoidal and triangular. The Reynold number that be tested in this study from 100 to 1000. From the result it can see that the rectangular shape has the highest of the heat transfer coefficient while the triangular shape is the lowest. The increases of Reynold number will rises linearly the pressure drop. The heat transfer and Poiseulle number also will increase by increase the Reynold number. For triangular, by increasing the tip angle of the channel will make the Poiseuille number. (Gunnasegaran *et al.*, 2010)

The three-dimensional heat transfer in microchannel is be analysis by Qu and Mudawar. They were designed a rectangular microchannel heat sink that use silicon as the material and used water as cooling fluid. The Reynold number they use is 140, 700 and 1400. The result of experiment is the average temperature is decrease and a higher average Nusselt number is obtained with the Reynold number is increase. They says that the effect of solid thermal conductivity on the average Nusselt number are so small between copper and silicon material. (Qu and Mudawar, 2002a)

Qu and Mudawar are doing an experimental and numerical study of pressure drop and heat transfer in single phase microchannel. They fabricated the rectangular microchannel heat sink by using an oxygen – free copper with 231μm wide and 713μm depth. The cooling liquid that be use is deionized water. The Reynold number at 139 to 1672 for heat flux at 100 W/cm^2 and at 385 to 1289 for heat flux at 200 W/cm^2. The results show that the largest pressure drop of 0.86 bar at the heat flux 100 W/cm^2 and Reynold number at 1672. The pressure drop for heat flux 200 W/cm^2 is lower to due to high water temperature will decrease the water viscosity. They says that the temperature will decrease by increase the Reynold number. It is expected that the fluid flow and heat transfer within channel should follow the Navier Stokes equation when the characteristic dimension is greater than 100μm and Reynold number is below 1700. (Qu and Mudawar, 2002b)

An analysis on the heat transfer characteristic of forced convection across microchannel heat sink by Zhao and Lu. They used two different approaches for this analysis that are fin approach and porous medium approach. The fin approach will contribute to a constant temperature distribution in the channel. While the porous medium approach will predict that fluid temperature will decrease from its maximum value at the bottom wall. Then it reach minimum and increase again because of the effect of the top wall temperature. (Zhao and Lu, 2002)

The investigation of heat transfer in rectangular are be made by Lee, Garimella and Liu. They experimental the rectangular microchannel with copper as the material using deionized water as working fluid. The flowrate is set from 0.1 to 2.2 l/min with the range of Reynold number is 300 to 3500. From the result that they were compare between the numerical prediction and experiment is be made. It seems that the Nusselt numbers from the experiment is higher than prediction. But it is accepted because its particularly close.

The dimensionless thermal axial distance is increases and it seem that the heat transfer coefficiency initially decreases sharplly. The higher heat transfer coefficient at low dimensionless thermal axial distance is because of the thinner boundary layer at high Reynold number (Lee, Garimella and Liu, 2005)

An experimental study is be made by Pate, Jones and Bhavnani on cavity induced two-phase in microchannel. The dimension of the channel is 200µm width and 346µm depth and the material that be used is silicon. The cavity is pyramidal shape and be located at the base of the channel. There are three value of mass flux that be study 535, 1069 and 2138 kg/m^2s. Based on the results, a maximum temperature overshoot that be recorded is 11.4°C for mass flux 535 kg/m^2s. they says that by increasing the value of mass flux will decreases the value of temperature overshoot. The cavities have the potential to produce a repeatable bubble flow and eliminate the potentially unstable slug formation. It also will affect the onset of nucleate boiling (ONB) by lowering the ONB flux and move the point ONB to the upstream. (Pate, Jones and Bhavnani, 2006)

Chein and Chen have study on the effect of performance in microchannel heat sink by change the inlet and outlet. They design five rectangular microchannel with a different inlet and outlet and use silicon as a material. There are N, D, S, U and V-type while I-type is a normal inlet and outlet. In I, N, D and S-type there are existed of recirculation of zone at the corner inlet and outlet while at U and V-type there are no recirculation zone due to a small vertical dimension of plenum. They says when the fluid velocity is low, a higher average of fluid temperature is resulted and the effect of fluid velocity will be at the entrance of microchannel. The result shows that the U and V- type have a better surface temperature uniformly compared to the I, N, S and D-type. (Chein and Chen, 2009)

Mohammed, Gunnasegaran and Shuaib were study on the effect of heat transfer and fluid flow in microchannel by using nanofluid. They used a rectangular shape for microchannel and aluminium-water (Al_3 O_3) as the working fluid. The range of Reynold number is 100 to 1000 and the heat flux is 100 to 1000 W/m^2. The particle of volume fraction is increase because of the nano particles is higher dynamic viscosity and lower heat capacity make the nano particles will reduce the temperature compare to pure water. But it only achieved when the heat flux is 1000 W/m^2 compare to 100 W/m^2 and 500 W/m^2. The heat transfer coefficient will enhance when the volume fraction increases from 0% to 2.5%. But when volume fraction is 5%, the nano particle will not enhance the heat transfer or performing as pure water. The pressure drop at low Reynold number is no difference with pure water but more significant at large Reynold number. (Mohammed, Gunnasegaran and Shuaib, 2010)

A microchannel with cross-flow synthetic jets have been study by Chandratilleke, Jagannatha and Narayanaswamy. The result that they get from this experiment is when the flow on the microchannel is increase will cause the jet to be swayed downstream. This will blocking the ability of the jet to penetrate through the boundary layer to the heater to bring about the favourable thermal characteristic. This jet technique is able to increase heat transfer in microchannel without higher flowrate or pressure drop. (Chandratilleke, Jagannatha and Narayanaswamy, 2010)

A study of liquid friction factor in microchannel heat sink by Steinke. He made a rectangular microchannel with the width of 200µm, depth of 250µm and length of 10mm. He used water as working fluid and the range of Reynold number is 14 to 789. He says as seen from the equation, the channel dimensions have the effect on the friction factor calculation. The result of the dimension that be taken from optical measurement technique seems that a microchannel width is 201±5 µm and depth is 247±5 µm. It seems that due to destruction, the profile that suppose to be a rectangular microchannel is actually found in a trapezoidal shape. (Steinke, 2016)

2.2 Microchannel with Cavities

Koşar, Kuo and Peles focus on the boiling of water in microchannel heat sink with reentrant cavities on the sidewalls. They make every channel wall, an array of 100 interconnected reentrant cavities are 100µm and the diameter of cavities is 50µm with 7.5µm mouth to enter the cavities. They says when the mass velocity increases the boiling mechanism will transition from a nucleate to a convective dominant mechanism increases. For low mass velocities (41and 83 kg/m^{2}s) will be considered as the nucleate dominant region. For the moderate mass velocity (166 kg/m^{2}s) at low heat flux (below 88W/cm^2) is catogeries to the nucleate boiling region. For mass velocity (166 and 302 kg/m^{2}s) are catogeries as convective boiling dominant. (Koşar, Kuo and Peles, 2005)

Kuo and Peles make an experimental about flow boiling instabilities in microchannel with re-entrant cavities. They created three design that nonconnected re-entrant cavity (1NR), interconnected re-entrant cavity (2IR) and plain wall microchannel (3PW). The 1NR an array of 100 re-entrant cavities spaced 100µm apart in sidewall of microchannel, the diameter is 25µm and 7.5µm of mouth to enter the cavities. 2IR is similar to 1NR except 20µm wide interconnection to connect the re-entrant cavities. The mass flux with range from 86 kg/m^2 to 520 kg/m^2. The result show that at low heat flux will existed of single-phase liquid flow that make the linear temperature increase with heat flux. The device with re-entrant cavities (1NR and 2IR) initially stable boiling until the heat flux reach a condition that the boiling instable with temperature fluctuation. The condition is be mark as onset of flow oscillation (OFO) and it corresponding when temperature fluctuation above than 5^{o}C. For 3PW the trend of boiling is different from the device with re-entrant cavities, it can see that the transition from onset of nucleate boiling (ONB) to critical heat flux (CHF)is more rapid and for mass flux is below 160 kg/m^{2}s the CHF boiling region is maintained. For 1NR the stable boiling is when the heat flux is between 81 to 99 W/cm^2

and the temperature after OFO is increase from 5 to 27^{o}C. For 3PW the OFO is triggered with ONB and initial temperature is 10^{o}C that will increase to 22^{o}C before the CHF condition. The stable boiling condition for 3PW is very limited, (Kuo and Peles, 2008)

Chai et al. says by adding the triangular re-entrant cavities can increase the heat transfer performance compared to straight microchannel. When the Reynold number is at 1015.96, the Nusselt number will reach 1.80 times higher than the straight microchannel. The heat transfer performance will be enhanced by increase the Reynold number, although the rising extent become smaller (Chai *et al.*, 2010).

The microchannel heat sink was added an offset fan shaped re-entrant cavities in the side wall by Chai et al. They used the computational fluid dynamic to simulate the flow and heat transfer in the heat sink. The comparison between the equation and the numerical prediction is it found that the numerical prediction is slightly lower that the equation and the maximum error is about 2.8%. The stagnation zone is form because of the flow of water in the heat sink is so slow. The stagnation zone will reduce the pressure drop by make the main flow slip over but drastically decrease the heat transfer. (Chai *et al.*, 2011)

Chai et al. make an experimental on the periodic expansion constriction cross sections in microchannel heat sink. They created a three design of microchannel, first is a microchannel with fan shaped re-entrant cavities (F) with the field angle of cavities is 120^{o} and 0.1mm of radius. Second is a microchannel with triangular re-entrant cavities (T), the depth of triangular cavity is 0.05mm and 0.06 mm for the length of expansion cross section and 0.14 mm for the length of constriction cross section. Third is a rectangular straight microchannel (R). All microchannel has a length of 20mm, a width of 10mm and a depth of 0.35mm and be manufactured by the dry etching based micro fabrication technologies. The comparison between numerical prediction and experimental data shows that an excellent agreement with the maximum difference are 1.02% for the R, 6.16% for F and 5.96% for T. They says that when Reynold number is below 300, the pressure drop of heat sink with periodic expansion constriction cross sections is lower than the heat sink R. the increasing of Reynold number will increase the Nusselt number value that make the heat transfer is enhanced for the periodic expansion constriction cross section and the proposed heat sink has a higher grow rate than the heat sink R. The heat transfer coefficient is compared with heat sink R and for heat sink F is increase 12.5-72.3% and for heat sink T is 21.5-80.4% for Reynold number between 147.23 to 752.34. (Chai *et al.*, 2013)

Xia, Zhai and Cui are investigated the characteristic of water when using a microchannel heat sink with fan-shaped re-entrant cavities and internal ribs (FRCR). The microchannel material is using silicon material and the relative rib height is adjusted with the range of Reynold number is from 150 to 600. The length, width and depth of the heat sink are 10mm, 3mm and 0.35mm. The result that they get is friction factor is decrease when they use a microchannel with fan-shaped re-entrant cavities without rib. When using the rib it is observed that the friction factor is higher than the rectangular microchannel. The friction factor is decreases linearly when the Reynold number increasing for rectangular microchannel and FRCR but only when Reynold number is below 800. For Reynold number is below 600, the measured friction factor in FRCR is 6.5 times higher than the rectangular microchannel. The location of vortices is formed have a low heat transfer coefficient while Nusselt number is reach maximum value beside the rib zones. The flow disturbance is increases at the rib zone because of the expansion of the flow area. They says that due to the flow characteristics, we can enhance heat transfer by increase the length of converging zones (rib zone) and reduce the length of the diverging zones (cavity zone). (Xia, Zhai and Cui, 2013)