THERMAL AND FLOW ANALYSIS OF HEAT SINKS FOR CPU COOLING BY ANSYS

INTERAN A/L RAMAN

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

THERMAL AND FLOW ANALYSIS OF HEAT SINKS FOR CPU COOLING BY ANSYS

INTERAN A/L RAMAN

This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering with Honours

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2019

DECLARATION

I declare that this project report entitled "Thermal and Flow Analysis of Heat Sinks for CPU cooling by ANSYS" is the result of my own work except as cited in the references.

Signature : ....

Name : INTERAN A/L RAMAN

Date : ....

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Honours.

Signature : ....

Name of Supervisor : DR. MOHD ZAID BIN AKOP

Date : ....

DEDICATION

To my beloved mother and father.

ABSTRACT

The technology development nowadays towards the revolution of industry 4.0, where the development of digital computer in advance model and it's needs are increasing day by day. When there are pros there will be cons as well, where the reliability of electronic components are getting affected extremely by the heat dissipation at which the junction operates. To work on higher processor units nowadays CPU needs a greater operating power, and as CPU design engineers there are forced to make the system smaller in the term of size because customer at this time wants everything to be in pocket size. As the size reduced the problem of transferring heat to the surrounding and controlling the temperature becomes vital. Currently the CFD simulations have become a problem solver where it's does not cost anything in the term of price and it's widely used in the studies of heat transfer. This report tells about the material that suit well to be heat sink. There are three type of material analyzed in this research, which is steel, aluminium and copper. The parameter such as heat dissipation, number of fins, fin thickness are taken from past experimental research, while the geometry of CPU chassis, CD, DVD, and HDD are taken from manufacturer data base. Besides that, in this research work the flow around the chassis has been analyzed to increase the cooling performance. The result of three different material type of heat sinks are compared in order to get the fine material that suit well to be heat sink fabrication material. Furthermore, the simulation of copper material heat sink shows a better characteristics in the simulation for heat transfer and having more inlet slots increase cooling capacity of heat sinks due to a large air flow inside the chassis.

ABSTRAK

Perkembangan teknologi sekarang ini ke arah revolusi industri 4.0, di mana perkembangan model komputer digital kian meningkat dan keperluannya semakin meningkat setiap hari. Apabila terdapat kebaikan, terdapat keburukan juga, di mana komponen elektrik dalam CPU mengalami masalah pemindahan haba yang ketara. Selain itu, untuk CPU bekerja pada prestasi penuh ianya memerlukan kuasa elektrik yang tinggi. Casis CPU juga pada masa sekarang making dikecilkan untuk mesra pengguna. Oleh disebabkan itu, reka bentuk model CPU lebih kecil dalam saiz. Lagi kecil casis CPU lagi susah untuk pemindahan haba di dalamnya. Ini disebabkan oleh ruang yang tidak cukup untuk udara sejukkan "heat sink". Pada zaman sekarang simulasi CFD telah menjadi sesuatu kaedah yang dapat menyelesaikan pelbagai masalah di mana ia tidak membebankan apa-apa dari segi harga dan ia digunakan secara meluas dalam kajian pemindahan haba. Laporan ini menceritakan tentang bahan yang sesuai untuk menjadi "heat sink". Terdapat tiga jenis bahan yang dianalisis dalam kajian ini, iaitu keluli, aluminium dan tembaga. Parameter seperti pelesapan haba, jumlah sirip, ketebalan sirip diambil dari penyelidikan eksperimen yang lalu, manakala geometri casis CPU, CD, DVD, dan HDD diambil dari jurnal masa lalu. Selain itu, dalam kajian ini, aliran angin di sekitar casis telah dianalisis untuk meningkatkan prestasi penyejukan "heat sink". Hasil daripada simulasi tiga jenis bahan tembaga menunjukkan hasil yang baik untuk menjadi "heat sink" yang bagus.

ACKNOWLEDGEMENT

Firstly, I wouldn’t be here if not for the mercies, favor, and blessings of my Lord and personal savior. I would like to thank God for seeing me through my journey here at Universiti Teknikal Malaysia Melaka so far and for the incredible and exciting adventures he has in store for me.

Besides that, I would like to say thanks to my advisor Dr.Mohd Zaid Bin Akop for his support and encouragement during my period of studying on my degree and this research. Also, huge thanks to Dr. Mohamad Shukri Bin Zakaria who guided me on the meshing of CFD fluid fluent while working on this project.

I would like to thank my course mates for giving me their support, patience and encouragement. Finally, I would like to thank my family for their support.

CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBEREVATIONS

ATX	Advanced Technology Extended
CD	Compact Disc
CFD	Computational Fluid Dynamic
CPU	Computer Programming Unit
DVD	Digital Versatile Disc
HDD	Hard Disk Drive
I/O	Input / Output
PC	Personal Computer

LIST OF SYMBOL

A	=	Area
^{o}C	=	Degree Celsius
h	=	Heat Transfer Coefficient
K	=	Kelvin
k	=	Thermal Conductivity
m	=	meter
Q	=	Heat Transfer
T	=	Temperature
W	=	Watt

CHAPTER 1

INTRODUCTION

1.1 Background

21st century is keep growing beyond limitation not only in technology but every aspects and each angles. The devices in the form of electronics are replacing the human minds due to the incredible reliability in our daily routines. However, when there are pros there also will have cons. Which is our modern era wants everything to be smaller if can in the size of a pocket. The smaller the electronics devices gets the higher the heat distribution will occur. The research is on the Central Processing Unit (CPU) heat sinks plate which functioning as a cooler for the electronic devices inside CPU. The heat occur on electronics devices such as integrated circuit (IC), resistances, capacitances, diodes, graphic cards, random access memory (RAM) and other smaller chips. This heat may not release to the air if there is no heat transferring component from inside to outside. This can lead to the damage on electronics component because electronic components do have their specific temperature before over-burn (The and Publishing, 2016).

To overcome temperature rises issue there are some ways to reduce the CPU electronic devices temperature. Which are fans, thermoelectric coolers, heat sinks, plates, forced air blowing systems and also using liquids. Generally, electronics devices on CPU use forced air cooling method with the heat sink plate. This is because its high capability in the terms of larger heat transfer rate, faster cooling, light weight, price, area occupied and reliability. Plate fin and pin fin heat sink concept are using widely. This is because its own special advantages. One of the advantages of plate fin is it has small pressure drop, easy to create and also the simplest design. The pin fin having a large heat transfer rate (Desai *et al.*, 2016).

In this project the evaluation will based on the computational fluid dynamics (CFD) approach. The CFD ANSYS CFX software used to identify a cooling solution for the desktop computer. The future modern CPU will dissipate heat in the range of 100-150W. High dissipation of heat can cause the personal computer (PC) to lag and also gives a high risk to the electronic components due to the high temperature. On behalf of that, heat sink with better dimensions of fins and design will needed for the cooling process inside the CPU. Figure 1.1 shows a simple heat sink which used as heat removal source inside PC (Zhang, Long and Zhang, 2018).

Figure 1.1: Heat Sink

(Source: wikipedia.org/wiki/Heat_sink)

1.2 PROBLEM STATEMENT

Nowadays CPU using more electric power around 100-150W. The higher the power it's uses the higher the heat dissipation rejected inside the CPU. When the heat sink materials thermal resistance is high the longer it's takes for the cooling process and this will drag the performance of CPU to the minimum or prostate characteristics mention above expected in future that will make the heat transfer rate more difficult. If a good thermal conductivity heat sinks material placed inside the CPU it will help the cooling process much more faster and also will increase the performance of CPU much greater.

1.3 OBJECTIVE

The objectives of this project are as follows:

1. To study the thermal heat distribution in CPU unit on the heat sinks caused by electronic components by analyzing experimental data using CFD simulation.
2. To compare and analyze steel, aluminium and copper, that suit perfect as heat sink production material.

1.4 SCOPE

The scope of this project are:

1. Based on the simulation approach by using ANSYS only.
2. Neglecting the heat transferring from the small electronic parts such as resistor, diode, transistor, capacitor, and the CD, DVD and also HDD is consider as lumped media.
3. The model of CPU based on the standard size only (standard ATX board is 12 x 9.6 in or 305 x 244 mm).
4. Only three types of material is considered (steel, aluminium and copper).

1.5 GENERAL METHODOLOGY

The action or steps that need to take out to achieve the objectives in this project are given below:

1. Literature review

   Journals, articles, books, internet webpages and any other resources that related to the project.

2. Dimensions gathering from journals

   The dimension of the plate-fin or heat sink is based on the standard size.

3. Simulation

   Computational fluid dynamic approach will be based on the ANSYS software.

4. Analysis and proposed solution

Analysis and data of the heat transfer rate and flows on how the heat transfer will be presented.

5. Report writing

A report regarding about the simulation will be written at the end of this project.

CHAPTER 2

LITERATURE REVIEW

2.1 Heat Sinks

Heat transfer has huge role in global, including electronic components, structures and also in industrial automation. The development to industrial 4.0 has lead the usage of electronics components increasing day by day. Besides that the components has exacerbated the heating problem. The smaller the object the larger the heat rejected by it and its absorption temperature also much smaller. So to overcome this issue the heat sinks can be use as solution. Figure 2.1 shows example of variety designed heat sinks (Strategia *et al.*, 2016).

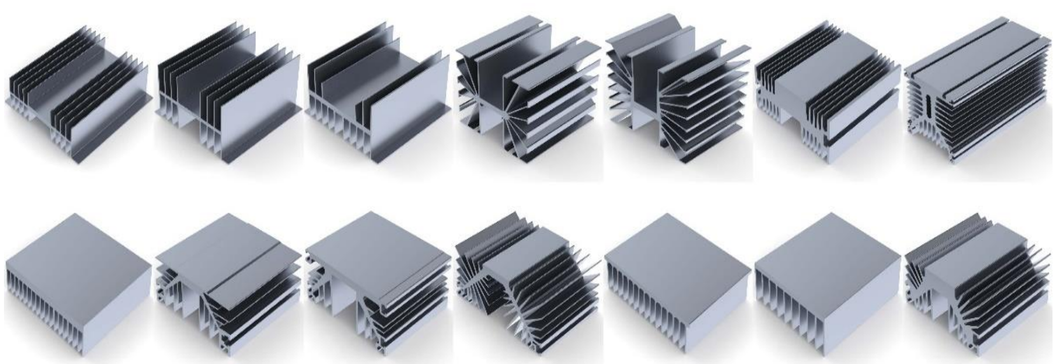

Figure 2.1: Types of Heat Sinks

(Source: vrelectronics.com)

Heat sink is an object that fabricated using custom design and aluminium alloy as material. The main objective of heat sink is to transfer the heat created by integrated chips (IC) and electronics products. Heat sink in computer programming unit (CPU) is used to cool down the graphics processor, random access memory, hard disk, and electronic components. This is because without the heat sink the electronic components might burn due to the overheat or long term operations (Haghighi, Goshayeshi and Reza, 2018).

To get a good heat sink model some characteristics should be consider such as the shape, material, angle of the heat sink placement, size, and lastly heat transfer rate for the material or thermal conductivity. Design of the heat sink is depend on the space and area that available but normally they maximize the area in order to get a larger heat transfer coefficient. The increase in surface area can lead to maximum absorption of heat and this can lead to fast cooling in CPU. However too long also not suitable. The design of giving gap in between the fins is helpful for the air to pass through the heat sink. Generally, the heat transfer of the created heat to the heat sinks happens through conduction between material and later the heat transferred to the surrounding by convection. The Eq [1] is for conduction heat transfer, while Eq [2] for convection heat transfer (Ali and Kurdi, 2018).

$$[Q = kA\ (T2-T1) \div L] \qquad [1]$$

$$[Q = hA\ (T2 - T1)] \qquad [2]$$

Aluminium is the heat sink material that widely used in CPU development. Aluminium has a greatest thermal conductivity around 229 W/m.K which is k in thermal language. Thermal conductivity is a rating which set to the material on how efficiently it can transfer the rate of heat transfer. The bigger the thermal conductivity (k) value the higher the heat transfer rate. Moreover, this can lead to instant cooling in electronic components ('Cfd Analysis of a Notebook Computer Thermal', 2008)

2.2 Computational Fluid Dynamics (CFD)

CFD simulation is a part of fluid mechanics study that uses numerical method analysis and data structures to perform calculation and give solutions to the problem that involves fluid flows, heat, mass transfer, chemical reactions (explosions) and related phenomena. Computer is a source that used to conduct the simulations with the surfaces defined by boundary conditions. A high capability computers can give us a better solutions and often required to solve the largest and most complex geometrics. Navier - Stokes equations, is used in CFD as foundation. Figure 2.2 below shows the flow involve in CFD. With CFD, the area of interest is subdivided into a large number of cells or control volumes. In each of these cells, the Navier - Stokes partial differential equations can be rewritten as algebraic equations that relate the velocity, temperature, pressure, and other variables, such as species concentrations, to the values in the neighboring cells. These equations are then solved numerically, yielding a complete picture of the flow down to the resolution of the grid. The resulting set of equations can then be solved iteratively, yielding a complete description of the flow throughout the domain (Uddin, 2009).

CFD is a simulation that used widely in research and mechanical engineering problems and also companies including in aerospace, aerodynamics, system design, combustion system, and also fluid flows to get solution in the term of numerical.

There are three stages in CFD which is :

1. Pre-Processing is the first step of the CFD simulation process which involving the geometry of the design. In this pre-processing process the needs to identify the fluid domain is necessary. The chosen domain later will divided into smaller grids, which called meshing. There are a lot of software in market which can be used such as Gridgen, CFD-GEOM, ANSYS, ANSYS ICEM CFD, and TGrid. This research using ANSYS to run the pre-processing process.
2. Solver Execution is a solving process after the physics problem has been identified in the pre-processing process. The flow physics model, fluid material properties, and also boundary conditions fixed to solve using a computer. Solver execution can be done by ANSYS FLUENT. ANSYS, Star CCM, CFD++, and OpenFOAM. Solver execution in this research will be ANSYS.
3. Post-Processing is a process of viewing the results in many way like contour plots, vector plot, streamlines and data curve. Each views will make the result finding more understandable and also for appropriate graphical representations report.

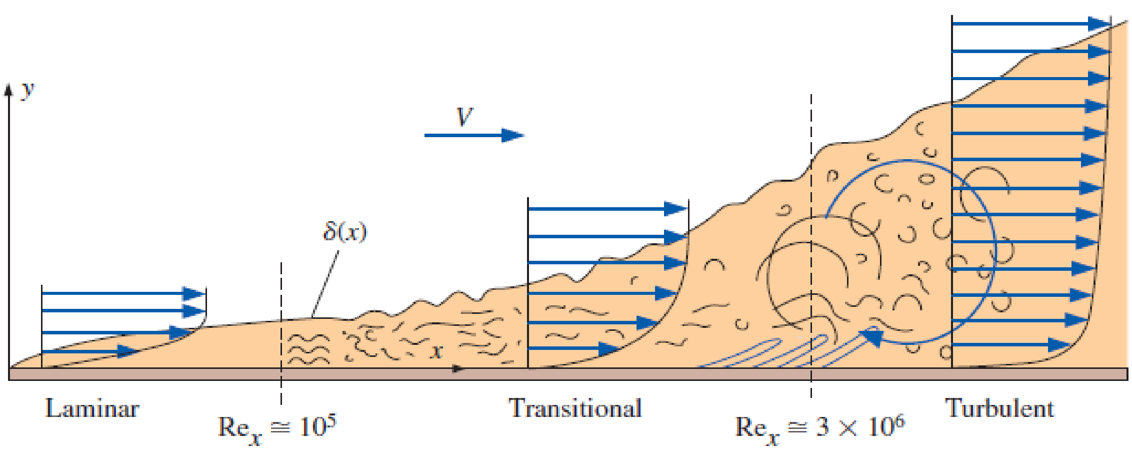

Figure 2.2: Physical characteristics of boundary layer

(Source: Burbuja.info)