ANALYSIS OF HEAT TRANSFER IN PORTABLE POWER SUPPLY

AHMAD NAZRIN BIN ALI @ YUSOF

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

SUPERVISOR DECLARATION

"I hereby declare that I have read this thesis 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 (Automotive)"

Signature: ……………………………………….

Supervisor: Dr. MOHD AZMAN BIN ABDULLAH

Date: 30 JUNE 2015

ANALYSIS OF HEAT TRANSFER IN PORTABLE POWER SUPPLY

AHMAD NAZRIN BIN ALI @ YUSOF

This thesis is submitted in partial fulfillment of the requirements for the Bachelor of Mechanical Engineering (Automotive)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

JUNE 2015

DECLARATION

"I hereby declare that the work in this thesis is my own except for summaries and quotations which have been duly acknowledged."

Signature: ………………………………

Author: AHMAD NAZRIN BIN ALI @ YUSOF

Date: 30 JUNE 2015

DEDICATION

I dedicated this thesis to my beloved family especially to my father and my mother, to my supervisor, Dr. Mohd Azman Abdullah and my friends

ACKNOWLEDGEMENT

First of all, thank to God for giving me health, strength and opportunity to successfully completed Final Year Project. Eventhough there are few problems that I counter during conducting this project, but with God guidance and blessing, I finished my project.

I would like to thank to my beloved family especially my father and my mother, who's always give motivation and never stop pray for my success. They are my light and the root for my strength.

I also would like to express my gratitude to my supervisor, Dr. Mohd Azman Abdullah, who greatly support and encourage me while conduct this project. He has generously spending his time and never stops giving guidance to me. Without him, I would never successfully complete this project.

I would like to use this opportunity to express my gratitude to all parties at UTeM for their great kindness and cooperation. Last but not least, I would like to thanks to all my friends that gives support and helping me for finishing this project.

ABSTRACT

Solar, wind, and fuel cell are some example of power source that are environmentally friendly. Hence, portable power supply product that uses solar energy as energy source was introduced. This product can supply electric energy to home furniture such as fan, lamp and drill. As the portable power supply product continue to grow, thermal issues are becoming extremely important for the product reliability. Portable power supply is a hand–held electronic equipment that is placed in a completely sealed enclosures due to safety factors. Since the cooling system of this product relies primarily on natural convection, the effective thermal management was quite low. Any electronic component will produce heat while operate and each electronic component have their own maximum temperature allowed during the operation. For this project, battery, adapter and inverter of the portable become the main heat source. Therefore, CFD model was develop by using ANSYS FLUENT software to study heat flow inside portable power supply. CFD model results were analysis and study to optimize thermal removal system of the portable power supply. Plus, experiments also will conduct for this project in order to study heat flow inside Portable power supply thoroughly. CFD model analysis show that by placed main heat source such as battery at the middle position compare to rear position will have better heat removal system. Besides that, by hollowed, added fan blower to the portable power supply, the temperatures inside the portable power supply will decrease and the efficiency of heat removal system will increase.

ABSTRAK

Tenaga solar, angin dan sel bahan api merupakan contoh sumber tenaga yang mesra alam. Oleh itu, produk pembekal tenaga mudah alih yang menggunakan solar sebagai sumber tenaga telah diperkenalkan. Produk ini boleh membekalkan tenaga eletrik pada perkakas rumah seperti kipas, lampu dan alat pengerudi. Apabila produk pembekal tenaga mudah alih semakin berkembang, isu sistem penyingkiran haba menjadi amat penting untuk ketahanan produk. Pembekal tenaga mudah alih adalah alat eletronik yang mempunyai struktur yang ditutup sepenuhnya disebabkan faktor keselamatan. Oleh kerana sistem penyejukan produk ini bergantung kepada perolakan semulajadi, keberkesanan pengurusan haba adalah amat rendah. Setiap komponen elektronik menghasilkan haba apabila digunakan dan setiap komponen elektronik mempunyai nilai suhu maksimum tersendiri untuk berfungsi. Bagi projek ini, bateri, adapter dan inverter di dalam produk pembekal tenaga mudah alih merupakan pembekal haba yang utama. Model CFD dibina dengan menggunakan perisian ANSYS FLUENT untuk mengkaji aliran haba di dalam produk. Hasil kajian model CFD akan dianalisis dan dikaji bagi mengoptimumkan sistem penyingkiran haba dari produk pembekal tenaga mudah alih. Tambahan, eksperimen juga akan dijalankan bagi projek ini untuk mengkaji aliran haba di dalam pembekal mudah alih dengan lebih teliti. Analisis model CFD menunjukkan bahawa dengan meletakkan sumber haba utama seperti bateri di posisi tengah berbanding posisi belakang akan mempunyai sistem penyingkiran haba yang lebih baik. Selain itu, dengan menebuk lubang dan menambah kipas pada produk pembekal mudah alih, suhu di dalam produk akan menurun dan kecekapan sistem penyingkiran haba akan meningkat.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

A	=	surface area
v	=	velocity
P	=	pressure
T	=	temperature
p	=	density
c_p	=	specific heat
k	=	thermal conductivity
μ	=	viscosity
v	=	voltage
^{o}C	=	celcius
cm	=	centimeter

LIST OF ABBREVIATIONS

FYP	Final Year Project
PSM	Projek Sarjana Muda
CFD	Computational Fluid Dynamic
Exp	Experiment
CFM	Cubic Feet Meter
CAD	Computer-aided Design
DC	Direct current
AC	Alternative Current
PPS	Portable Power Supply
IC, OC	Inlet closed, outlet closed
IO, OC	Inlet open, outlet closed
IC, OO	Inlet closed, outlet open
IF, OO	Inlet fan, outlet open
IO, OF	Inlet open, outlet closed
IF, OF	Inlet fan, outlet fan

LIST OF APPENDIX

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

Nowadays, there are a lot of researches and developments have been done in many fields in order to accommodate to the arising of technologies. Not forgotten, the environmental issues also become main topic as the technologies shine and arise. In order to develop and create product that meet both customer demand and green technology are very challenging. A portable power supply is a product that uses 12 V battery (Nicd) that recharges by solar energy and convert it into electrical energy. This product can produce 240 V and can supply electrical energy to home furniture such as fan, lamp, iron, electric drill and others. This product produces electrical energy without using any fuel and does not produce any pollutants like any type of generator that use fuel. Thus, this product develops in order to meet both customer demand and green technology.

1.2 PROBLEM STATEMENT

Portable power supply is machine that can be very useful to human. Consumers can get electrical power supply anywhere and anytime they want. This product focuses on green technology which uses battery that can recharge by the solar energy. But, the problem is the battery inside the product become hotter while being used or being charger. There is temperature limit for each component inside portable power supply to function properly. The increment of the temperature inside the machine could cause the machine to failure or damage and make it not safe to be used. CFD model are develop to study the heat flow in portable power supply. There are few elements that being used as manipulate variable such as battery position, window position and the present of blowing fan in order to optimize thermal performance and improve the reliability of the product. Besides that, experiment also will be conducted during this project in order to study heat flow in PPS thoroughly.

1.3 OBJECTIVE

The purpose of this project is to develop CFD model and study heat flow in PPS. CFD modeling is a faster and more efficient tool than experimental cut-and-try methods for improving and optimizing the thermal management for high density power converter. However, experiment still be conducted to determine temperature inside PPS in order to get more data for heat transfer study. Generally, every electronic component produce heat while operate and each electronic component has their own maximum allowable operating temperature. It is important to optimize thermal management to avoid machine failure and increase reliability of the machine.

1.4 SCOPE

There are four scopes need to be done for this project. The first one is develop CFD model by using ANSYS software. Second, use few elements as manipulated variable such as battery position and fan blower present while setup the parameter. Another scope is conducted the experiment that use inlet, outlet hole and fan present as manipulated variable. The last scope for this project is study and analysis heat flow of portable power supply.

1.5 THESIS OUTLINE

The project background, problem statement, objectives and scope of this research have been discussed in this chapter. The research of this project generally divides to two which are develop CFD model and conduct experiment in order to study heat flow inside the model. First of all, this project will focus on CFD simulation of the model by using ANSYS FLUENT software. CFD modeling is a faster and more efficient tool than experimental methods for study heat flow. Plus, CFD modeling is more simple and cheap rather than experimental method. Three models are develop for this project. Model 1 and model 2 will use battery position as manipulative variable. Model 3 is developing to compare temperature of CFD result with experiment result. Besides that, there are three stage of the experiment that conduct for this project which are stage 1 (before hollowing), stage 2 (after hollowing) and stage 3 (with fan). Second chapter will discuss about study research that help and support while investigate this project.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In this chapter, the research resources and literature review in the related field will be discussed thoroughly as it is an important step in every or whatever project to be done. It is an important method in order to get more understanding of this project and to gain ideas and improvements of the project. The literature review will be discussed in three subtopics to give clear view and understanding of the project, which are develop CFD model using ANSY software, changing the parameter of the setup and study the heat transfer. Analysis of heat transfer in portable power supply is a project that study heat transfer of the model that been researched and development which is still not completely efficient for use due to thermal removal problem. As the densities of the portable power supply continue to grow, environmental and thermal issues are becoming extremely important to the product quality.

2.2 CFD ANALYSES OF A NOTEBOOK COMPUTER THERMAL MANAGEMENT SYSTEM

According to Tari and Yalzin (2010), the increasing energy costs and consumer awareness on environmental issues together with the advances in related electronic components shifted the demand in the personal computer industry towards notebook computers which have low energy consumption and less noise compare to desktops. But, due to the compact chassis, thermal management of a notebook is more difficult than a desktop. Heat dissipation to ambient air from a typical desktop computer can be easily achieved using forced convection due to greater air volume inside the chassis for circulation and large chassis surface area for placing the vents and fans. This is different for notebook computer which should be small as possible in term of size and weight. Therefore, there is not much space left inside chassis among the components that are placed very tightly. Every electronic component produce heat while operate and each electronic components has their own maximum allowable operating temperature. For notebook computer, CPU is the main heat source and produce most of heat among others component. But, even the smallest heat dissipation should be considered, because the components are packed together inside the chassis very tightly and component temperatures depend on all heat sources that produced. It is important to optimize thermal management to avoid machine failure and increase the reliability of the machine.

The first part involves a CFD analysis of a notebook computer. Electronic component boxes and computer systems of various sizes have been numerically modeled and CFD simulation results have showed very good agreement with experimental test result. In addition, CFD simulation is fastest and efficient tool to study thermal flow of the model. In the study, the cooling system of notebook computer that was Sony Vaio PCG-GRX316MP is numerically investigated to access the heat loads on the active and passive paths. For the chassis as shown in Figure 1; CPU, CPU heat sink, heat pipes, heat exchanger, fans, aluminum heat dissipation plates, RAM, DVD, battery, PCMCIA card, hard disk drive (HDD), speakers, ventilation holes, PCB and miscellaneous cards attached to PCB are modeled according to measured dimensions. The components with little effect on the fluid flow and heat transfer are not modeled. Plus, there are two fans in the system, the first one is Fan 1 used to provide airflow to the fin-tube type remote heat exchanger (RHE) and the second one is Fan 2 that attached to an aluminum heat dissipation plate on the graphics chip and south bridge. The RHE is attached to the condenser ends of two heat pipes which transfer heat from the heat sink attached to the CPU. The heat pipes are represented as the solid rods having the same physical dimensions with the actual heat pipes and a high thermal conductivity in the axial direction taken as 40000 W/m.K.

For setup solution in CFD simulation, no slip boundary condition is applied for the chassis and the component walls in the domain. The heat transfer mechanism outside the notebook chassis was assumed to be natural convection and the convective effects of the flow coming through the fan exits and the ventilation holes are neglected in CFD modeled. The ambient temperature was 25 °C. Inside the chassis, radiative transfer and compressibility effects are neglected. With a conservative approach, radiative transfer from outer surfaces of the chassis to the surroundings was also neglected in this experiment. The simulations using the model of the notebook were performed for five different cases shown as **Table 2.1**. Case I corresponds to the standard use of the notebook. Case II is for the standard use while charging the battery of the notebook. Case III corresponds to the optical drive use. Case IV is for 100% CPU load. Case V is the extreme case that stresses the thermal management system of the notebook.

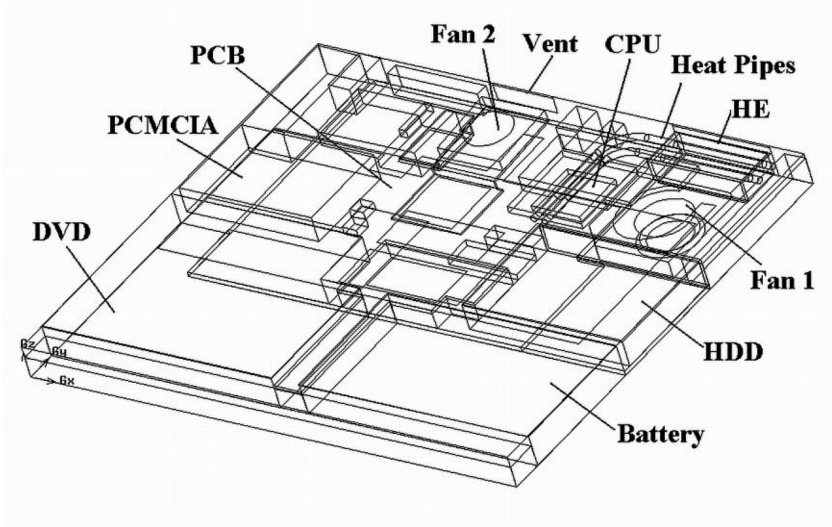

Figure 2.1: Computational domain (Tari and Yalzin ,2010)

Table 2.1: Heat dissipation values, average and hot spot temperatures of the components (Tari and Yalzin, 2010)

Component	CASE Heat Dissipation (W)					T_{max} (°C)	CASE CFD Avg. T (°C)					CASE CFD Hot spot T (°C)				
	I	II	III	IV	V		I	II	III	IV	V	I	II	III	IV	V
CPU	21	21	21	30	30	100	53.5	53.9	54.6	65.6	67.1	57.8	58.2	58.9	71.8	73.2
RAM	0.5	0.5	0.5	0.5	0.5	70	43.2	44.3	47.2	50.0	54.9	45.7	46.6	48.2	54.3	57.7
HDD	5	5	5	9	9	60	47.9	49.5	49.9	61.2	64.3	49.1	50.7	50.9	63.6	66.6
Graphics card	2	2	2	2	2	85	44.8	45.3	47.7	49.9	53.4	45.2	45.7	48.3	50.4	53.9
South bridge	0.5	0.5	0.5	0.5	0.5	85	43.8	44.5	46.9	49.9	53.6	44.2	44.9	47.2	50.8	54.5
PCMCIA	1	1	1	1	1	70	40.8	41.5	45.6	45.2	50.4	41.4	42.1	46.1	46.2	51.4
DVD	0	0	5	0	5	60	35.3	36.4	47.2	38.6	51.2	41.3	42.9	48.5	48.0	54.4
Battery	0	2	0	0	2	55	34.9	42.0	37.4	39.4	48.5	45.0	47.3	47.3	55.6	59.9
PCB	0	0	0	0	0	70	43.2	44.1	46.8	50.0	54.4	57.1	57.6	58.2	69.8	72.3
TOTAL	**30**	**32**	**35**	**43**	**50**											