'I admit that have read

this report and to my opinion this report

fulfill in terms of scope and quality for the certificate of

Bachelor of Mechanical Engineering (Thermal-Fluid)'

Signature   : ⟨signature⟩

Name of Supervisor 1: Puan Ainil Jesita Binti Jalaluddin

Date   : 13 May 2008

THERMAL COOLING OF COMPUTER HARDWARE

NURUL ASIKIN BT MOHAMAD

This report is submitted in partial fulfillment of the requirement for the
Bachelor of Mechanical Engineering (Thermal-Fluid)

FACULTY OF MECHANICAL ENGINEERING
UNIVERSITI TEKNIKAL MALAYSIA MELAKA (UTeM)

May 2008

I hereby declare that this project report is written by me and is my own effort and that no part has been plagiarized without citations.

Signature   : *Nurul*

Name of Writer   : Nurul Asikin Binti Mohamad

Date   : 13 May 2008

ACKNOWLEDGEMENT

First and foremost, all the praise to Allah the Al-mighty for his blessings and benevolence, that his project has been successfully completed. By the Grace of Him and His Knowledge that He granted me, I have been able to write this humble report to the completion.

With sincere gratitude and appreciation, I would like to thank my supervisor, Mrs. Ainil Jessita binti Jalal for her valuable guidance and encouragement throughout the accomplishment of this project. All her advice and ideas have contributed to the success of this project. It's been an honor for me to working under supervision of such a great person like her.

I would like to thanks a lot to my beloved mum and all my family members for all your affectionate caring, and above all your sacrifices and prayers, moral and material support accorded to me until successful completion of this project. Last but not least, I am happy to thank my fellow friends, my housemate, and individual who is directly or indirectly contributed to this project. Thank you very much and may Allah bless all of you.

ABSTRACT

The air flow analysis of considering heat-generating components in a three dimensional desktop computer was investigated in this paper. A well known computational fluid dynamics (CFD) was employed to simulate the desktop computer chassis enclosure. The CFD software that used is the ANSYS Workbench and ANSYS CFX 10.0. The parameters are focused on the outlet air velocity and inlet velocity. The CFD analysis result will confirm by the experiment. This paper describes the methodology of CFD analysis for the desktop computer design, and describes the experimental details to predictions. The comparison result for both methods will be discussed in this paper.

ABSTRAK

Kertas kerja ini mengkaji tentang pengaliran halaju udara di dalam ruang *CPU* computer yang mengandungi beberapa komponen-komponen yang menghasilkan kadar pembebasan haba yang tinggi. Dua kaedah akan digunakan bagi menganalisa kadar aliran udara ini. Kaedah yang pertama adalah dengan melakukan eksperimen , dimana halaju di bahagian kemasukan dan pengeluaran serta suhu bagi beberapa komponen yang dikenal pasti telah mengeluarkan haba yang tinggi telah direkodkan. Nilai-nilai yang didapati dari eksperimen ini akan digunakan untuk membuat simulasi menggunakan program CFX versi 10.0. Keputusan yang diperolehi dari kedua-dua cara ini akan dikaji dan dibincangkan di dalam kertas kerja ini.

CONTENT

LIST OF TABLE

LIST OF FIGURE

CHAPTER I

INTRODUCTION

1.0 Background

The central processor unit (CPU) is an important component in a computer system. A lot of small components are located inside the CPU such as the motherboard, heat sinks, circuit boards etc. These components generate heat, so they must be cooled to make sure their temperature does not reach the maximum allowable temperature value. Typical limiting CPU case temperature are in the range of 70 to 85 °C but it is depends on the different CPU types.

The CPU is cooled using heat sinks which operate by carrying the heat either by convection or radiation. Heat sinks must have good heat conduction properties such as aluminium or copper, and placed near to the heat sources. The heat sink has metal fins of thin cross sections to increase the area for heat dissipation. A motor driven fan may also be placed on top so that it draws the hot air away faster than natural convection.

The heat dissipating by components inside the computer will affect the efficiency of the computer. The heat dissipating system includes the chassis, a motherboard which has a set of heat sink module installed on its top surface, and a power supply with a cooling fan.

The inlet air will first pass through the thermal module and removes the heat generated by the CPU, then flows to the heat sink and removes the heat generated by the CPU chipsets. Finally the air will flows into the power supply and removes the heat generated by power supply.

1.1 Objective of the research

The objectives of this research are:

a) To develop a simulation of air flow inside the CPU chassis based on practical measurement,

b) Compare experimentally and simulation for the velocity at the outlet vent of the chassis.

1.2 Problem statement

Newer computers come with larger memory and faster speed due to the customer needs. The increase of the speed, processor and memory will consume more power and raise the temperature inside the system. Overheating of CPU components will affect the stability, speed and performance of the computer.

The ambient or the room temperatures also affect the performance of the computer. If the inlet air temperature is high, it will fail to cool the heat generated by the components inside the chassis. Computers that suffer this problem will become slow and unstable. They will sometimes lock up or *hang* unpredictably. The life cycle for the component, especially the electronic components will decrease.

1.3 Scopes of the Study

The scopes of this research are:

a) Gather experiment data for input and validation of CFD simulation

b) Generate CAD drawing of computer hardware

c) Simulate thermal cooling of computer hardware and compare with collected experimental data.

d) Investigate effect of different turbulent model available in CFX.

1.4 Limitations

The research has been limited to:

a) The desktop computer that have been used consist of Cooler Master Wave Server Chassis, P5W deluxe Asus Motherboard and Intel core2duo E6600 processor

b) Several assumptions have been made in the simulations such as the internal components are represents as lumped object. The turbulent air flows were assumed. No buoyancy effect and no solid conduction included.

1.5 Expected Outcomes

During project researching and analyzing, the expected outcomes are:-

a) The error between the comparison experimental data or results of calculation and simulation model less than 10%.

1.6 Project organization

The first part of the report contain of the outline, objectives and scope of the research. This will follow by the summarization of the researches from the previous studied. In chapter three the methodology of the experiment and the simulation will be stated. The next chapter will discuss the tests that were carried out and show the results obtained. The final chapter will conclude the project

CHAPTER II

LITERATURE REVIEW

2.0 Introduction

Literature review is based on term on related topic that was studied and has been discussed by a professional and has given a complete explanation on how the things of study work out. From research, there are several journals that related with this research.

The previous efforts to simulate three dimensional modeling computational modeling of air cooled desktop computer can be found in several publications. J.S. Chiang, S.H. Chuang and Y.K. Wu in their journal had done numerical simulation of heat transfer in a desktop computer with heat generating component. J.Y. Chang and C.W. Yu works on identification and simulation of minimum air flow design for desktop computer

Lee and Mahalingam used Flotherm code to simulate detailed flow and temperature fields within a computer chassis with two fans. They also measured the temperature data at selected locations to verify the computational results. A research to evaluate the velocity and the temperature fields of air flow in a computer system enclosure using computational fluid dynamics (CFD) was done by T.Y. Tom Lee.

2.1 Heat transfer

The heat transfer method in this research will be heat transfer primarily by convection. Several components and printed circuit board (PCB) inside the chassis that generate heat transfer heat by conduction. Since the components inside the chassis was assumed as a lumped object, heat transfer by conduction is neglected. Therefore only heat transfer from the components to the air, that is convection, is considered.

The air inside the enclosure flows through the inlet and exhaust fans. Therefore the heat transfer in both the inlet and outlet boundaries are considered as forced convection. So, the buoyancy will also be neglected.

2.1.1 Conduction heat transfer

Heat transfer by conduction can be defined as the energy transfer from the high temperature region to the low temperature region when a temperature gradient exists in a body.

$$q \;=\; @kA\,A^{\Delta T.}{}_{\Delta X}$$

Where; q = The heat transfer rate

$\Delta AT + \Delta Ax$ = The temperature gradient in the direction of the heat flow

2.1.2 Convection heat transfer

Convection heat transfer is heat energy transfer between a solid and a fluid when there is a temperature difference between the fluid and the solid convection above a hot surface occurs because hot air expands, becomes less dense and rises.

The temperature of solid due to an external field such as fluid buoyancy can induce a fluid motion. This is known as natural convection and it's a strong function of the temperature difference between the solid and the fluid. Blowing air over the solid by using external devices such as fans can also generate a fluid motion. This is known as force convection

In this research, several components inside the chassis will have the heat transfer by conduction. The heat transfer from the components to the air can be defining as convection heat transfer. Since the focus of this research is to predict air flow and temperature, no solid conduction was solved.

2.2 Air velocity

The actual air flow rate delivered from the fans was unknown and controlled by the flow impedance of the enclosure. To estimate the actual air flow rate, it is necessary to adjust the flow rate to correspond to the predicted pressure rise across a fan to match the given fan characteristic (provide by manufacturer).

Three flow rates (34, 51 and 68 m³/h) were simulated, and the corresponding pressure rises across the fan were calculated. Results of the intersection of the fan characteristic curve and fan operating curve determined that actual flow rate at the fan to be determined that the actual flow rate at the fan to be approximately 51 m³/h, as shown in figure 1 below. This value gives an average normal velocity of 1.71 m/s at the fan inlet.
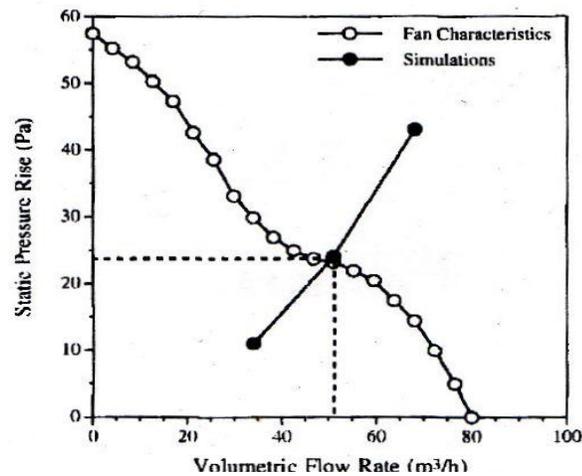
Figure 2.1 Determination of actual flow rate at fan

(Source: T.Y. Tom Lee (1994))

In this research, the inlet air velocity will be measured using air velocity meter. This is more accurate than estimating the average using method above.

The other way to used the hydraulic characteristic curve as shown in figure 2.1 is by using the equation of the form

$$\Delta P = K \frac{\rho u_i^2}{2}$$

Where; ΔP = pressure different

ρ= inlet air density

u= inlet air velocity

The equation is used to account for flow losses across the front air inlet. Based on J.Y Chang study, the loss coefficient was set at K=1.3.

2.2.1 Air flows
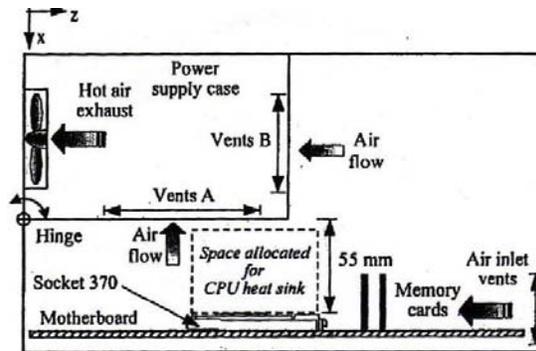
Figure 2.2 layout of major chassis component around CPU

(View from the bottom of the chassis)

(Source: J.Y. Chang (2000))

The figure above shows the flows of the air in the chassis. Air velocity and the pressure drop will be measure at the inlet vents and at fans. In actual system operation, the fan speed is controlled by a temperature sensor in the power supply case. This reduces air flow and fan noise when the system is not fully loaded. During actual system operation, the fan speed operated at 3300 to 3500 rpm. The account for fan speed variation on the fan curve, the following fan laws were used

$$P_1 = P_2 \left(\frac{n_f}{n_2} \right)^{2f} \left(\frac{\rho_f}{\rho_2} \right)^{g}$$

$$v_1 = v_2 \left(\frac{n_f}{n_2} \right)^{g}$$

These equations apply for the fixed fan diameter. The estimated fan curve a certain rotation will be obtain using hydraulic characteristic curve as shown in figure 2.2. The

2.3 Temperature measurement

The temperature in the chassis and at the selected location will be measured. During the experiment, several T-type (cooper-constantan) thermocouples (with ±0.1°C uncertainty) were attached along the component side of the powered PCB. The front ends of this thermocouple were titled normal to the PCB, and the tips were positioned in the air approximately 2 mm above the components.

From these several thermocouples, air temperatures between boards were measured. Experiments were run to estimate the proportion of heat dissipated through the noncomponent side of the board. This was done by measuring board temperatures on the both sides and then calculating the conduction heat loss through the board. Board temperatures were measured by additional T-type thermocouples that were positioned on both sides of the powered PCB. The tips of the thermocouples were bounded against the board by the high-conductivity epoxy. The locations of these thermocouples on the PCB are shown in figure 2.2.

The entire computer chassis was put inside a temperature-controlled and the temperature was set to 20°C. The computer system was turned on and the power was supplied to the resistors and multi fans. When the temperature reached steady state, five consecutive readings at each thermocouple position were recorded and averaged.
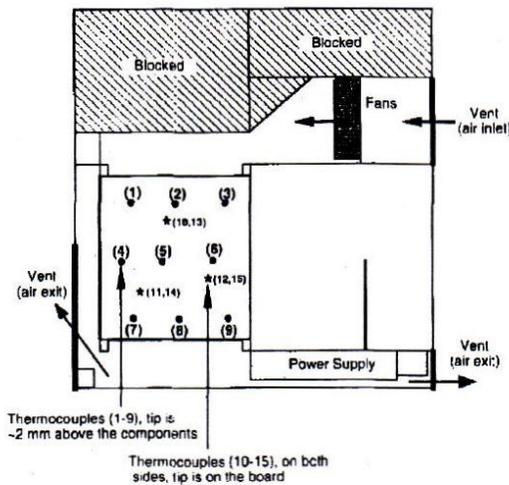
Figure 2.3 thermocouples location

(Source: T.Y. Tom Lee (1994))

The temperature of the CPU chipsets was measure to determined the heat dissipated by the component. A thermocouple was installed in a machined groove into the copper case. A 4mm square groove was machined up to the center of the copper case. A stainless steel sheathed J-type thermocouple (0.25mm diameter) was placed into the grooved and the gap was filled with thermally conductive epoxy to make it flush with the CPU case surface.

In this research, the temperature will be measure at 6 different locations in the chassis. The CPU chipset also installed with the thermocouple. The chassis temperature will be in steady state before the reading was obtained.

2.4 Simulation

CFD program PHOENICS was used to simulate the thermal behavior inside an enclosure. This program is base on structured mesh and finite volume formulation to discretize the Navier-Stroke equations.

The grid distributions are non-uniform and refines in the vicinity of high power components. Three components of Navier-Stroke equations, energy equation, turbulent kinematics energy equation κ^{a} and turbulent energy dissipation rate ε^{a} equation are coupled and solved together.

2.4.1 Basic assumption

Three-dimensional, steady state turbulent air convection in the electronic enclosure was assumed. The Reynolds number based on the fan inlet velocity and the air gap between the PCB's was larger than 3000. Preliminary studies assuming laminar air flow overestimated air temperatures and created large errors when compared to the experimental data. The flow may be still in the transitional region between PCB's. However, the software can only choose either laminar or turbulent model, and the turbulent air flow was assumed subsequently. Constant properties for air were assumed in all computational domains. The external surfaces of the enclosure were assumed to be adiabatic. For the outlet vents, a constant external static pressure boundary condition was applied. For the inlet vent, a constant external stagnation pressure boundary condition was applied. Since either a grille or a perforated plate was located at the vent, the pressure drop across the vent must be considered. This pressure drop (ΔP) is given by

$$\Delta P = \frac{f l}{b} \rho u_i^2 \frac{g}{2}$$