"I declare that I had read this thesis and according to my opinion, this thesis is enough to fulfill the purpose for award of the

Bachelor Degree in Mechanical Engineering

from the aspects of scope and quality"

SIGNATURE : ______________________

SUPERVISOR: DR. MOHD. YUSOFF BIN SULAIMAN

DATE : 30/5/06

HEAT AND MASS FLOW PREDICTIONS THROUGH A COMPACT HEAT EXCHANGER UNIT (CHE) WITH INDUCED FLOW USING CFD MODELING.

FADHLIZA ABDUL LATIP

This thesis is submitted to the Faculty of Mechanical Engineering as a partial fulfillment of the award of Bachelor Degree in Mechanical Engineering

FACULTY OF MECHANICAL ENGINEERING
KOLEJ UNIVERSITI TEKNIKAL KEBANGSAAN MALAYSIA

MAY 2006

"I hereby declare this thesis entitle

"Heat and Mass Flow Predictions through a Compact Heat Exchanger Unit (CHE)

with Induced Flow Using CFD Modeling".

Is the result of the work of myself except for the references which

I had clarified the sources"

SIGNATURE : ____________________

WRITER : FADHLIZA BINTI ABDUL LATIP

DATE : 20 MAY 2006

This book is dedicated to my mom, Merliah Mohamad and to my dad, Abdul Latip Hj. Kadir for their love, support and dedication and commitment to family. Both of you are the best thing I ever had in this world. To my brothers and sisters your supporting will never end.

A special dedication also goes to my beloved friends, for their care and support.

ACKNOWLEDGEMENT

I would like to express my grateful appreciation to the many individuals who has directly and indirectly contributed to this book. First and foremost to my parents; Abdul Latip Hj. Kadir and Merliah Mohamad for not being tired giving me supports. I also thank to Dr. Mohd. Yusoff B. Sulaiman, my supervisor who are provided an excellent suggestion, valuable comments, criticisms and praise. Thank are also due to my course mates, BMCT and BMCS for their patience, understandings and support through the preparation of this text, although I get into some hard condition.

ABSTRACT

This study is for the predictions of heat and mass flow distribution upstream and downstream of compact heat exchanger unit with induced flow. The induced flow is been created by the fan geometry that will be developed in this project. The fan geometry will design by using Computational Fluid Dynamics (CFD). The PHOENICS-VR had been chosen as mechanism on doing the designation. It is because the flow obstruction and pattern for the compact heat exchanger unit been simulate. Upstream is the flow at the leading edge while downstream is the flow at the trailing edge. The flow distribution was shown in a contour color which different color give different value. It is not only the attraction of doing the project but also as an easy way to understand what actually happen to the tested design. The heat transfer mechanism that involved is called forced convection. Heat transfer by convection involving fluid and solid surface. Forced convection happens when motion of the fluid is imposed externally.The predicted data will compare to the experimental data that been acquired from other.

ABSTRAK

Secara keseluruhannya, projek ini adalah berkaitan tentang ramalan penyebaran haba dan jisim melalui unit padat penukaran haba dengan aliran yang dihasilkan oleh sesuatu. Aliran akan dihasilkan menggunakan kipas geometri yg akan dibangunkan menggunakan rekabentuk bendalir berkomputer iaitu *Computational Fluid Dynamics (CFD)*. Software PHOENICS VR dipilih sebagai pamudahcara dalam pengendalian projek ini. Hal ini adalah kerana proses penghalangan dan corak aliran hendak disimulasikan dalam rekabentuk ini. Mekanisma pemindahan haba yang terlibat dalam projek ini adalah mekanisma perolakan paksaan. Penyebaran aliran jisim ditunjukkan dalam kontur warna di mana, setiap warna yang berbeza mempunyai definisi nilai yang turut berbeza. Ini bukan sahaja sebagai satu daya tarikan dalam pengendalian projek ini, malah satu kemudahan untuk pemahaman bagi perlakuan ygang berlaku pada rekabentuk yang diuji dan dikaji. Pemindahan haba secara perolakan melibatkan bendalir iaitu angin dan permukaan pejal iaitu plat. Perolakan paksaan berlaku apabila pergerakan bendalir didesak secara luaran. Data yang diramalkan dibandingkan dengan data yang diperolehi secara experimentasi yang dijalankan oleh orang lain.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURE

NO. OF FIGURE	TITLE	PAGE

NO. OF FIGURE	TITLE	PAGE

LIST OF SYMBOLS

SYMBOL	DEFINITION
F_p	fin pitch
K	temperature in Kelvin
L_c	louver gap
L_p	louver pitch
M_{LD}	mass concentration
N	the fan speed (rpm).
Nu	Nusselt number
Nu_{LD}	Nusselt number for a single-phase air flow
Pr_L	Prandlt number
q	Heat transfer amount
q_{max}	Heat transfer maximum amount
Q	volumetric flow rate (cfm)
Re	Reynolds number
Re_{Dh}	Reynolds number- hydraulic diameter
Re_{Lp}	Reynolds number- louver
T	temperature
t	thickness
U_m	velocity of air

GREEK	DEFINITION
θ	angle in degree
ε	heat transfer effectiveness
ρ	density of air
μ	dynamic viscosity
τ	the fan torque (lbf-in)

SUBSCRIPT	DEFINITION
$_{max}$	maximum

LIST OF APPENDIX

CHAPTER 1

INTRODUCTION

This study is basically about predicting a numerical heat and mass flow distribution upstream and downstream. The predicted flow is through a compact heat exchanger unit with induced flow using computational fluid dynamics (CFD) modeling. Then, the predicted data need to be compared to the data which is acquired from the experiment.

Since this study involving two components which are fluid (air) and solid (plate), the heat transfer mechanism involved is convection. Convection is the transfer of heat within of fluid by the mixing of one portion of the fluid with another. The induced flow produced by a fan modeling classified the convection as a forced convection. The forced convection experienced when motion of the fluid is imposed externally.

External fluid motion created by the fan, designed using Computational Fluid Dynamics modeling. The air flows though the fan also is modeled. The air will flow through the compact heat exchanger and the flow profile / pattern will be simulated. The numerical prediction about flow distribution can be acquired according to the simulation.

Besides focusing on the flow distribution obstruction pattern, this study also stress on using the computational fluid design (CFD) software. This is one of the important parts in engineering field. This exposure can be the additional value to the participant which is not everyone had experienced.

1.1 PROJECT TITLE

This project was entitle, "*Heat and Mass Flow Predictions through a Compact Heat Exchanger Unit (CHE) With Induced Flow Using CFD Modeling*".

1.2 OBJECTIVE

In this study, the main objective is to predict heat and mass flow distribution upstream and downstream numerically. The prediction is through a compact heat exchanger (CHE) unit with induced flow by using Computational Fluid Dynamics (CFD) modeling.

However, on archiving the successful of the study, this project was divided into two parts;

i) PSM 1

ii) PSM 2

1.2.1 OBJECTIVES FOR PSM 1

1.2.1.1 To prepare for PSM 2

1.2.1.2 To collect information about the project and totally understand what is the purpose of the study.

1.2.1.3 To study the theory that is related to the case study

1.2.1.4 To do some literature review on the project case study

1.2.1.5 To search as many as reference could be helped in the case study

1.2.1.6 To fulfill the requirement in completing the course

1.2.2 OBJECTIVES FOR PSM 2

1.2.1.1 To continue PSM 1

1.2.1.2 To design the model using chosen software

1.2.1.3 To simulate the model and get the result

1.2.1.4 To analyze and discuss the data that had been obtained

1.2.1.5 To fulfill the requirement in completing the course

1.3 SCOPE

There are several scopes as the limit or actions need to be covered in reaching the successful of this research.

1.3.1 Design a CFD fan geometry
1.3.2 Model air flow through a fan
1.3.3 Simulate the flow obstruction for the CHE unit. (Project schedule)
1.3.4 Simulate heat and flow pattern for a CHE unit
1.3.5 Acquire experimental data and compare result

1.4 IMPORTANCE OF STUDY

Forced convection can be applied in unaccountably application in mechanical field. This research can be study the performance of heat exchanger. Thus, we can improve the existence unit. By using CFD software, we will able to redesign the fan model to get the better result.

1.5 PROBLEM STATEMENT

The goal of this study is to get numerical predictions of heat and flow distribution upstream and downstream of compact heat exchanger unit with induced flow. The compact heat exchanger is a car radiator and the induced flow will created by the fan. In air-cooled CHE, fan is an important part. In designing the fan, there are several factors needed to be considered in order to get the required speed and flows are number of the fan blade and shape of the blade. The aim towards which the work has to do for this project is to finish within the timeframe with outstanding quality.

CHAPTER 2

THEORY

The heating and cooling have been the most important processes in the engineering field. The applications are uncountable. Heat and mass flow distribution are directly involved in these processes.

2.1 Heat Transfer

Heat flow is other process by which the internal energy can change it's through. The concept is more subtle and covers a range of physical processes. We can summarize that heat flow is the exchange of internal energy between thermodynamic systems. No mechanical work involve in this processes. It is mainly a positive quantity for thermodynamic processes by which heat is converted to work. The three most important physical processes by which internal energy is exchanged between thermodynamic systems are conduction, convection and radiation. The heat transfer mechanism that involve in this study is convection.

Convection is used to describe the combined effects of conduction and fluid flow. The type of fluid in this study is air. Convection of heat transfer will have a dependence on the viscosity of air in addition to its dependence on the thermal properties of fluid such as thermal conductivity, specific heat and density. This is expected because viscosity influences the velocity profile and correspondingly the energy transfer rate in region near the wall in the case study. Heat transfer by convection, need to consider the velocity of the fluid because the temperature gradient is dependent on the rate at which the fluid carries the heat away; a high velocity produces a large temperature gradient, and so on.

Thus, convection was divided into two conditions; free convection and forced convection. *Free convection* happens when heating exposed to ambient room air without any external sources motion. The result of density gradient is the movement of air near the plate. Then, *forced convection* experienced when motion of the fluid is imposed externally such as by a pump or fan. The example is fan-powered heater, where a fan blows cool air past a heating element, heating the air. We will study in the phenomenon of forced convection deeply in this research by using a fan modeling. The fan will blow the air through a compact heat exchanger unit. Just because air was induced by a fan, the heat transfer mechanism that involved is called forced convection.

2.2 Heat Exchanger

In some systems, the amount of heat generated within the circuit cannot be dissipated through the walls of the components and reservoir. In this system a heat exchanger is required to prevent the system from running above normal operating temperatures. A heat exchanger is a device for transferring heat from one fluid to another, where the fluids are separated by a solid wall so that they never mix. In hydraulic system, there are two types of heat exchanger are used: air-cooled and watercooled. They are widely used in refrigeration, air conditioning, space heating, power production, and chemical processing.

One common example of a heat exchanger is the radiator a car, in which the hot radiator fluid is cooled by the flow of air over the radiator surface. A fan is used to blow air over a bundle of tubes through which the hydraulic fluid flows (figure 2.1). A bundle of tubes is used rather than one large tube because the surface area is much greater. The surface area exposed between the hot fluids (hydraulic oil) and the cooling fluid (air) is the most important factor in the heat exchange.

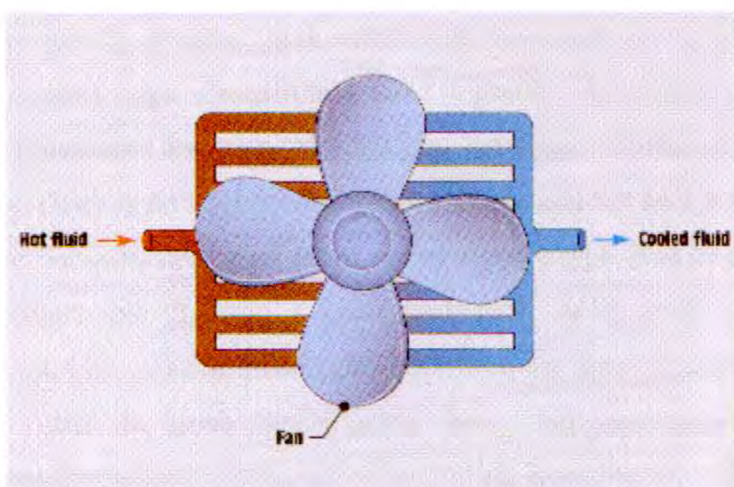

Figure 2.1: Air cooled heat exchanger