INDOOR AIR INVESTIGATION USING CFD APPROACH

THENG KAI XUAN

B041210226

BMCT

Email: thengkaixuan@gmail.com

Final Report

Projek Sarjana Muda

Supervisor: DR. TEE BOON TUAN

Faculty of Mechanical Engineering

Universiti Teknikal Malaysia Melaka

JUNE 2015

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 (Thermal-Fluids)."

Signature : ……

Supervisor : Dr. Tee Boon Tuan

Date : ……

INDOOR AIR INVESTIGATION USING CFD APPROACH

THENG KAI XUAN

This Technical Report is submitted to

Fakulti Kejuruteraan Mekanikal, Universiti Teknikal Malaysia Melaka

In partial fulfilment for

Bachelor of Mechanical Engineering (Thermal-Fluids) with honours

Faculty of Mechanical Engineering

Universiti Teknikal Malaysia Melaka

JUNE 2015

DECLARATION

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

Signature : ……

Author : Theng Kai Xuan

Date : ……

For my beloved Dad and Mum

ACKNOWLEDGEMENT

I would like to express my greatest gratitude to those who have been giving me helps and supports in completing this project. My special appreciation goes to my supervisor, Dr. Tee Boon Tuan for his guidance and advices throughout the project.

I would also like to thank the examiner panels, Dr. Yusmady bin Mohamed Arifin and Dr. Mohd Basri bin Ali for their suggestions on improving my work. Besides that, I would like to thank the management of Mechanical Engineering Faculty (FKM) in providing the necessary equipment and tool to complete this project.

Last but not least, sincere thanks to my family and friends for their continuous supports and encouragement all this time.

Abstract

This project uses experimental and simulation approaches to determine the required parameters, air temperature and airflow velocity in evaluating the indoor air condition. The results gained from simulation are then compared with the experimental values in order to determine the reliability of the Computational Fluid Dynamics (CFD) software. The indoor air condition is also analyzed and compared with the value as recommended by Malaysian Standard (MS) 1525. FLUENT by ANSYS is used for the CFD simulation process in this project. It is recommended that the airflow parameters are kept in certain range to ensure the comfort of the occupants. Good agreement has been achieved between experimental and numerical methods for the analysis of air temperature, with the percentage error of 1.5%. The difference of experimental and numerical values of airflow velocity is considerably large, with percentage error of 53.8%, however, it is within an acceptable range. It is also observed that the values of air temperature and airflow velocity are greater when occupants exist.

Abstrak

Kajian ini menggunakan dua kaedah yang berbeza, iaitu melalui eksperimen dan simulasi untuk menentukan parameter yang diperlukan untuk menilai halaju dan suhu dalam keadaan dalaman. Data yang diperolehi melalui simulasi dibandingkan dengan data eksperimen untuk menentukan kebolehpercayaan perisian Pengkomputeraan Dinamik Bendalir. Keadaan udara dalaman juga dianalisis dan dibandingkan dengan nilai yang direkomenkan oleh Malaysian Standard (MS) 1525. Parameter pengaliran udara perlu dikawal untuk memastikan keselesaan orang dapat dicapai. FLUENT ANSYS dipilih sebagai perisian komputer yang digunakan untuk tujuan simulasi. Beberapa kajian yang dibuat oleh penyelidik lain yang berkaitan dengan pengaliran udara dalam bilik tertutup telah dianalisis. Cara penyelesaian yang digunakan oleh penyelidik tersebut juga dijadikan sebagai rujukan untuk kajian ini. Persetujuan dicapai antara data eksperimen dan data simulasi yang diperolehi untuk suhu dalaman, manakala pembezaan antara keputusan eksperimen dan simulasi untuk halaju dalam kajian ini adalah agak besar, walaubagaimanapun, pembezaan ini masih boleh diterima. Data yang diperolehi dalam kajian ini turut menunjukkan bahawa nilai kedua-dua parameter, iaitu suhu dan halaju adalah lebih besar apabila kewujudan penghuni dipertimbangkan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

SYMBOLS	DESCRIPTION
°C	Degree Celcius
ε	Epsilon
h	Hour
m	Meter
%	Percent
s	Second
W	Watt

LIST OF ABBREVIATIONS

ABBREVIATION	DESCRIPTION
ACH	Air change per hour
ACMV	Air conditioning and mechanical ventilation
a.k.a.	Also known as
ASHRAE	American Society of Heating, Refrigeration, and Air-Conditioning Engineers
BTU	British Thermal Unit
CAD	Computer-Aided Drawing
CFD	Computational Fluid Dynamics
Clo	Clothing unit
etc.	et cetera
HVAC	Heating, ventilation and air conditioning
ISO	Organization for Standardization
MS	Malaysian Standard
PMV	Predicted Mean Vote
PPD	Predicted Percentage Dissatisfied

PPS	Pusat Pengajian Siswazah
Re	Reynolds number
RH	Relative humidity
RNG	Renormalization group
UFAD	Underfloor air distribution
VAV	Variable air volume

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

Recently, the studies of airflow in an enclosed space have been emphasized as the consequences of raising awareness on the effect of airflow distribution to the temperature and comfort level of the space. These effects can be further extended to relate with the health problems of the occupants, based on a study done by Daisey et. al. (2003) which shows that many classrooms have inadequate ventilation. The analysis of indoor air distribution is also done to enhance energy saving and indoor air quality by controlling the contaminants in a space (Cao, 2006).

For indoor condition of a building, the main concern about the airflow distribution goes to the ventilation and comfort level. Countries in tropical zone are considerably warm and humid as compared to other region, therefore, air conditioners and fans are used to achieve a good ventilation and acceptable comfort range. The air conditioners are used to maintain the temperature and humidity of the indoor condition with comfortable range while fans promote even airflow distribution for better ventilation. Human metabolism generates heat and this heat need to be dissipated to the surrounding in order to achieve neutrality while the thermal equilibrium with the surroundings is balanced. Thus, the surrounding temperature need to be maintained lower than the human's body temperature for heat equilibrium

to occur so that the comfort level of the occupants in the building could be maintained. Innova (2002) states that the first comfort condition is thermal neutrality, which means that a person feels neither too warm nor too cold.

In order to simulate the airflow distribution in an indoor building with fast and low cost approach, Computational Fluid Dynamics (CFD) is used to predict the behaviour of fluid flow by solving mathematical equations (Nielsen et. al., 2007), either by numerical methods, algorithms or both depends on the suitability and the choice of users. Some fluid behaviour which can be predicted by using CFD are fluid flow, heat transfer, mass transfer and chemical reactions. To simulate the fluid flow in a room, no equipment or apparatus is needed, instead, a computer-aided design (CAD) drawing and some defined boundary conditions would do the job. Thus, it ease the work of the investigator, and the simulation could be done in short time without requiring high cost equipment. However, the time taken for the simulation depends on the computing speed and uncertainties are hard to avoid.

1.1 Problem statements

This research is conducted to find the answers to the following questions:

i. What is the effect of interior arrangement in a lecture hall to the airflow distribution?

ii. What is the difference in airflow distribution in a lecture hall with/ without the presence of human load?

iii. Is the number of air conditioner in the lecture hall appropriate to provide even cooled air distribution to the occupants? Can the number of air conditioner be decreased at the same time remain the same or similar cooling and air distribution effect by rearranging the position of air conditioner?

1.2 Objectives

i. To evaluate air distribution performance in enclosed rooms through measurements and CFD techniques.

ii. To compare and validate results obtained from both the measurements and CFD simulation techniques.

iii. To optimize the distribution of airflow in the lecture hall by recommendation through simulation.

1.3 Scopes

The scope of this study is focused more on the airflow distribution in lecture hall. The scopes of this study are as follows:

i. To determine the boundary parameters for valid CFD modeling applied in the airflow of lecture hall.

ii. Air distribution parameters in two mechanically ventilated lecture hall (air-conditioner and fan) obtained are airflow velocity and temperature distribution.

iii. To evaluate the current air distribution performance of the lecture hall.

iv. Obtain and analyze the predicted values of important air distribution parameters by using CFD simulation.

v. To investigate the effect of interior arrangement and the presence of human load to the airflow distribution in the lecture hall.