

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

CHAPTER	TITLE	PAGE
	DECLARATION	ii
	DEDICATION	iii
	ACKNOWLEDGEMENT	iv
	ABSTRACT	v
	ABSTRAK	vi
	TABLE OF CONTENTS	vii
	LIST OF TABLES	xi
	LIST OF FIGURES	xiii
	LIST OF SYMBOLS	xvi
	LIST OF ABBREVIATIONS	xvii
	LIST OF APPENDICES	xix
1	INTRODUCION	1
	1.1 Problem statements	2
	1.2 Objectives	3
	1.3 Scopes	3
	1.4 Expected Outcomes	4

CHAPTER	TITLE	PAGE
2	THEORY	5
2.1	Indoor Air Parameters Standards	6
2.2	Air Distribution Type	7
2.3	CFD Simulation	9
3	LITERATURE REVIEW	12
3.1	Thermal Comfort Conditions in a room ventilated with Split System - Numerical and Experimental Analysis by Pereira et. al. (2012)	12
3.1.1	Methodology	13
3.1.2	Results	14
3.1.3	Conclusion	17
3.2	Thermal Comfort Study of an Air-Conditioned Lecture Theatre in the Topics by Cheong et. al. (2001)	18
3.2.1	Methodology	18
3.2.2	Results	20
3.2.3	Conclusion	22
3.3	Thermal Comfort Prediction of an Underfloor Air Distribution System in a Large Indoor Environment by Kim et. al. (2013)	23
3.3.1	Methodology	23
3.3.2	Results	24
3.3.3	Conclusion	27

CHAPTER	TITLE	PAGE
	3.4 Overall Comparison of Studies Done	27
4	METHODOLOGY	29
	4.1 Experimental Method	30
	4.2 Numerical Method	33
	4.2.1 Geometry	34
	4.2.2 Mesh	35
	4.2.3 Problem setup	36
	4.2.4 Solution	38
5	RESULTS AND ANALYSIS	40
	5.1 Air Investigation of Lecture Hall Without Human Load	40
	5.1.1 Temperature	40
	5.1.2 Airflow velocity	42
	5.2 Airflow Investigation of Lecture Hall With Human Load	45
	5.2.1 Temperature	45
	5.2.2 Airflow velocity	47
	5.3 Air Velocity and Temperature Analysis	50

CHAPTER	TITLE	PAGE
6	CONCLUSION AND RECOMMENDATIONS	51
6.1	Conclusion	51
6.2	Problems of Studies	52
6.3	Recommendations	53
6.3.1	Improvements of project	53
6.3.2	Suggestions for future works	56
	REFERENCES	57
	APPENDICES	62

LIST OF TABLES

TABLE	TITLE	PAGE
3.1	Boundary conditions of inlet diffusers and extract grilles (Source: Cheong et. al., 2001)	20
3.2	Corroboration between the measure and predicted results for air temperature and velocity (Source: Cheong et. al., 2001)	20
3.3	Analysis conditions for simulation (Source: Kim et. al., 2013)	24
3.4	Air velocity in occupied and unoccupied zone for cases (m/s) (Source: Kim et. al., 2013)	25
3.5	Mean temperature in the occupied and unoccupied zones (Source: Kim et. al., 2013)	27
3.6	Comparison of previous studies	28
4.1	Details of meshing sizing	36
4.2	Statistics of meshing	36
4.3	Name selection for each boundary condition	36
4.4	Setup for solver of general problem	37
4.5	Selection of model	37

4.6	Setting for boundary conditions	38
4.7	Solution methods setting	39
5.1	Comparison of measured and simulated temperature in the lecture hall	41
5.2	Comparison of measured and simulated airflow velocity in the lecture hall	43
5.3	Temperature difference of simulation with and without occupants	46
5.4	Airflow velocity difference of simulation with and without occupants	48
6.1	Comparison of simulated temperature value of original and repositioned air conditioners	55
6.2	Comparison of simulated airflow velocity value of original and repositioned air conditioners	55

LIST OF FIGURES

FIGURE	TITLE	PAGE
2.1	Mixing air distribution (Source: Engineering Bulletin, 2010)	7
2.2	Displacement ventilation (Source: Engineering Bulletin, 2010)	8
2.3	Unidirectional air pattern (Source: Engineering Bulletin, 2010)	8
2.4	Layout of ANSYS	9
2.5	CFD simulation process	10
3.1	Location of measuring points (Source: Pereira et. al., 2012)	13
3.2	Air temperature profiles - Experimental measurements (Source: Pereira et. al., 2012)	14
3.3	Air velocity profiles - Experimental measurements (Source: Pereira et. al., 2012)	15
3.4	Pathlines inside the classroom (Source: Pereira et. al., 2012)	15

3.5	Supply and return temperature cycle (Source: Pereira et. al., 2012)	16
3.6	Experimental results for the PMV and PPD (1.1 m height) (Source: Pereira et. al., 2012)	16
3.7	Numerical results for the PMV and PPD (1.1 m height) (Source: Pereira et. al., 2012)	17
3.8	Location of measuring points in the lecture theater (Source: Cheong et. al., 2001)	19
3.9	Air temperature profile (Source: Cheong et. al., 2001)	21
3.10	Carbon dioxide concentration profile (Source: Cheong et. al., 2001)	21
3.11	Distribution of subjective response on temperature, humidity, air movement and overall thermal comfort (Source: Cheong et. al., 2001)	22
3.12	Boundary condition and the locations of the diffuser (Source: Kim et. al., 2013)	24
3.13	Airflow distribution in the model space (Source: Kim et. al., 2013)	25
3.14	Temperature distribution in the model space (Source: Kim et. al., 2013)	26
4.1	Research methodology	30
4.2	Layout plan for measuring points	31
4.3	Anemometer	31
4.4	Lecture hall for airflow analysis	32
4.5	Data collection by using anemometer	33
4.6	Airflow simulation process	34

4.7	Isometric view of the lecture hall	35
4.8	Side view of the lecture hall. Without occupants (up) and with occupants (down)	35
5.1	Graph of measured and simulated value of temperature	41
5.2	Temperature Contour of the model space	42
5.3	Graph of measured and simulated value of airflow velocity	44
5.4	Velocity magnitude pathlines	45
5.5	Air Temperature Comparison for simulation with and without occupants	46
5.6	Comparison of temperature distribution for simulation without occupants (up) and with occupants (down)	47
5.7	Airflow velocity Comparison for simulation with and without occupants	48
5.8	Comparison of airflow velocity distribution for simulation without occupants (up) and with occupants (down)	49
6.1	Temperature pathlines of original position (left) and repositioned (right) split unit air conditioners	54
6.2	Airflow velocity pathlines of original position (left) and repositioned (right) split unit air conditioners	55

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

APPENDIX	TITLE	PAGE
A	Flow chart of Projek Sarjana Muda	62
B	Project timeline Gantt Chart for Projek Sarjana Muda	63

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.