SIMULATION AND MEASUREMENT FOR INDOOR AIR FLOW

LIM DA PING

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 (Design and Innovation)"

Signature: ……..

Supervisor: Dr. Tee Boon Tuan

Date: ……..

SIMULATION AND MEASUREMENT FOR INDOOR AIR FLOW

LIM DA PING

This Technical Report is submitted to
Fakulti Kejuruteraan Mekanikal, Universiti Teknikal Malaysia Melaka
In partial fulfilment for
Bachelor of Mechanical Engineering (Design and Innovation) with honours

Faculty of Mechanical Engineering
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

JUNE 2013

DECLARATION

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

Signature: ……

Author: ……

Date: ……

For my beloved Mum and Dad

ACKNOWLEDGEMENT

Firstly, I would like to express my gratitude to my supervisor Dr. Tee Boon Tuan, who guides me throughout this project. Under his guidance, I had gained lots of knowledge and able to complete this final year project 1 report successfully.

Besides that, I would also like to send my gratitude to En. Muhammad Zulfattah Bin Zakaria for evaluating my final year project. Several inputs which were suggested earlier had given me an idea and guidance to improve my final year project.

Furthermore, I would like to thank Faculty of Mechanical Engineering (FKM), Universiti Teknikal Malaysia Melaka (UTeM) for giving a fully support and allows me to utilize the facilities equipment's which are available in laboratories.

Last but not least, I would like to send my appreciation to my family for their continual encouragement, recognition and support over long months during the preparation of this project.

ABSTRACT

Air flow inside a room is relatively important as most of the people spend their daily time indoors. The main purpose of this research is to investigate the air flow motion inside a meeting room by using experimental and computer software simulation method. The focus of this research is on a meeting room which is ventilated by the mechanical ventilation system. The indoor air parameters for this research include air velocity, temperature and also relative humidity. A measuring device which is air velocity meter is chosen to measure the indoor air parameters. However, for the meeting room air flow motion, simulation is analyzed by using Computational Fluid Dynamics (CFD) software named ANSYS FLUENT software. The results from the experiment and simulation are compared with respect with accuracy, sensitivity and applicability.

ABSTRAK

Pengaliran udara dalam sesebuah bilik dianggap penting kerana kebanyakan manusia menghabiskan masa harian mereka dalam bilik. Matlamat utama kajian ini adalah untuk mengkaji pengaliran udara dalam sebuah bilik mesyuarat dengan menggunakan cara kajian dan kaedah perisian simulasi. Fokus untuk kajian ini adalah sebuah bilik mesyuarat yang pengudaraannya dialirkan oleh sistem pengudaraan mekanikal. Parameter pengudaraan dalaman untuk kajian ini termasuk kelajuan angin, suhu udara dan kelembapan relatif. Alat pengukuran meter halaju udara telah dipilih untuk mengukur parameter udara dalam bilik. Tetapi, penggerakan aliran udara, simulasi dilakukan oleh perisian CFD iaitu ANSYS FLUENT. Keputusan dari kaedah kajian dan simulasi akan dibandingkan berdasarkan ketepatan, sensitiviti dan kebolehgunaan.

TABLE OF CONTENTS

CHAPTER	TITLE	PAGE
	SUPERVISOR DECLARATION	i
	DECLARATION	iii
	DEDICATION	iv
	ACKNOWLEDGEMENT	v
	ABSTRACT	vi
	ABSTRAK	vii
	TABLE OF CONTENTS	viii
	LIST OF TABLES	xi
	LIST OF FIGURES	xii
	LIST OF APPENDICES	xiv
CHAPTER 1	**INTRODUCTION**	1
	1.1 OVERVIEW	1
	1.2 OBJECTIVES	3
	1.3 SCOPE	3
	1.4 PROBLEM STATEMENTS	3
	1.5 EXPECTED OUTCOMES	4
CHAPTER 2	**LITERATURE REVIEW**	5
	2.1 MECHANICAL VENTILATION IN ROOMS	5
	2.2 GENERAL FLOW BOUNDARY CONDITION	7
	2.3 COMPUTATIONAL FLUID DYNAMICS	9
	2.4 TURBULENCE MODELING	9
	2.4.1 STANDARD κ-ε MODEL	10
	2.4.2 RNG κ-ε MODEL	11

2.4.3 REYNOLDS STRESS TURBULENCE MODEL 12
2.4.4 LARGE EDDY SIMULATION TURBULENCE MODEL 13
2.5 PREVIOUS STUDIES FROM OTHER RESEARCHERS 14
2.5.1 SIMULATION AIR FLOW IN LECTURER ROOM BY JOHNSON, ADNAN AND TEE 14
2.5.2 CFD VALIDATION OF THE THERMAL COMFORT USING DRAFT RATES BY MIKA AND JOHN 17
2.5.3 NUMERICAL STUDIES ON THE INDOOR AIR FLOW IN THE OCCUPIED ZONE OF VENTILATED AND AIR-CONDITIONED SPACE BY CHOW W.K AND FUNG W.Y 18

CHAPTER 3 METHODOLOGY 21
3.1 RESEARCH METHODOLOGY 21
3.2 ROOM BOUNDARY CONDITION 23
3.3 INSTRUMENTATION 23
3.3.1 ROOM PHYSICAL DATA MEASUREMENT 24
3.3.2 INDOOR AIR PARAMETERS MEASUREMENT 24
3.3.3 ANSYS FLUENT SOFTWARE 25
3.4 PROCEDURES 25
3.4.1 MEASUREMENT PROCEDURES 25
3.4.2 SIMULATION PROCEDURES 26
3.4.3 DATA ANALYSIS 27
3.5 EXPERIMENTAL CONDITION ASSUMPTION 28

CHAPTER 4 RESULTS 29
4.1 RESULTS ANALYSIS 29

	4.2 MEASUREMENT RESULTS	29
	4.3 SIMULATION RESULTS	33
	4.4 STREAM PLOT FOR SIMULATION	34
	4.5 COMPARISON RESULTS ANALYSIS	39
CHAPTER 5	**DISCUSSION**	41
	5.1 DATA COMPARISION FOR MEASUREMENT AND SIMULATION	41
	5.2 COMPARISION WITH PREVIOUS STUDY	42
CHAPTER 6	**CONCLUSION**	44
	6.1 CONCLUSION AND RECOMMENDATION	44
	REFERENCES	46
	APPENDIX A	48
	APPENDIX B	49

LIST OF TABLES

NO	TITLE	PAGE
Table 2.1	Percentage of Results with AbsoluteDifference of Air Velocity Value of Below 0.10 m/s	16
Table 2.2	Percentage of Results with Absolute Difference of Air Temperature Value of Below 1^0C (Johnson Lim, Adnan, & Tee, 2002)	16
Table 2.3	Average Velocity Values for Experiments and Simulations Cases (Tinker, 2007)	18
Table 3.1	Description for Meeting Room Simulation in ANSYS FLUENT Software	26
Table 4.1	Physical data of Meeting Room Dimension	29
Table 4.2	Measurement Data for the Meeting Room	30
Table 4.3	Average and Range of Measurement Parameter in Meeting Room	31
Table 4.4	Simulation Data for the Meeting Room	33
Table 4.5	Comparison for Air Velocity for Both Measurement and Simulation	39
Table 4.6	Comparison for Air Temperature for Both Measurement and Simulation	40
Table 5.1	Comparison of Air Parameter with Previous Researcher	42

LIST OF FIGURES

NO	TITLE	PAGE
Figure 2.1	Schematized Representation of Mixing and Displacement Ventilation Principle; Flow Pattern (left) and Variable Gradient of the Height (right). At a given Height the Variable has been Averaged over the Total Horizontal Plane in the Room	6
Figure 2.2	Specification of the General Flow Boundary Conditions Using the Results of a Multi-zone Infiltration Model .	8
Figure 2.3	Stream Plots for RNG k-e Turbulence Model Simulation in Lecture Room BK17	15
Figure 2.4	Stream Plots for RNG k-e Turbulence Model Simulation in Lecture Room BK23	15
Figure 2.5	Test Room Plan and Dimensions from Z and X-directions	17
Figure 2.6	Location of the field measuring points at the occupied zone of test sites (Office Room, Gymnasium, Railway Concourse)	19
Figure 2.7	Validation of the numerical results (all at the occupied zones of the test sites)	20
Figure 3.1	Procedure Flow Chart	22
Figure 3.2	Overview of the Meeting Room	23
Figure 3.3	Air Velocity Meter Measuring Device	24
Figure 4.1	Graph of Air Velocity Versus Zone	31
Figure 4.2	Graph of Air Temperature Versus Zone	32
Figure 4.3	Graph of Relative Humidity Versus Zone	32
Figure 4.4	The Overall Flow Pattern Inside the Meeting Room	34

Figure 4.5	The Air Velocity Flow Pattern Came Out From the Inlet	35
Figure 4.6	The Air Velocity Flow Pattern Came Out From the Inlet	36
Figure 4.7	Closer Look For the Inlet of the Air Diffuser	36
Figure 4.8	Air Flow Pattern on Cross-section of Z-plane	37
Figure 4.9	Temperature Distribution Inside the Meeting Room	38

LIST OF APPENDICES

NO	TITLE	PAGE
A	Zone Division and the Position of the Air Diffuser in the Meeting Room	47
B	Project Timeline Gantt Chart for PSM I	48

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

Nowadays, as majority people spend most of their time indoors, therefore the indoor air flows inside a building is very important. As we all know, people seems to be more productive and effective when they feel comfortable in their working place. A room which is well ventilated and temperature controlled will provide a better occupant environment compare to the one which is too cold, too hot or even poorly ventilated. Therefore, with the introducing of mechanical ventilation, it has become the primary mechanism which used to maintain the acceptable indoor air quality and provides a suitable thermal comfort inside a building.

The concern about the indoor air flow has increased since the past few years due to the knowledge about the important of thermal comfort and air quality. Basically, there are four main factors that affect the thermal comfort and air quality of an indoor, which are air temperature, air velocity, relative humidity and also radiation. Hence, ventilation plays an important role in providing productive and comfortable indoor environment and also securing in contaminant removal. Over the recent years, both experimental measurements and computational simulations have been used to predict the indoor air flow precisely. Most experiments use a full-scale test chamber to setup an artificial environment to isolate the space from the external. This method would be costly and the time consume is normally long. Through the highly improved and modernization of computers and numerical method, numerical

approach by using a Computational Fluid Dynamics (CFD) is now been widely used to model the indoor air flows.

Computational Fluid Dynamics (CFD) is a type of fluid mechanics which uses mechanical method and algorithm to solve and analyzed the problems that related to fluid flows. It is used to replace the governing partial differential equations or integral equations of fluid flows with numbers, and use those numbers in either space or time or both to obtain a finalize numerical description of the completed fluid flow of interest. CFD has recently become the premier tool for fluid flow simulation due to the highly improve and growth in computer technologies. Its application not only specifically for modeling buildings but also include those aircraft aerodynamics, hydrodynamics of ships, meteorology, engineering in biomedical, pollutant effluents studies and etc.

In this research, the studies are stress on modeling the indoor air flow of a meeting room. The measurement will focus on the three parameters which are air temperature, air velocity and relative humidity. Moreover, the air flow pattern will be monitor and analyze by the help of CFD air flow simulation.

1.2 OBJECTIVES

a) To conduct the measurement of indoor air flow patterns, with respect to accuracy, sensitivity and applicability.
b) To reveal current limitation in computational simulation for indoor air flow and also its sensitivity to input parameters. (Attention is put on the computational results rather than to improve the modeling)
c) To investigate a type of flow problem that subscribes to the current view in ventilation, for example task conditioning.

.

1.3 SCOPE

a) To conduct practical measurement involving air velocity, temperature and relative humidity by using available measuring equipment.
b) To perform a suitable simulation technique by using the available CFD software and comparing its validity and sensitivity with the measurement data.

1.4 PROBLEM STATEMENTS

Most of the indoor air flows inside a room are actually complicated as cause by many factors such as pressure gradient, thermal buoyancy force and etc. This research is conducted to find out that if the measurement results for actual room condition is compatible with the results from CFD simulation. Hence, resolving the relationships between the geometric room parameters with the air flow patterns they produced is important.

1.5 EXPECTED OUTCOMES

a) Measurement of physical data by using available measuring device.
b) Set of measurement data of indoor air parameters by using suitable measuring device.
c) Results of analysis of the indoor air flow by the CFD simulation.
d) Comparison of the data and results between the measurement and simulation of indoor air flows.

CHAPTER 2

LITERATURE REVIEW

2.1 MECHANICAL VENTILATION IN ROOMS

Basically, there are two types of mechanical ventilation used to ventilate the space of a room, which are mixing ventilation and displacement ventilation. Mixing ventilation is a traditional method of supplying air to ventilate the spaces in a room. In mixed flow ventilation, the cool air is supplied from the ceiling or through a higher place where the incoming air is supplied at a relatively high velocity to cause a high degree of mixing with the air in the room to take place.

However, displacement ventilation is a method where the low air velocity is supplied from the bottom of a room (floor level). The air which is from the lower part of the room is supplied upward by rising convection flows from heat sources in the room and will be removed at the upper level (ceiling level). The characteristics of this type of ventilation includes low location of supply opening and high location of return opening, free convection around heat sources, stratified flow in the room, vertical temperature and concentration gradient and also air movement is controlled by buoyancy.

Figure 2.1 shows the characteristic differences between two types of ventilation. The left side of the figure shows the flow pattern while the right side is the averaged velocity magnitude, temperature and contaminant concentration as function of the height.

It is actually possible to ventilate a room naturally. This can be done by refreshing indoor air with outside air through openings in the windows or door. For meeting room, natural ventilation unable to maintain the high level of indoor air quality for a longer period. Therefore, mechanical ventilation is mostly used (Loomans, 1998).

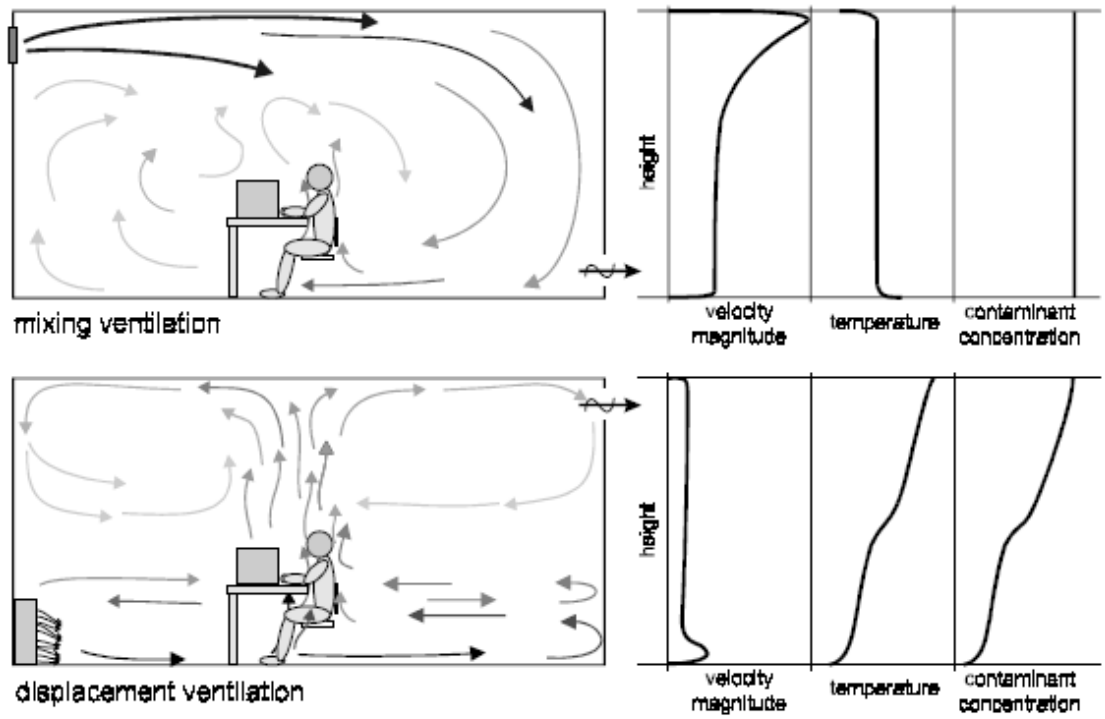

Figure 2.1: Schematized Representation of Mixing and Displacement Ventilation Principle; Flow Pattern (left) and Variable Gradient of the Height (right). At a given Height the Variable has been Averaged over the Total Horizontal Plane in the Room (Loomans, 1998).

2.2 GENERAL FLOW BOUNDARY CONDITION

In the term of Computational Fluid Dynamics (CFD), the boundary conditions can be specified in three ways:

1) Dirichlet condition, e.g. $\emptyset = f$ on ∂C, where $\emptyset$ is the independent variable, C is the computational domain, and ∂C is the boundary.
2) Neumann (derivative) condition, e.g. $\partial\emptyset/\partial n = f$ or $\partial\emptyset/\partial s = g$on ∂C, where n is the outward normal distance and s is the distance along the boundary.
3) Robin (mixed) condition, e.g. $\partial\emptyset/\partial n + k\emptyset = f$ on ∂C. The computational algorithm (based on the governing equations) extrapolates these boundary conditions to provide the solution in interior, C.

The flow boundary conditions denote the boundary conditions for the momentum equation. The conventional specification of the flow boundary conditions is to specify a constant velocity at supply inlets (Dirichlet condition), a non–slip condition at wall surfaces (Dirichlet condition), and zero gradient condition at exhaust outlets (Neumann condition). However, in reality, there is always a leakage or opening inside a building which allow the air to escape to outside. This type of air flow is uncontrolled and may affect the steady supply of the mechanical ventilation as well as flow pattern change in the room depends on the level of the uncontrolled flow. The general flow boundary conditions can be consider as more realistic boundary treatments, where the steady supply of the mechanical ventilation and interaction between mechanical and nature ventilation are considered. The boundary value at various openings in either single or multi zone infiltration model can be defined.

A number of single-cell and multi zone models have been developed since twenty years ago. These models generally consider the mechanical ventilation, tightness of the building, terrain and shielding conditions and also climate conditions. The predicted air flow rates across the envelopes, between the rooms, and through the mechanical ventilation system can be used as the boundary conditions in CFD simulation as shown in Figure 2.2.The specification of the general flow boundary conditions is complicated and difficult when specifying the area and position of the leakage or opening as found in formulating multi zone models (Yuguo Li, 1994).

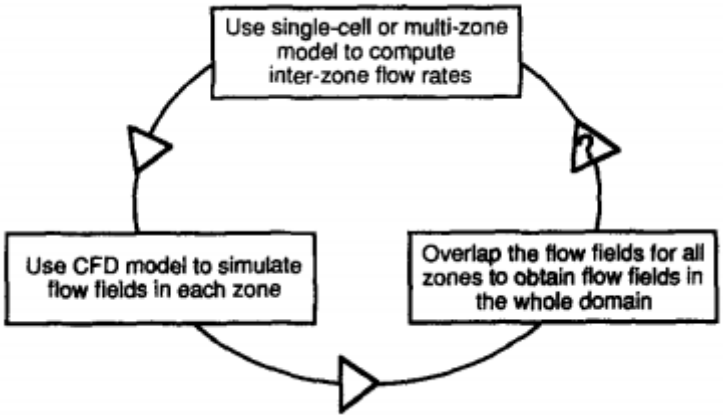

Figure 2.2: Specification of the General Flow Boundary Conditions Using the Results of a Multi-zone Infiltration Model (Yuguo Li, 1994).

2.3 COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is used to predict and model the internal and external air flow inside a building. Many commercial CFD software which are available in market can be used to predict the air flow pattern ventilate by air distribution devices and thermal sources. From the perspective of CFD, air flows in a room can be considered complex. This is because in the reality, at a reasonable ventilation rates, the air flow is completely turbulent in the supply of air ducts, air conditioning system and downstream edges of the obstacles. Therefore, indoor air flow simulation use turbulence models to compute the mean values with the help of computers (Cao, 2006).

2.4 TURBULENCE MODELING

Turbulence model can be defined as a computational procedure which is used to close mean flow equations system. For the applications in most engineering, resolving the details of the turbulent fluctuations is not necessary. This is because the turbulence models allow the calculation of mean flow first before calculating the full time-dependent flow field. There are few types of turbulence models which are commonly used such as Reynolds-averaged Navier-Stokes, large eddy simulation, RNG κ-ε, etc. (Bakker, 2006)