CFD MODELLING OF BUILDING VENTILATION

LIM HENG YANG

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

CFD MODELLING OF BUILDING VENTILATION

LIM HENG YANG

**This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering**

Faculty of Mechanical Engineering

UNIVERSITY TEKNIKAL MALAYSIA MELAKA

AUG 2020

DECLARATION

I declare that this project report entitled "Computational Fluid Dynamics Modelling of Building Ventilation" is the result of my own except as cited in references

Signature	:	…………………………….
Name	:	…………………………….
Date	:	…………………………….

APPROVAL

I hereby declare that I have read this project report 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.

Signature : ....

Name of Supervisor : ....

Date : ....

DEDICATION

To my beloved mother and father

ABSTRACT

The study of how thickness of wall affects the flow rate throughout the building when using the Computational Fluid Dynamics (CFD). By comparing the result to the experimental result would make sure the CFD simulation is in correct matter. Three thickness of the wall has been chosen from the actual size of the wall and simulate under CFD by using different turbulence models. The result for the full flow field is recorded and compared with another simulation method which is the domain decomposition method. The domain decomposition method is simulating without the computational domain. This method greatly reduced the time taken for the simulation which gives the result of the internal flow at the inner building only. From the result taken, the thickness of the wall does not affect the flow rate throughout the building and the velocity ratio throughout the building also constant from the inlet to the outlet of the building. Both methods able to get the velocity ratio which is near to the experimental and both methods give the same volume flow rate which the difference is approximately 4% only.

ABSTRAK

Kajian ini mengenai bagaimana ketebalan dinding mempengaruhi kadar aliran di kawasan bangunan apabila menggunakan Computational Fluid Dynamics (CFD). Dengan membandingkan hasilnya dengan hasil eksperimen akan memastikan hasil simulasi CFD adalah betul. Ketebalan tiga dinding telah dipilih dari ukuran sebenar dinding dan simulasikan di dalam CFD dengan model turbulensi yang berbeza. Hasil daripada full flow field akan direkodkan dan membandingkan dengan kaedah simulasi yang lain iaitu Domain decomposition. Kaedah ini ialah menjalankan simulasi tanpa computational domain dan ini akan mengurangkan masa yang diperlukan dan akan mendapat hasil aliran bangunan dalaman sahaja. Dari hasil yang diambil, ketebalan dinding tidak akan mempengaruhi aliran di bangunan dalaman dan nisbah halaju di bangunan juga tetap dari tempat masuk sampai keluar bangunan. Kedua-dua kaedah dapat memperoleh nisbah halaju yang hampir dengan eksperimen dan kedua-dua kaedah tersebut memberikan kadar aliran isipadu yang sama dan perbezaannya hanya 4% sahaja.

ACKNOWLEDGEMENTS

The finished of this thesis is thanks to all the guidance and assistance given to me by many individuals and groups. I extremely appreciate the help from them all along to finish this thesis writing.

First and foremost, I would like to express my sincere and acknowledgement from my supervisor, Dr. Cheng See Yuan from University Technical Malaysia Melaka (UTeM) which is from Faculty of Mechanical Engineering which he helped and supported me during this period and appreciate his guidance and teaching to complete this thesis. Without his guidance and knowledge that were given to me, I would be facing a lot of problems during the process of finishing the thesis.

Furthermore, I would like to thank my friends, Fong Yee Pin and Yeow Yee Sin for teaching and motivates me to complete the thesis and we shared a lot of knowledge such as learning Ansys software together starting from zero.

Lastly, I would like to thank my parents for their moral support and support me throughout my life. I would also like to thank all people who supported and guided me throughout this process.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
IAQ	Indoor Air Quality
LES	Large Eddy Simulation
PIV	Particle Image Velocimetry
PSM	Projek Sarjana Muda
RANS	Reynolds- averaged Navier- Strokes Models
RNG	Renormalization Group
RSM	Reynolds Stress Model
SBS	Sick Building Syndrome
SST	Shear Stress transport

LIST OF SYMBOLS

Q	=	Volume Flow Rate
C_d	=	Discharge Coefficient
A	=	Cross Sectional Area
ρ	=	Density of Air
Δp	=	Pressure Difference
V_r	=	Reference Wind Speed
C_p	=	Pressure Coefficient
ΔT	=	Temperature Difference
$\bar{T}$	=	Mean Temperature
G	=	Gravity Acceleration
H	=	Total Height of the Opening
V	=	Air Velocity Leaving the Opening
P_w	=	Pressure Due to Wind
U_z	=	Mean Wind Velocity at A Specific Height
Re	=	Reynolds Number
U	=	Fluid Velocity
L	=	Length of the Fluid
μ	=	Dynamic Viscosity of the Fluid
Z_0	=	Aerodynamics Roughness Length
U^*_{ABL}	=	Atmospheric Boundary Layer Friction Velocity
κ	=	Von Karman Constant
Z	=	Height Coordinate
K	=	Turbulent Kinetic Energy

I_u	=	Streamwise Turbulence Intensity
A	=	Parameter Range Between 0.5 To 1.5
ε	=	Turbulence Dissipation Rate
ω	=	Specific Dissipation Rate
C_μ	=	Empirical Constant (0.09)
ν	=	Kinematic Viscosity of the Fluid

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Building ventilation is about to control the airflow and the quality inside the building. Based on (Li and Nielsen, 2011), maintaining good airflow and quality at this era becomes more and more challenging for the engineer due to that the building is now taller, larger, and deeper and this will affect the air distribution. To solve these cases, doing computational fluid dynamics (CFD) before construction to test the ventilation is one of the ways. Building ventilation is important due to that human spend around 80% of the time in indoor especially their homes and working environment (office). Therefore, a good Indoor Air Quality (IAQ) is important to avoid the disastrous consequences on human health such as Sick Building Syndrome (SBS) (Norhidayah *et al.*, 2013).

There are two ways to maintain ventilation in the building which is natural ventilation and mechanical ventilation. Mechanical ventilation is mean by using ducts or fans to circulate airflow, but it does not provide real ventilation due to there is no introduction of fresh air. Mechanical ventilation gives a lot of advantages, but the consumption rate of energy is high such as the air-conditioning used up to half of the annual energy consumption with the energy ratio up to 100kWh/m^2. (Bastide *et al.*, 2006).

Nowadays, most of the buildings will rely on mechanical ventilation but to reduce their annual heating period, natural ventilation is more advantages to reduce the energy consumption for the ventilation. Natural ventilation should be promoted due to it provides better health and environmental concerns and used about 15% lesser cost compare to air-conditioned equivalent. (Cheung and Liu, 2011). Natural ventilation is mean by airflow at the surrounding of the indoor and outdoor by wind or local density differences and it does not consist of any mechanical part. This ventilation mostly used in a public area such as schools, offices, etc. (Jiru and Bitsuamlak, 2010).

To get the flow pattern of the building in a controlled environment, Computational Fluid Dynamics (CFD) is the way to solve it because CFD can take full control of the boundary condition. From the observation of the flow pattern, good ventilation able to obtain from the design of the building based on the flow pattern. Due to this, CFD now is widely used to solve the ventilation problem that can be easily simulated at PC (Li and Nielsen, 2011). Different buildings have different kinds of windows such as the opening is at different positions and some of them contain different thicknesses also. So, to get more information about the thickness of the window, CFD simulation is needed (Wang E., 2017). There are two ways to determine the flow inside the building which is the full flow field method and domain decomposition method. The full flow field considered the building inside a computational domain which able to visualize the flow inside and outside of the building with the boundary condition is at the computation domain inlet. The airflow is started at the computational domain and will flow towards the building which is at a distance from the computation domain. The domain decomposition is analyzed the outside and inside of the building separately. The simulation is done on the outside of the building first, in which the flow condition is set at the computational domain inlet. From the result, the flow condition near the building is extracted and placed on the inlet of the building model without the computational domain. This enables to reduce the number of the element during the simulation (Ramponi and Blocken, 2012).

1.2 PROBLEM STATEMENT

The airflow for the full flow field and domain decomposition method might be different and it is interesting to see the velocity profile between these two methods. Obtain the effect of the thickness of the wall to the airflow of the building and check whether the thickness will give different results by Computational Fluid Dynamics (CFD). There are many different turbulence models from the Ansys software, to get the best parameter setting through comparison from a reference (Journal or Book).

1.3 OBJECTIVE

The main objectives of this research project are shown below:

a) To identify the velocity profile for the full flow field and domain decomposition method

b) To predict the airflow inside the building by changing the thickness of the wall of the building model

c) To perform Computational Fluid Dynamics (CFD) simulation on the full flow field and domain decomposition method

1.4 SCOPE OF PROJECT

The scope of the present studies is shown below:

a) Karava's Building model with the scale of 0.1m x 0.1m x 0.08m with ratio down to 1:200 is used with 3 thicknesses of the wall.

b) The computational domain of 5L of the building model is being used while the downstream is at 10L of the building model.

c) Simulation of the full flow field and domain decomposition method and compared with the experimental result.

d) The airflow rate inside the building for the full flow field and domain decomposition method will be obtained with different thicknesses of the wall of the building model.

1.5 GENERAL METHODOLOGY

The actions that need to be carried out to achieve the objectives in this project are listed below.

1. Literature review

   Journals, articles, or related information regarding the project will be reviewed.

2. Simulation

   Simulation of the Computational Fluid Dynamic (CFD) to get see flow patterns in the full flow field and domain decomposition method.

3. Analysis and proposed solution

   The analysis will be on how the existence of the window of the building affects the pressure differences at the surface of the building

4. Report writing

   A report on this study will be written at the end of the project.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter is talking about the journal or any related to the CFD of building. The knowledge or theory from the journal that previously done might give some help to this current research.

2.2 NATURAL VENTILATION

Natural Ventilation is to make sure the fresh air able to supply into the building for heat dissipation and more comfortable to live in. There is another method which is mechanical ventilation which involves mechanical appliances to cool down the building. There are two types of natural ventilation which are the control and uncontrolled (Infiltration). Controlled Natural Ventilation means by action which purposely lets the air going through such as window or door. This usually is controlled by us depending on the situation. Infiltration is the flow that unable to control it such as the airflow movement that went through the cracks, gaps in between the building structure. This can be stopped if the cracks or gaps are plugged (Bhatia, 2014). Figure 2.1 shows that ventilation is a combination of infiltration, ventilation by natural or mechanical means. Natural ventilation is one of the required lowest energy building design. Although the invention of the air – conditioning system provide a better indoor environment but the negative impact that the air – conditioning, and refrigerants bring a serious problem to the environment.

Therefore, natural ventilation plays an important role in which it needs a good design to get good ventilation. Due to the problem of scaling and hard to represent natural ventilation in the laboratory, this causes Computational Fluid Dynamics (CFD) to become more popular as it can use for design even in both mechanical and natural ventilation systems (Awbi, 1996).

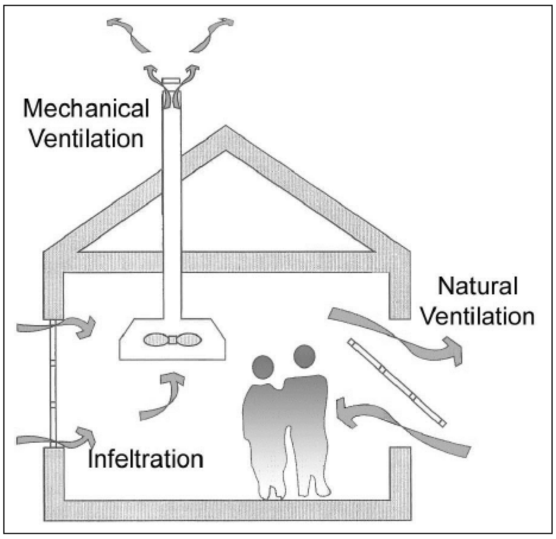

Figure 2.1: Ventilation that consist of infiltration, natural and mechanical means. (Asfour, 2015)

There are two sources of natural ventilation which is: -

i) Wind – The air will flow based on the pressure which is from high pressure to low pressure. This effect is based on the principle of Bernoulli which uses differences in pressure to move the air. Natural wind pressure usually varies from 0.004 to 0.14 inches of water column (Bhatia, 2014).

The volume flow rate (Q) can be calculated using the formula below:

$$Q = C_d A \sqrt{\frac{2\Delta p}{\rho}} \qquad \text{Equation 2.1}$$

Where C_d is the discharge coefficient, A is the area, ρ is the density and Δp is the pressure difference which can by calculate:

$$\Delta p = 0.5 \rho C_p V_r \qquad \text{Equation 2.2}$$

Where V_r is the reference wind speed and C_p is the pressure coefficient at the opening.

For number of opening in parallel:

$$C_d A = \sum (C_d A)_i \qquad \text{Equation 2.3}$$

And for number of opening in series:

$$\frac{1}{(C_d A)^2} = \sum \frac{1}{{(C_d A)_i}^2} \qquad \text{Equation 2.4}$$

ii) Buoyancy – The hot air will flow upward due to the hot air is contain less density compared to cold air. This will create a pressure difference that in turn induces air movement. This phenomenon is called “Thermal Buoyancy” and sometimes called “The Stack Effect”. The buoyancy – generated pressure ranges from -0.001 to 0.01 inches of water column (in-wc) which is quite low (Bhatia, 2014).

The volume flow rate (Q) through a large opening due to temperature difference is given:

$$Q = \frac{C_d}{3} A \sqrt{\frac{gH\Delta T}{\bar{T}}}$$

Equation 2.5

Where ΔT is the temperature difference across the opening at the buildings, $\bar{T}$ is the mean temperature (K), C_d is the discharge coefficient, A is the cross sectional area of the inlet (inlet = outlet), g is the gravity acceleration (m/s^2), H is the total height of the opening. The calculated flow rate able to compare with the American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE) which is a standard for different design and maintenance of indoor environments. It helps to indicate the minimum ventilation rate in Breathing Zone at different indoor.

2.3 AIRFLOW

Airflow is the movement of air from one area to another. Air able to move through the building by the differential pressure of indoors and outdoors, this can be created through natural force (wind-induced pressure difference). Figures 2.2 shows the airflow through the building with 0^o of wind. For example, the pressure difference induced by temperature gradients inside and outside of the building (Briney, 2018). To have airflow, there must be a pressure difference and flow in a continuous condition or opening connecting the points (Straube, 2008). As the wind hits a building, the high pressures area will generate on the windward wall while the lower pressures are on the leeward side of the façade of the building (Bhatia, 2014). Figure 2.3 shows the airflow with high pressures area and low pressures area.