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).

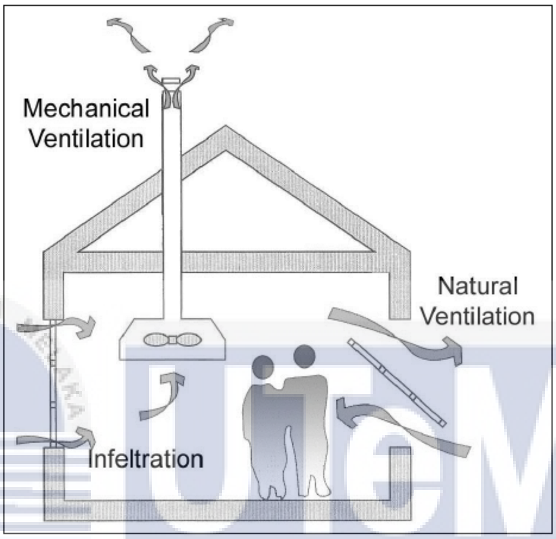

Figure 2.1: Ventilation that consist of infiltration, natural and mechanical means. (Asfour, 2015)

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° 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.

The rate of wind driven airflow through the opening in the building can be calculated by using:

$$Q = C_d A V \qquad \text{Equation 2.6}$$

Where,

Q= Volumetric flow rate through the opening (m^3/s)

A= Free area of inlet opening (m^2)

V= Air velocity leaving the opening, (m/s)

C_d = Discharge coefficient

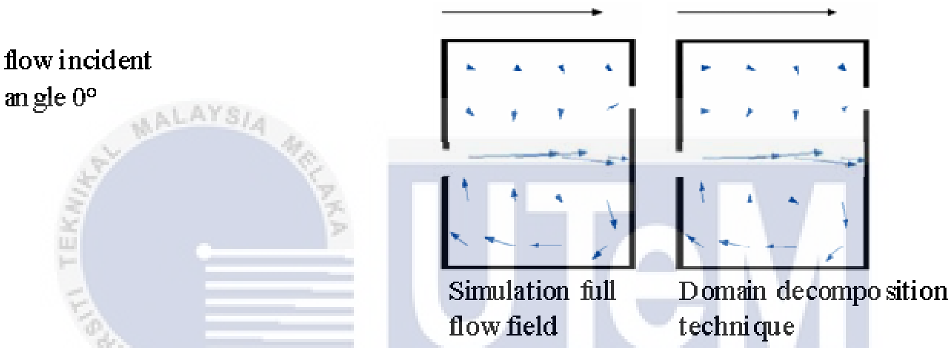

Figure 2.2: The flow rate of air through the building at full flow filed and domain decomposition technique. (Kurabuchi, Ohba and Nonaka, 2009)

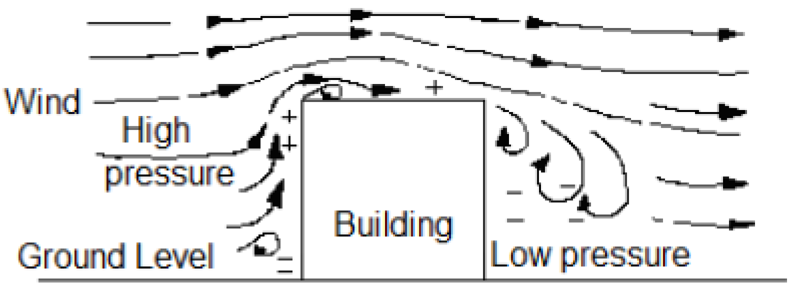

Figure 2.3: The flow rate around the building showing the high pressure region and low pressure region (Bhatia, 2014).

2.4 AIRFLOW PRESSURE

The air pressure at a given location is called a force exerted in all directions on the based on the weight of the air above it (Tiwari, 2018). The volumetric flow rate of outside air that enters the building depends on the difference pressure on the building envelope. When the wind hits the building wall, high pressure is formed at the upwind face and there is a low pressure region behind the building which is called the downwind façade (Charisi, Thiis and Aurlien, 2019). Full- scale and wind tunnel experiment is the best way to get the most accurate reading of wind pressure coefficient on current technology. The wind pressure that flows over the building can be defined as the wind pressure coefficient C_p (Charisi, Waszczuk and Thiis, 2017). The pressure coefficient can be express as:

$$P_W = \frac{1}{2}\rho U^2{}_z C_p \qquad \text{Equation 2.7}$$

Where

P_w is the pressure due to wind [Pa]

ρ is the density of air [kg/m^3]

C_p is the wind pressure coefficient at given position

U_z is the mean wind velocity at a specific high [m/s]

2.5 BOUNDARY LAYER

In fluid mechanics, the boundary is known as the thin layer of air or fluid which is near the surface and changes from zero to the free steam value away from the surface (Sampaolo, 2018). For the building cases, the boundary layer is around the floor of the building which the air velocity will increase starting from zero at the surface of the ground. For the computational fluid dynamics, the boundary layer value is able obtained from the solution generated by the Navier- Stroke Equations (Epifanov, 2011). There are two types of boundary layers which are laminar (layered) or turbulent (disordered) which is determined by the Reynolds number (Hall, 2015). The Reynold number able to determine from the formula below:

$$Re = \frac{\rho u L}{\mu} = \frac{uL}{\nu} \qquad \text{Equation 2.8}$$

Where

Re = Reynolds number

ρ = Fluid Density [kg/m^3]

u = Fluid velocity [m/s]

L = Length of the fluid [m]

μ = Dynamic viscosity of the fluid [kg/m.s]

ν = kinematic viscosity of the fluid [m^2/s]

Based on the Reynolds number calculated, the laminar is the one that got lower value which the streamwise velocity will change uniformly from the wall to the surrounding. For a higher value of Reynold numbers, it is called the turbulent and the streamwise velocity is not consistent and unsteady which will be swirling inside the boundary layer (Hall, 2015). Figure 2.4 shows the laminar and turbulent flow.

The several difference between laminar and turbulent is that the laminar flow's Reynolds number is less than 2000 while turbulent is greater than 4000. Laminar flow is very orderly and no mixing with other layers while turbulent flow is vice versa (Mishra, 2016).

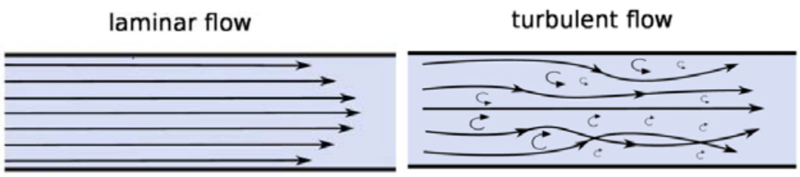

Figure 2.4 Laminar Flow (left) and Turbulent Flow through a pipe. (Harold, 2019).

2.6 COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is a science that able to produce a simulation of fluid flow based on the conservation of mass, momentum and energy with the aid of the digital computers (Sanz, 2017). Figure 2.5 shows the overall process of CFD. CFD used the Navier-strokes Equation as the governing equation. After the simulation, the simulation results will be compared to the experimental result (Zuo, 1981).

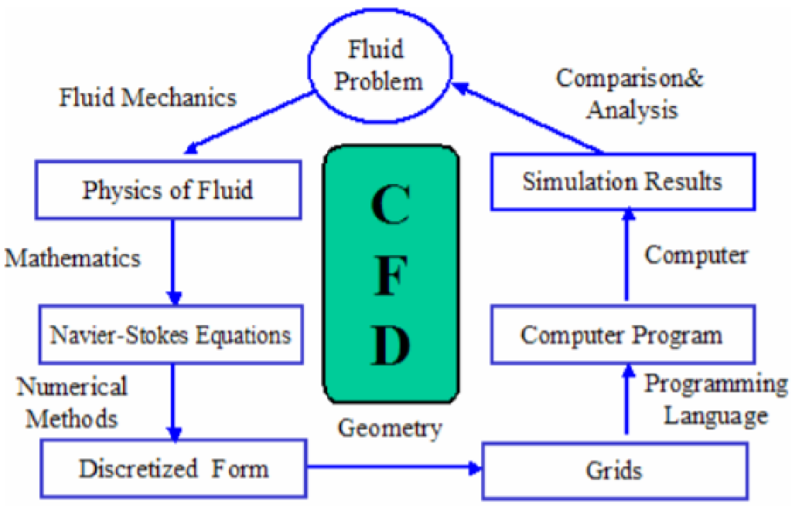

Figure 2.5 The overall process of CFD (Zuo, 1981)

2.6.1 GEOMETRY OF THE BUILDING MODEL

The building model is based on a 4 or 5- stories building in which the full-scaled dimension is assumed as 20m x 20m with 16m height. With the scale down ratio of 1:200, a 10 x 10 x 8cm of flat roof building model is used for the cross- ventilation and it is used for the atmospheric Boundary Layer Wind Tunnel (Karava, 2008). Based on Karava's work, this model is used with different opening positions to test the experimental result (Particle Image Velocimetry (PIV)). The opening area is also known by wall porosity which can be calculated by dividing the opening area over the wall area. Different wall porosity also one of the variables of this experiment (Karava and Stathopoulos, 2012). Figure 2.6 shows the different opening positions of the building model. Based on Ramponi (2012), the same building model is used to see the impact of the computational parameter to the generic isolated building. The CFD result for the full field calculation and domain decomposition method is compared to the experimental (PIV) result for Meroney's work (2009).

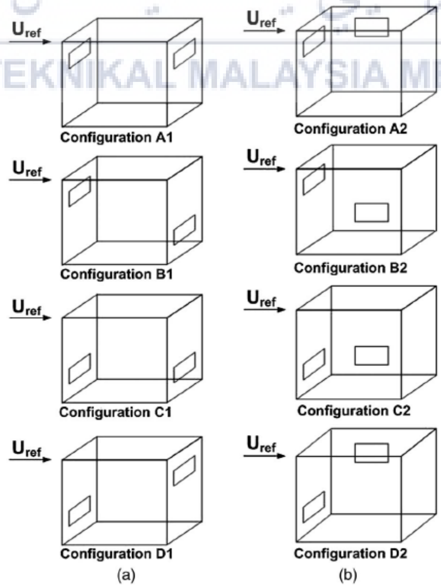

Figure 2.6 Different opening considered for the studying the effect of wall porosity and opening location on internal pressure (Karava and Stathopoulos, 2012).

2.6.2 COMPUTATIONAL DOMAIN

The computational domain able to represent the geometrical and use for the boundary condition imposition. The domain must cover up all the physical features of the model (Peles, 2014). For the building model, the computational domain for calculation of external and internal flow is a cuboid shape region that is covered up the building with 50cm tall, 150cm long and 100cm wide. The building is centered 50cm leeward from the entrance (Meroney, 2009). The domain size must be big enough because if it is too small, the flow speed will increase and reduced the pressure on the leeward wall (Oliveira and Younis, 2000). Based on Cheng (2007), the suggested domain size was 5L (L= Length of the building) upstream and 10L downstream, 5L away from each side, and 5L from the roof which is the same ratio as the Meroney (2009) cases.

2.6.3 MESHING

The grid designated the element on the model how the flow will solve and it gives a huge impact to the convergence and accuracy. A high number of cells or elements will provide a high accuracy result. There are a few types of cells such as tetrahedron, hexahedron, triangular, polyhedron (Bakker, 2002). For the building model, the wall should have finer mesh to get more details of flow and pressure (Yuan, 2007). The cell size may increase gradually when moving far away from the building because this able to reduce the simulation time and the details far away from the building are not that important compare to the one near the model. The number of cells can be increased by changing the coarse to the fine grid (Ramponi and Blocken, 2012).

2.6.4 TURBULENT MODELS

The aim of using the turbulent model is to predict the time-averaged field such as velocity, pressure, and temperature without measuring the full turbulent flow as in Reynolds- averaged Navier- Strokes Models (RANS) and Large Eddy Simulation (LES). RANS contain few different models such as Standard *k-ε* (*Sk-ε*), Realizable *k-ε* (*Rk-ε*), Renormalization Group *k-ε* (*RNG k-ε*), Standard *k-ω* (*Sk-ω*), Shear stress transport *k-ω* (*SST k-ω*), and Reynolds Stress Model (RSM). Based on Ramponi's (2012) research, the turbulence models are compared to the PIV experimental result and found out that the *SST k-ω* model which is the reference case and *RNG k-ε* model gives the best performance out of the others 6 models. The standing vortex upstream accuracy reproduces by the *SST k-ω* is slightly lower than *RNG k-ε*. Figure 2.7 shows the sensitivity analysis of the 6 models.

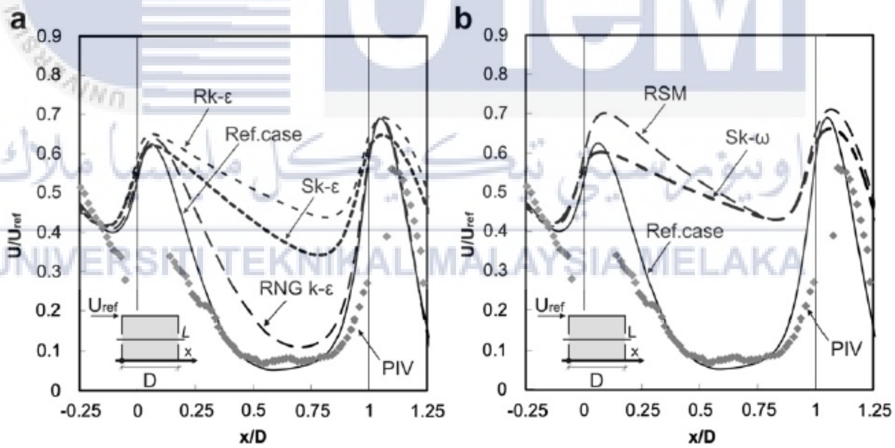

Figure 2.7 Impact of turbulence model on the streamline wind speed ratio along the centerline (Ramponi and Blocken, 2012).

2.6.5 BOUNDARY CONDITIONS

The boundary condition is required to define how the system operates such as inlet, outlet, wall, etc. A reference wind speed velocity at 6.97m/s and turbulence intensity of 10% is used (Ramponi and Blocken, 2012). This formula is used to get the boundary condition which is:

$$U(z) = \frac{u^{*}_{ABL}}{\kappa} \ln(\frac{z + z_0}{z_0}) \quad \text{Equation 2.9}$$

$$k(z) = a(I_u(z)U(z))^2 \quad \text{Equation 2.10}$$

$$\varepsilon(z) = (\frac{u^{3*}_{ABL}}{\kappa(z + z_0}) \quad \text{Equation 2.11}$$

$$\omega(z) = \frac{\varepsilon(z)}{C_\mu k(z)} \quad \text{Equation 2.12}$$

Where

Z_0 = Aerodynamics roughness length = 0.025mm

U^{*}_{ABL} = Atmospheric boundary layer friction velocity

κ = Von Karman constant (0.42)

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)

The outlet is defined as the gradient of all variables are taken to be zero and for the wall boundary condition is taken as the non-slip wall function (Endo *et al.*, 2006).

2.6.6 CFD SIMULATION METHOD FOR BUILDING MODELS

Based on previous research by Meroney (2009), there are two types of methods that able to use when doing the simulation for building models which is the full flow field result method and another one is the domain decomposition method. The difference between these 2 is the full flow field method simulates the building model inside the computational domain to get the external and internal result while the domain decomposition method is to extract the external flow field result for the boundary condition at the indoor situation. This means by the external flow field and the internal flow field is simulated separately. The external flow field result near the building is extracted and put it as the boundary condition for the building model opening location which is without the computation domain. By doing this, the result from the external flow field no needs to be repeated when the condition of the opening at the building is changed. This enables to save some time when doing the iteration because for the domain decomposition and full flow field result are just about the same.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter will cover up the entire process of the project with an illustration of the flow chart as shown in Figure 3.1. The flowchart is to show how the progress of the PSM 1 for this research run. The information obtained from the previous chapter which is the literature review is mainly to help understand more knowledge and theory of the cases that will be studied. The methodology is an important part to make sure all the progress is done step by step to achieve the objective and problem. Different thickness of the wall and comparison of the full flow field and domain decomposition is done by researching several sources. Computational Fluid Dynamics (CFD) simulation is performed at ANSYS v16.0 Fluent for all the cases. The simulation result will be compared to the PIV experimental results. The CFD setup, validation, geometrical of the building, verification are discussed in this section.

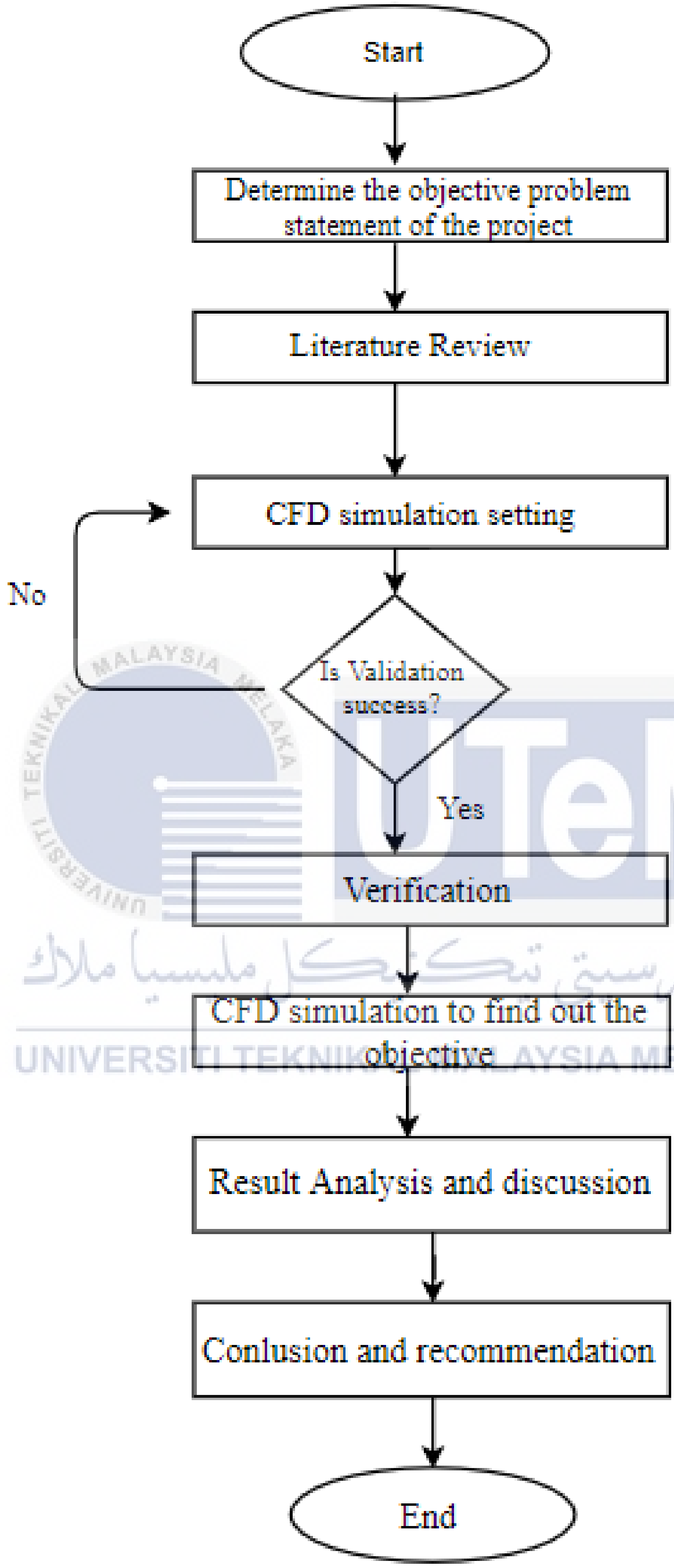

Figure 3.1: Flowchart of the methodology

3.2 GEOMETRY OF THE BUILDING MODELS

The building model chosen for this is the model that has the dimensions of width 0.1m, depth 0.1m, height 0.08m. The dimension is taken from the reduced scale of 1:200 scale of width 20m, depth 20m, and height of 16m. The model roof is to be flat. The ratio of the wall compared to the volume of the building is too big which will take it as a thin wall only. This model is used by Karava to test the experimental result which is the Particle Image Velocimetry (PIV). Due to all the previous research, most of them considered the thickness as a thin wall only, this thesis will determine how the thickness of the wall will affect the flow rate. 3 different thicknesses of the wall will be used to do the research which is 0mm. 2mm and 4mm. 4mm is chosen from the thickness of the rammed earth which the thickest is at 800mm with the ratio of 1:200 would become 4mm (D. Ciancio, 2015). As for the 2mm is the middle range for the 0mm to 4mm thickness. Figure 3.2 shows the geometry of the building model.

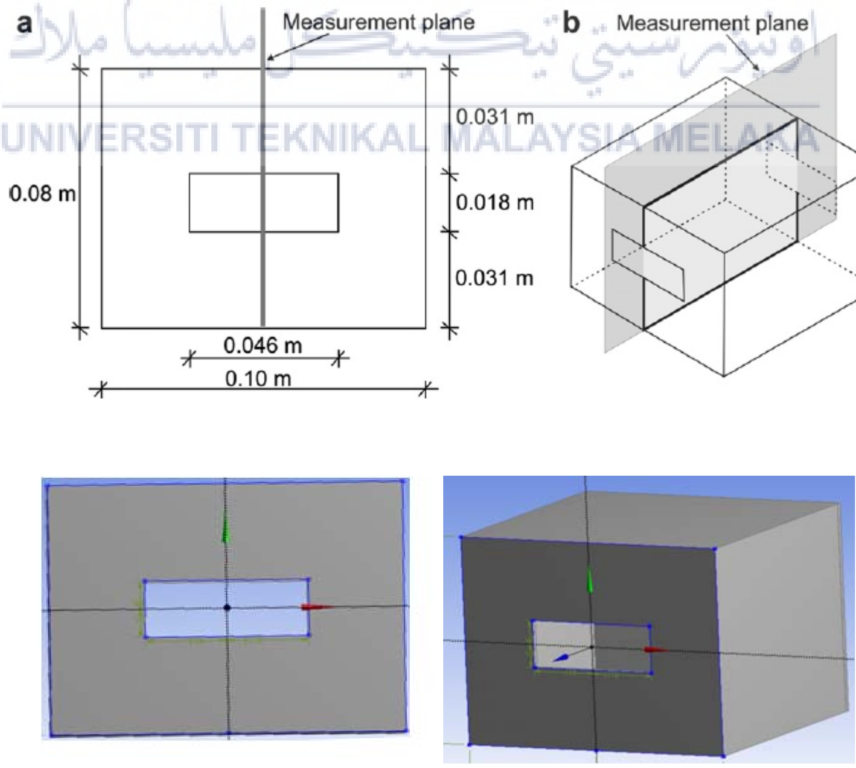

Figure 3.2 The dimension for the building geometry.

3.3 COMPUTATIONAL DOMAIN

The domain size used for this research is based on the work of Meroney (2009) which the computational domain is a rectangular region at the dimension of 0.5m tall, 1.5m long and 0.1m wide. The computational domain is shown in Figure 3.3 below. Based on this computational domain, the windward façade of the building is 0.5m which is 5L of the building model, while the leeward façade of the domain range is 10L of the building model, 1m. The height of the domain is at 0.5m which is 6.25H of the building model which is 0.08m for the height of the building model. For both sides of the building model is at 0.1m, which is 0.05m at each side, this is corresponding to 5W of the building model width. Based on the recommendation from previous work by Cheng (2007), the windward façade and the side of the building model must be 5 times of the building length, L and width, W, while the height of the domain is also 5 time the height of the building model. The downstream must be 10L. For the domain used for this thesis, the domain has fulfilled the requirement based on previous research. The full sizing is cut into half using the YZ plane due to it is symmetry as both sides have the same dimension and cutting it into half able to reduce the simulation time because both sides will give the same result due to the same dimension and same shape. Figure 3.4 shows the model is being cut into half.

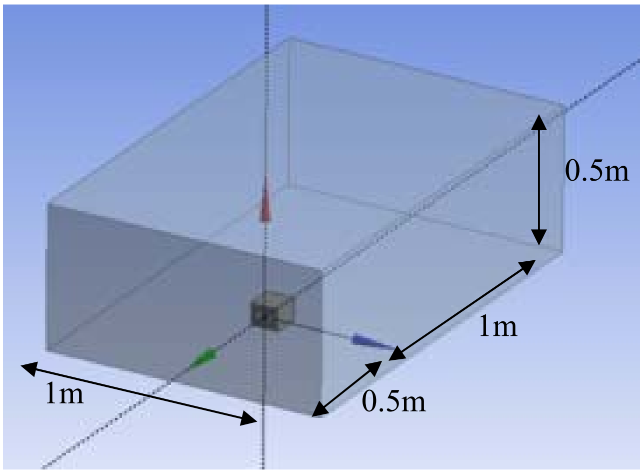

Figure 3.3 Full sizing of Computational Domain with Building Model.

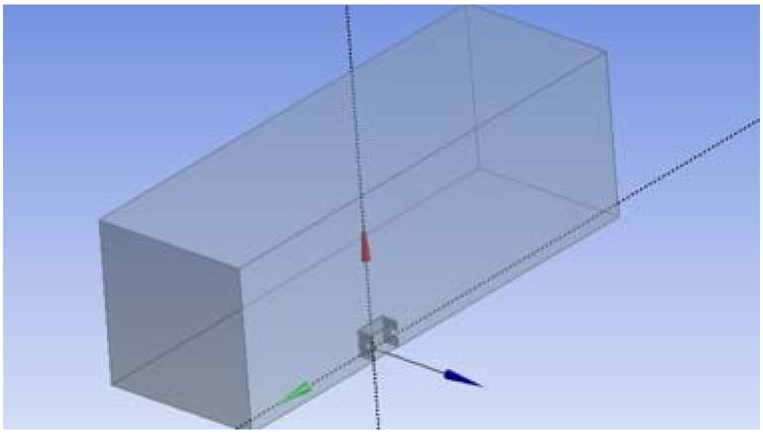

Figure 3.4 The Computational Domain cut into half by YZ plane.

3.4 MESHING

By referring to the finding of the previous research stated in section 2.6.3, the domain was set up at around 300,000 elements and 300,000 nodes. This amount of elements able to give an accurate result based on previous research which is done by Ramponi and Blocken, 2012. By using symmetry on the YZ plane also able to reduce to element size number. There are 3 regions required finer meshing a better accurate data gathering: the exterior surface of the building which includes the leeward of the model and the edge of the wall of the building model. The other one is at the surface of the building (internal). These are the three regions that were the main focuses to capture the flow around the domain.

The structured mesh is being used for this building model and the computational domain and building model is being sliced to evenly distributed shape. This is one of the main criteria that must be achieved before structured mesh can be done. After the slicing, the number of divisions is being used to make how fine is the element size. Figure 3.5 shows the structured mesh.

To make the building model element size smaller, higher number of division is being used which is at 26 divisions for the length while the opening(inlet) of the building model is 9 divisions while the wall of the model is 13 divisions.

To reduce the element waste around the edges which is far away from the building, different bias type is being used at the computational domain with a different number of bias factors. Figure 3.7 shows the element size around the computational domain and building. To get a nice result around the building, a small size of elements will be at the surround of the building with the help of a bias type.

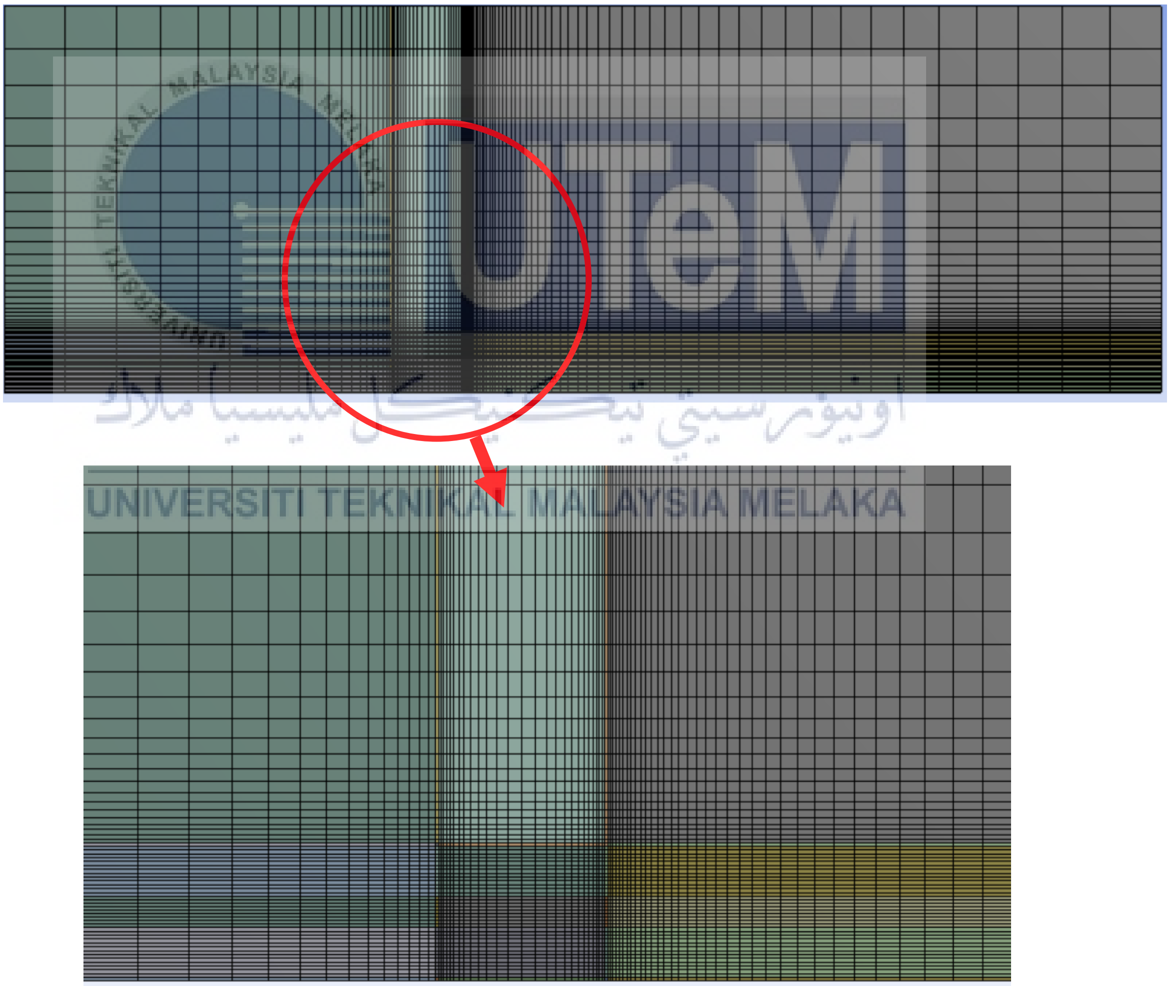

Figure 3.5: Structured Meshing at the computational domain and building model

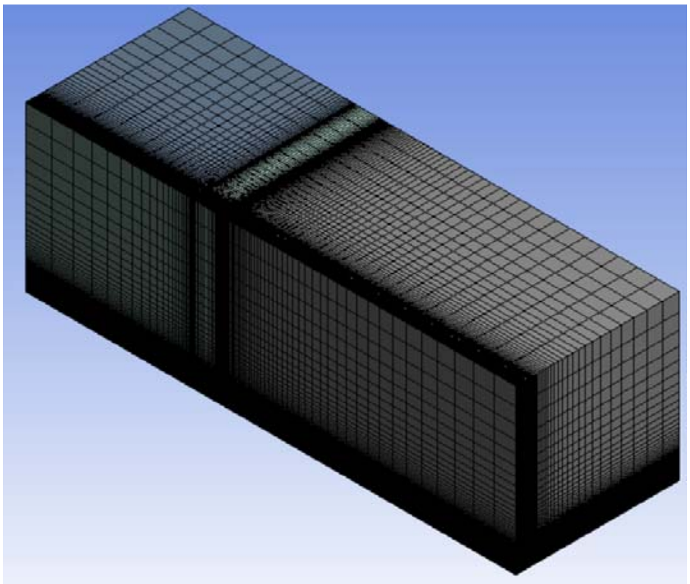

Figure 3.6: Strucutred mesh isometric view

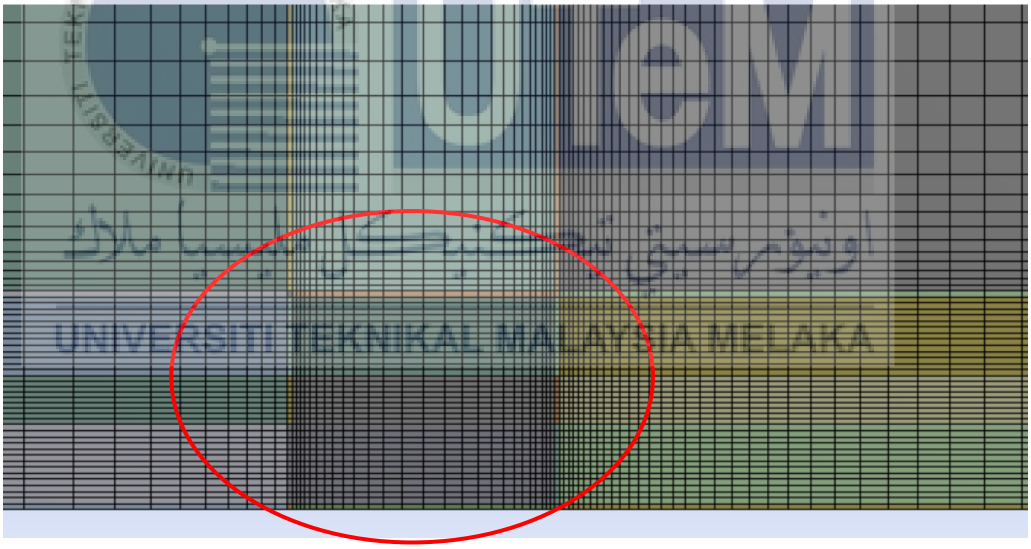

Figure 3.7: Smaller element around the Building Model.

3.5 TURBULENT MODELS

Based on the research done by Ramponi (2012) that is stated in section 2.6.4, the Shear- Stress Transport *k-ω* and RNG *k-ε* were tested in the simulation. From the result of comparison based on the research, it showed that the result from the Shear- stress Transport *k-ω* turbulence model has higher sensitivity in capturing the velocity around the building model compared to RNG *k-ε*. Hence, these 2 models are taken to do the simulation to see whether it will give the same result as the research stated in section 2.6.4.

3.6 BOUNDARY CONDITION

The boundary conditions are followed closely according to the setting by (Ramponi and Blocken, 2012) which is stated in Section 2.6.5. The wind was defined as the air which is at 1.225 kg/m^3 in density and 1.8 x 10^{-5} $kgm^{-1}s^{-1}$ viscosity which is the default setting at the ANSYS v16 software. Steady flow is chosen rather than the transient flow. No-slip condition is considered at the boundary condition which can be said as a solid body. The reference wind speed and turbulence intensity are fixed at 6.97 m/s and 10% (Building entrance height) (Ramponi and Blocken, 2012). Based on the information at section 2.6.3, the formula as shown below is used to calculate the u^*_{ABL} atmospheric boundary layer velocity, *k* the turbulent kinetic energy, *ε* the turbulence dissipation rate and *ω* the specific dissipation rate the z- coordinate is assumed to be fixed at the opening height which is 0.04m.

For the domain decomposition part, after running the external flow field result, the velocity near the front building is extracted. The value is put as the boundary condition at the domain decomposition method where there is no computational domain for it. This enable us to save a lot of time for internal flow field research as the number of elements decreases. For the ground surface boundary condition, roughness height for the ground surface is calculated using equation 2.13 which the C_s is the roughness constant at 0.874 The equation below is the same as the equation stated at 2.6.5

$$U(z) = \frac{u^*{}_{ABL}}{\kappa}\ln(\frac{z+z_0}{z_0}) \qquad \text{Equation 2.9}$$

$$k(z) = a(I_u(z)U(z))^2 \qquad \text{Equation 2.10}$$

$$\varepsilon(z) = (\frac{u^{3*}{}_{ABL}}{\kappa(z+z_0})) \qquad \text{Equation 2.11}$$

$$\omega(z) = \frac{\varepsilon(z)}{C_\mu k(z)} \qquad \text{Equation 2.12}$$

$$k_s = \frac{9.793z_0}{C_s} \qquad \text{Equation 2.13}$$

Where

Z_0 = Aerodynamics roughness length = 0.025mm

U^*_{ABL} = Atmospheric boundary layer friction velocity

κ = Von Karman constant (0.42)

z = Height coordinate (0.08m)

k = Turbulent kinetic energy

I_u = Streamwise turbulence intensity

a = parameter 1 is used in this case

ε = Turbulence dissipation rate

ω = Specific dissipation rate

C_μ = Empirical constant (0.09)

C_s = Roughness constant (0.874)

Based on the formula above, an UDF (User Defined Function) was edited and put it into ANSYS software (D.M. Hargreaves, 2007). Figure 3.7 shows the UDF profile which will be used at the inlet boundary condition.

```
#include "udf.h"
#define UREF 6.97 /* ref. speed in m/s */
#define CMU 0.09
#define VKC 0.4
#define ZREF 0.08 /* ref. height in m */
#define Z0 0.000025
#define ir 0.1

DEFINE_PROFILE(velocity_profile, thread, position)
{
        float x[ND_ND];
        float y;
        float u, u_star;
        face_t f;


        u_star = UREF*VKC/log((ZREF+Z0)/Z0) ; // ref [1]

        begin_f_loop(f, thread)
        {
                F_CENTROID(x,f,thread);
                y=x[1];
                u = u_star/VKC*log((y+Z0)/Z0);
                F_PROFILE(f,thread,position) = u;
        }
        end_f_loop(f, thread)
}


          /*  profile for kinetic energy  */


          DEFINE_PROFILE(k_profile, thread, nv)
          {
                 float x[ND_ND];
                 face_t f;
                 real u;
                 real i = 0.1;
                 float y;


                 float u_star ;

                 u_star = UREF*VKC/log((ZREF+Z0)/Z0) ; // ref [1]


                 begin_f_loop(f, thread)
                 {

                         F_CENTROID(x,f,thread);
                         y=x[1];
                         u = u_star/VKC*log((y+Z0)/Z0) ;
                         F_PROFILE(f,thread,nv)=1*pow((u*i),(2));

                 }
                 end_f_loop(f, thread)
          }
```

```
/* profile for dissipation rate  */

DEFINE_PROFILE(dissip_profile, thread, position)
{
        float x[ND_ND];
        face_t f;
        float u_star, y ;

        u_star = UREF*VKC/log((ZREF+Z0)/Z0) ; // ref [2]

        begin_f_loop(f, thread)
        {
                F_CENTROID(x,f,thread);
                y=x[1];
                F_PROFILE(f,thread,position)=pow(u_star,3.)/(VKC*(y+Z0));
        }
        end_f_loop(f,thread)
}
```

Figure 3.8 : UDF file for the boundary condition at the inlet.

3.7 SOLUTION METHODS AND MONITORS

The Simple Algorithm was applied for Pressure-Velocity Coupling. For the spatial discretization, the gradient used the Least Square Cell-Based for Gradient. The pressure interpolation is set as second order. The momentum, Turbulent Kinetic Energy and Specific Dissipation Rate are set as the second-order wind. Based on the Ramponi (2012) research, the second-order discretization scheme is to get better accuracy although first-order discretization scheme able to convergence faster in unstructured tetrahedral and pyramid cells, but the accuracy is not very accurate. The monitors for residuals was set to absolute convergence criterion. The absolute criteria for residuals of continuity, x- velocity, y- velocity, z- velocity, energy, k and ω were 0.00001.

3.8 VALIDATION

The validation of the CFD simulation settings was done by employing the Grid Independent Test and the turbulent model comparison toward the experimental result and previous research results. The most suitable CFD settings are selected to ensure the CFD simulations run the most effective and efficient way.

3.8.1 GRID INDEPENDENT TEST

In the Grid Independent Test, three different grid sizes were applied to study the grid sensitivity: coarse, medium, fine mesh. Each of the grid qualities differed from each other by 25% in the grid size. Table 3.1 shows the detail of each mesh quality setting.

Table 3.1: Grid Independent Test

Grid Quality	Coarse	Medium	Fine
Grid Size	125%	100%	75%
No. Of Element	150,920	297,724	503,388
Nodes	159,900	311,850	523,600
Velocity Ratio(u/U_{ref})	0.418	0.403	0.399

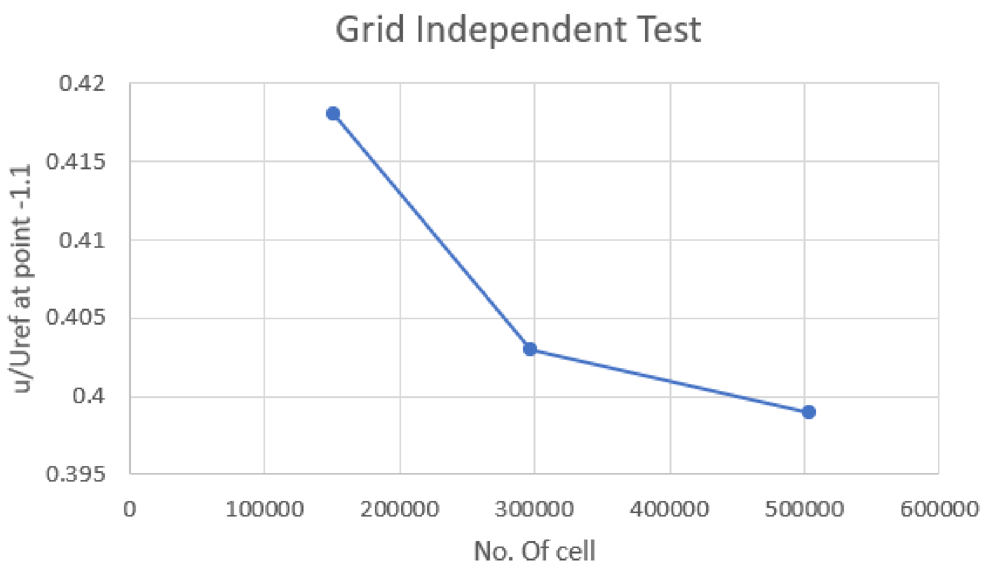

Figure 3.9: Velocity ratio at point-1.1 versus number of element

The Medium mesh quality increased the number of elements and nodes by 97.27% and 95% compared to the coarse mesh quality. The velocity ratio increased at around 3.6% when compared to the coarse mesh quality. The fine mesh quality increased the number of elements and nodes by 69.07% and 67.90% compared to medium mesh quality. The velocity ratio difference at only 1% compared to the medium quality. The change is not significant but the simulation increased up to 2 hours if compared to medium mesh quality. Therefore, the medium mesh quality will be chosen as the most practical application in the CFD simulation.

3.8.2 TURBULENT MODEL COMPARISON

The CFD result get from the ANSYS software will be compared to the journal's result which is close to the experimental result. There are 2 results needed to be compared which is the qualitative and quantitative results. Based on the figure below, it can see that the SST k-ω has the same result as the reference case, but for the RNG k-ε turbulence model, the flow at the interior is a bit underpredicted as the flow go to low, but still acceptable. The different flow inside the building is mainly due to the jet entering the building. Figure 3.10 shows the qualitative result based on 2 turbulence model. The quantitative result will decide which turbulence model will be used by comparing the velocity ratio to the x-axis of the building model.

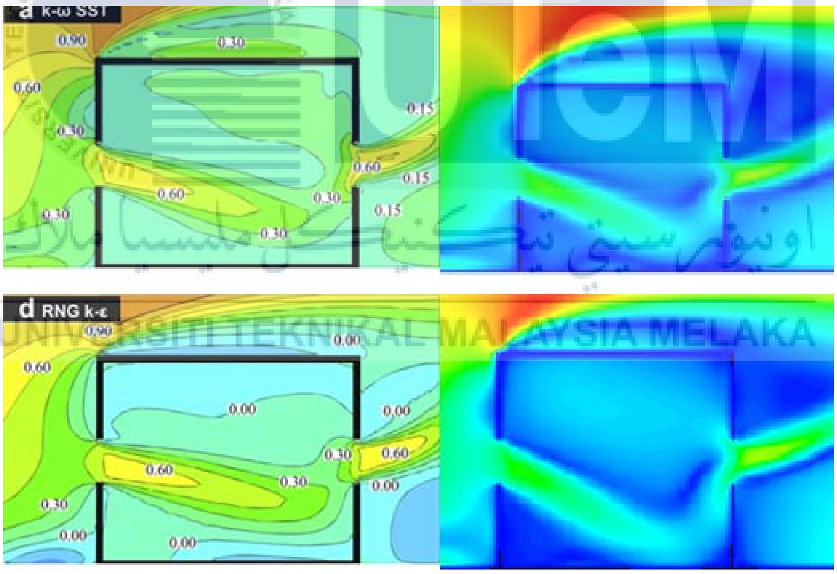

Figure 3.10 Different velocity contour compared to the reference case.

Based on the information stated in Section 2.6.4, different turbulence models are taken to compare which models provide the best result based on the reference case which is the SST k-ω model that the result is the closest to the experimental result. The best result might be different from previous research due to element cell count is different. Based on the Figure below, it can see that the outdoor flow rate for both turbulence models got the almost same result, but the SST *k-ω* model over predicted the outlet flow a little compared to RNG *k-ε* but it still acceptable as the main concern is the interior flow. For the internal area, SST *k-ω* got a better result compare to RNG *k-ε* which is SST *k-ω* over predicted the velocity up to 0.5 times (x/D= 0.4) while the RNG *k-ε* also overestimated but it is up to 2 times (x/D= 0.4). At x/D= -0.7, RNG *k-ε* also unable to get the same profile as the experimental and previous research done by Ramponi (2012). Therefore, SST *k-ω* is chosen for the turbulence model.

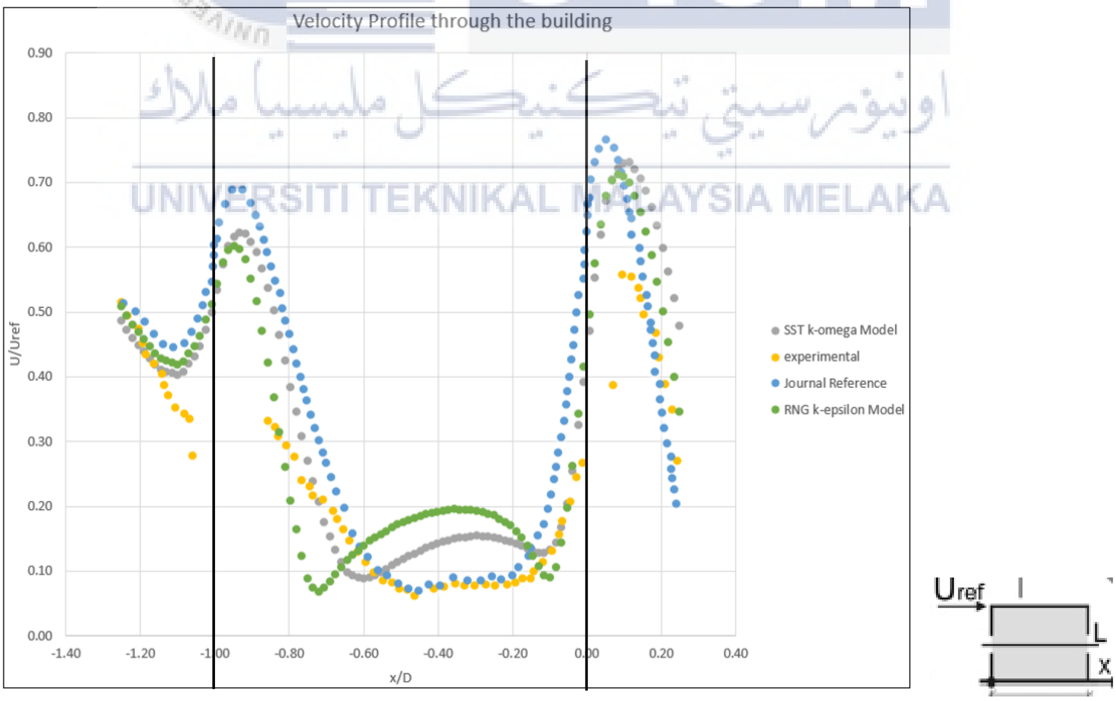

Figure 3.11 Comparison of Turbulence model (RNG k-ε and reference case)

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 THICKNESS OF THE WALL

There are 3 thicknesses of the building of the wall that going to be compared which is the 0mm (thin wall), 2mm, and 4mm. The 4mm is the thickness of the rammed wall which is at around 400mm at the real building. In CFD simulation, the size had been scaled down to 1:200. Therefore, it will be 4mm in CFD simulation. While the 2mm is the middle range of 0mm to 4mm.

4.2 VELOCITY PROFILE FOR DIFFERENT THICKNESS OF THE WALL

Based on figure 4.1 shows the velocity contour for different thicknesses of the wall of the building, a total of 3 thicknesses which is 0mm, 2mm, and 4mm thickness. These qualitative results showed the overall results are the same which does not have any significant difference by changing the thickness of the wall. Although the simulation time might be increase when adding in the thickness of the wall as the cell or element number increased.

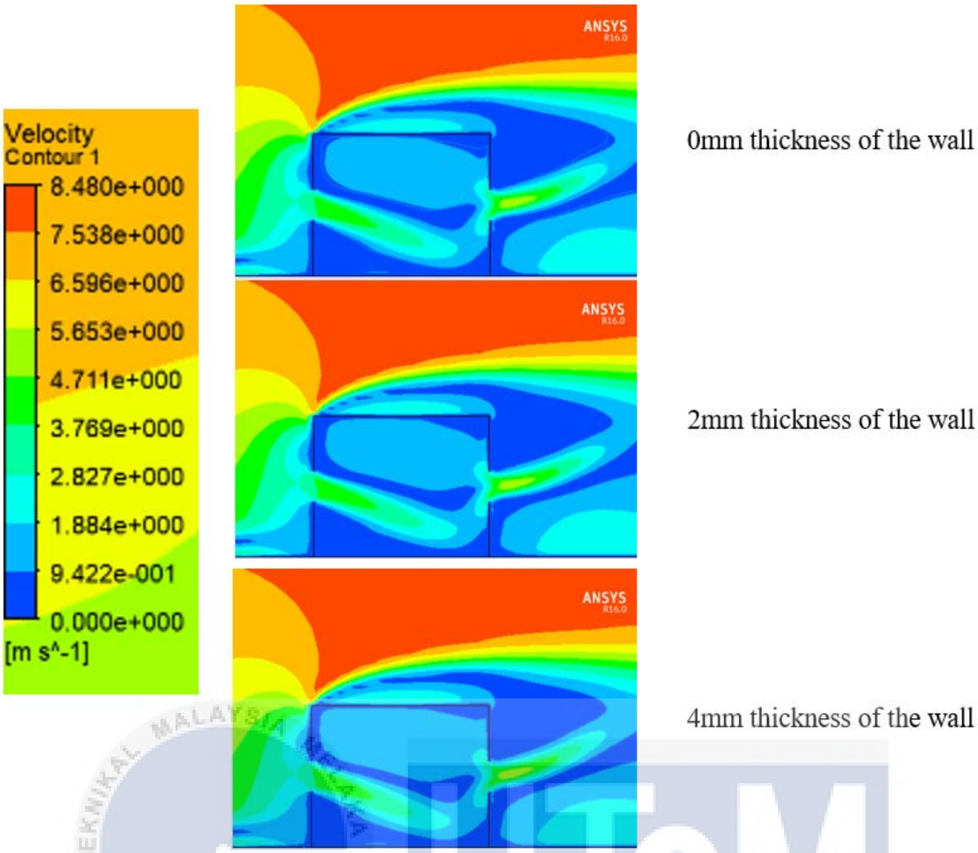

Figure 4.1 Velocity contour for different thickness of the wall of the building

Based on the figure below, comparing the 3 different types of the thickness of the wall with the same setting that has been used for the medium mesh quality and using the SST *k-ω* model. From the figure, the graph trend for 3 types is the same which gives the same result, but the slightly different is at around x/D=-0.7 point the velocity ratio for 0mm thickness is at 0.177, 2mm thickness at 0.153 and 4mm thickness of wall velocity ratio is at 0.129. The largest difference is when comparing 4mm thickness to 0mm thickness which the difference is up to 37%. If by comparing the 2mm to 0mm thickness the difference is at 18%. For other cases at x/D= -0.5, the velocity ratio for 0mm is at 0.11, 2mm thickness at 0.113 and 4mm thickness at 0.129. At this point, the largest difference is when comparing 4mm to 0mm which is at around 15% difference while when comparing

2mm with 0mm and thickness the difference is small which is at 15% and 2%. Other than between these 2 points, the other difference is below 5%.

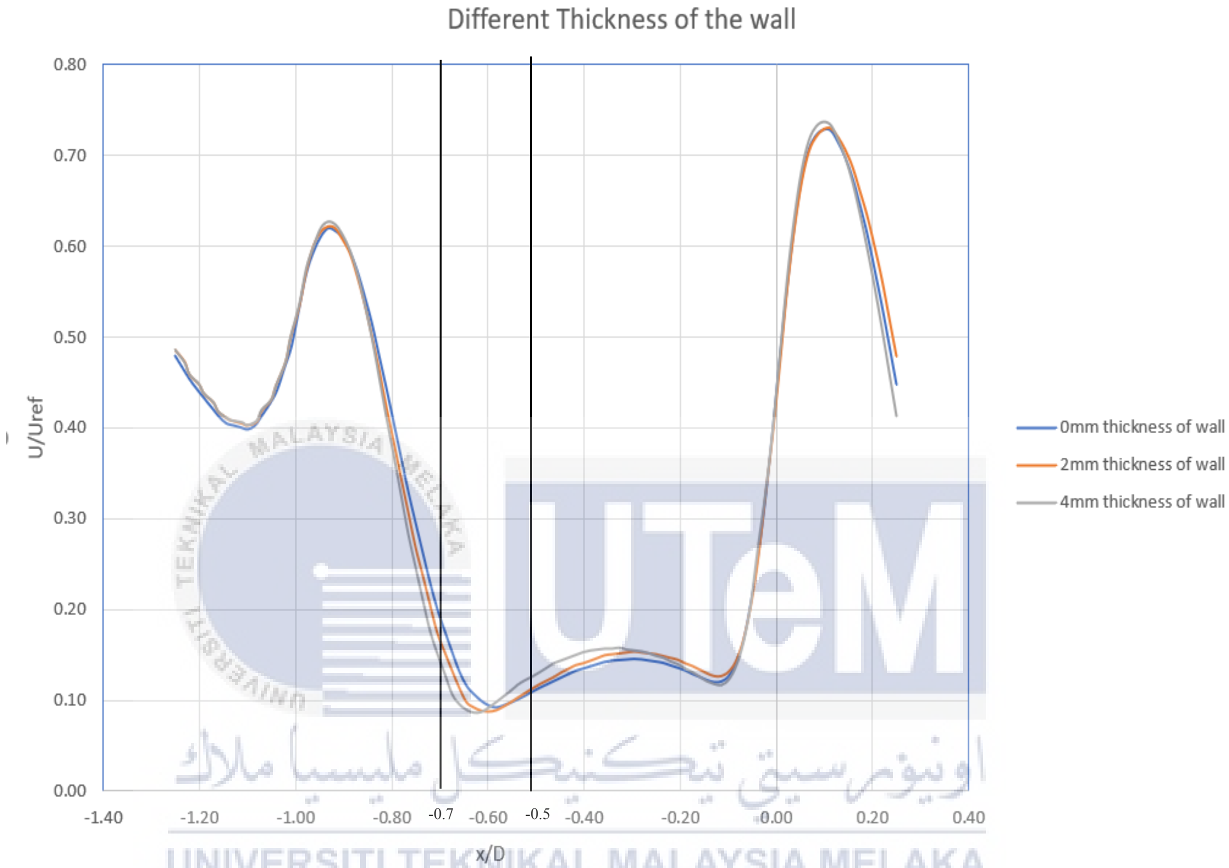

Figure 4.2Comparison between different thickness of wall in the velocity profile

Table 4.1 Average velocity at the surround of opening to outlet of the building

Thickness of the wall of the building model(mm)	Average velocity at the x/D from -1.25 to 0.25 (m/s)
0	2.424
2	2.443
4	2.401

From the table above, the average velocity at the inlet to outlet is almost the same which is at around 2.401 to 2.443. The difference between 0mm thickness to 2mm thickness is 0.8% only while the difference between 2mm to 4mm thickness is at 1.74%. Hence, it can be said that the difference average velocity between the 0mm to 4mm thickness wall of the building model is significantly small.

4.3 VOLUME FLOW RATE FOR DIFFERENT THICKNESS OF THE WALL

Based on the discussion at 2.3, The airflow rate is also one of the most important factors that need to consider if wanted to have a good ventilation rate for the building. Therefore, the parameter for the volume flow rate is being considered in this study. The table below shows the volume flow rate at the opening normal to the open surface with the opening area.

Table 4.2 The volume flow rate for different thickness of the wall

Thickness of the wall of the building model(mm)	Volume flow rate Q ($x10^{-3}$) (m^3/s)
0	1.380
2	1.391
4	1.397

From the table shown above, the variable thickness of the wall of the building gives approximately the same volume flow rate at the simulation. The difference between the 0mm to the 4mm is at around 1.23%. while the difference between 0mm thickness of the wall to the 2mm is around .8% only. From the analysis, the thickness of the wall of the building model does not give significant changes in the volume flow rate when simulating in CFD as the difference between the thickness of the wall to the volume flow rate is lesser than 2%.

4.4 DOMAIN DECOMPOSITION METHOD FOR DIFFERENT THICKNESS OF THE WALL

The same thickness of the walls which is 0mm, 2mm, and 4mm is taken to do the test for the domain decomposition method. This method is to simulate without the computational domain but extracting the boundary condition from the computational domain to put it at the inlet of the building only. Figure 4.3 shows the result for the domain decomposition and the result get is around the same but the most obvious difference is at the point where x/D = -0.7. The velocity ratio, u/U_{ref} at this point is 0mm thickness at 0.0841, 2mm thickness at 0.0908, 4mm thickness at 0.0878. The largest difference is when comparing the 2mm thickness to 0mm thickness which is at around 8% difference. For the point at x/D= -0.15, the value for 0mm, 2mm, 4mm is at 0.177, 0.185 and 0.180. The largest error is around 4.5%. Figure 4.4 shows the qualitative result which is the velocity contour with various thicknesses of the wall of the building model. The overall is about the same and there is only a slight difference near the inlet there.

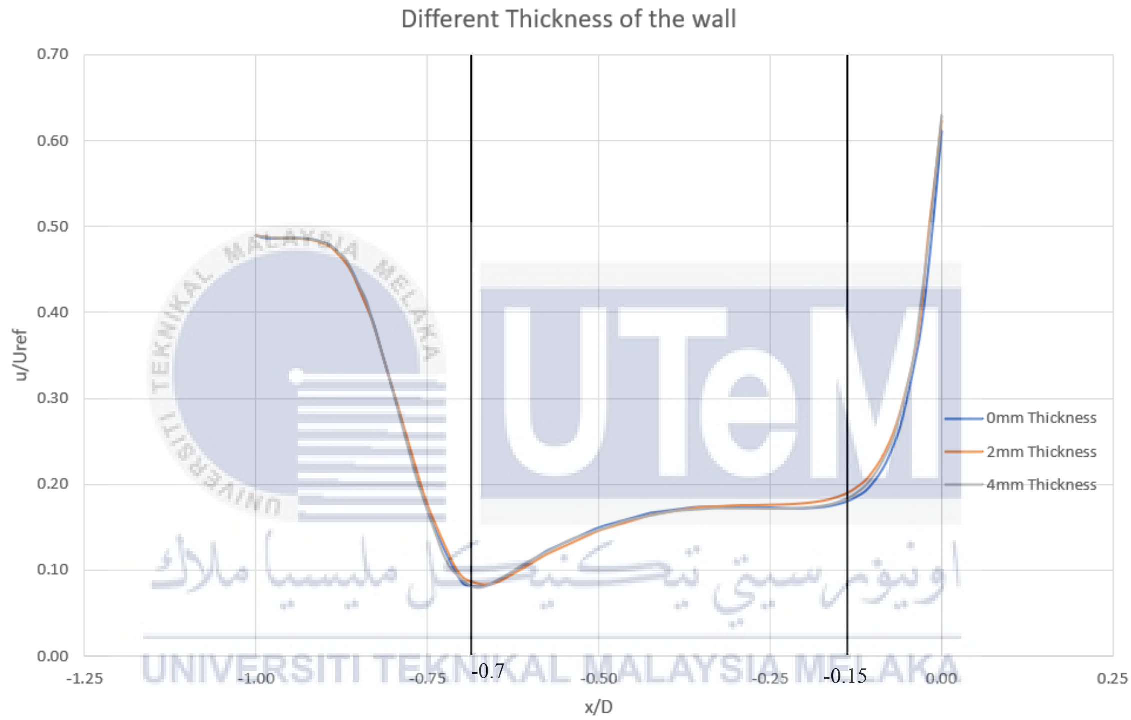

Figure 4.3: Domain Decomposition method for different thickness of the wall

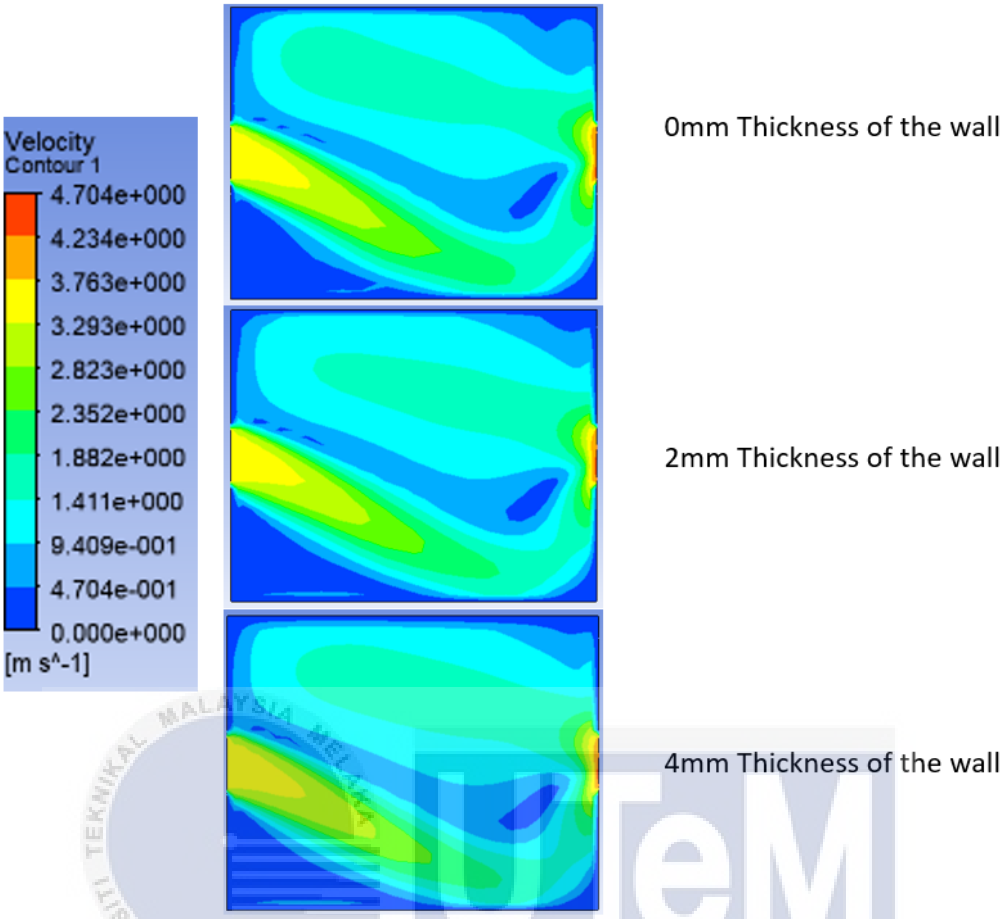

Figure 4.4: Velocity contour for the domain decomposition method for different thickness of wall

Table 4.3 Average velocity at the opening to the outlet of the building.

Thickness of the wall of the building model(mm)	Average velocity at the x/D from -1.00 to 0 (m/s)
0	1.704
2	1.666
4	1.660

For the average velocity at the interior of the inlet to outlet of the building model, the average shown in the above table. The largest difference is when comparing the 0mm thickness of the wall to the 4mm thickness where the difference is at 0.044m/s or approximately 2.6%. If comparing the 0mm to 2mm thickness, the error is at most 2.3% only. The value decreased if compared to the nearer thickness value.

Table 4.4 The volume flow rate for different thickness of the wall using domain decomposition method

Thickness of the wall of the building model(mm)	Volume flow rate Q ($x10^{-3}$) (m^3/s)
0	1.3428
2	1.3402
4	1.3432

From the table above, the volume flow rate based on the thickness of the wall of the building model for 0mm and 2mm thickness of the wall show approximately 0.2% of the error while the 2mm to 4mm only difference at 0.0032 or 0.3% only which can be assumed as equal volume flow rate.

4.5 COMPARISON BETWEEN DOMAIN DECOMPOSITION METHOD VS FULL FLOW FIELD

After comparing the thickness with the different ways of simulation, the next comparison will be the same thickness but different ways of simulation. The figure below shows the result from the domain decomposition and full flow field with 2mm thickness of the wall. From the figure below, the overall trend is the same, but the velocity at the inlet there is much lower at the domain decomposition method while the full flow field velocity at the inlet is much faster. As for figure 4.6, the velocity profile trend is the same just that the inlet velocity and at the point x/D =-0.14 shows some different result. For the inlet part (x/D = -0.93), the velocity ratio for full flow field is at 0.622 while the domain decomposition is underpredicted at 0.486. The difference is at 0.136 or 23% difference. The large difference might due to that the domain decomposition method does not experience the vena contracta phenomena that lead to an increase in velocity after entering. The next point for the differences is at x/D=-0.14 where the full flow field velocity ratio at 0.130 while the domain decomposition at 0.187. The difference is at 0.57 or approximately 30% difference.

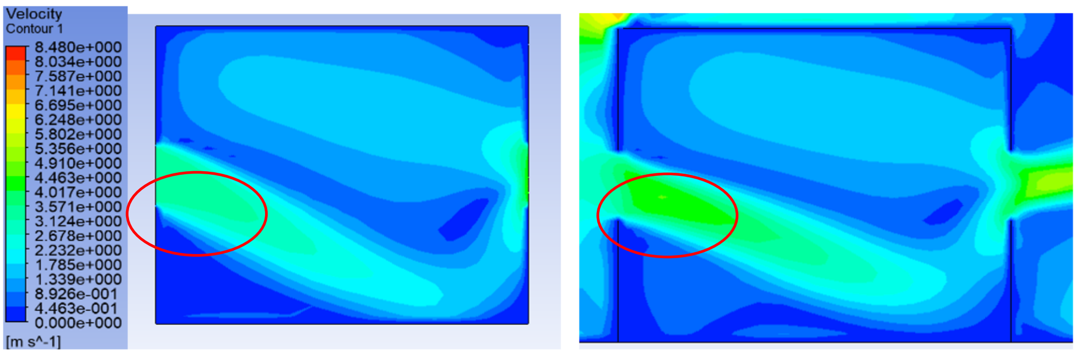

Figure 4.5 Comparison of domain decomposition method and full flow field for 2mm thickness of the wall

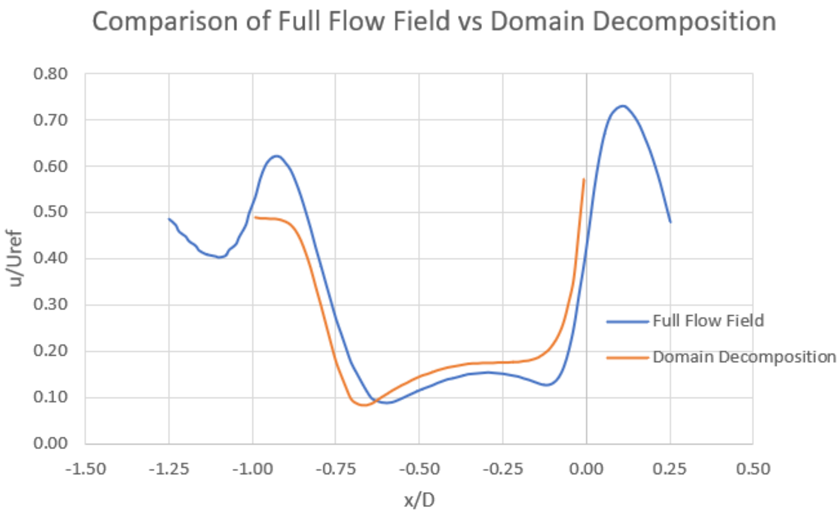

Figure 4.6 Velocity profile for domain decomposition and full flow field

Table 4.5 Comparison of Volume flow rate for difference type of simulation

Type of simulation	Volume flow rate Q (x10^{-3}) (m^3/s)
Full Flow Field	1.3912
Domain Decomposition	1.3402

From the table above, both types of simulation give the around same volume flow rate which is at full flow field 1.3912 m^3/s and domain decomposition at 1.3402 m^3/s. The difference is at approximately 4% when comparing these 2 results.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

The airflow rate through a building can be done by using either one of the methods which are the full flow field method and domain decomposition method. Both of them were able to get a good result which is similar to the experimental result which previously the researcher has done before. From the comparison of different thicknesses of the wall of the building, the result is about the same when comparing it with 3 thicknesses result which is the 0mm 2mm, 4mm (ratio 1:200). Both methods used for different thickness of the wall of the building also gives the same result. In conclusion, both methods, full flow field and domain decomposition will get the same result as experimental result and thickness of the wall of the building at the simulation would not affect the airflow rate throughout the building.

5.2 RECOMMENDATION

This study only focused on the room space in the building, but for the real case, it might have more room in a building. Therefore, different amount of room space of the building would be more interested. The domain decomposition method currently used average flow rate at the inlet of the building only but to get a better result a User Define Function should be added. By considering the weather condition would be more interesting such as considering the humidity, temperature at the surrounding of the building.

REFERENCES

Asfour, O. S. (2015) 'Natural ventilation in buildings: An overview', *Natural Ventilation: Strategies, Health Implications and Impacts on the Environment*, (January 2015), pp. 1–25.

Awbi, H. B. (1996) 'Air movement in naturally-ventilated buildings', *Renewable Energy*, 8(1–4), pp. 241–247. doi: 10.1016/0960-1481(96)88855-0.

Bakker, A. (2002) *Applied Computational Fluid Dynamics (Meshing)*, *Bakker*. Available at: http://www.bakker.org/dartmouth06/engs150/07-mesh.pdf.

Bastide, A. *et al.* (2006) 'Building energy efficiency and thermal comfort in tropical climates. Presentation of a numerical approach for predicting the percentage of well-ventilated living spaces in buildings using natural ventilation', *Energy and Buildings*, 38(9), pp. 1093–1103. doi: 10.1016/j.enbuild.2005.12.005.

Bhatia, A. (2014) 'HVAC – Natural Ventilation Principles Credit : 4 PDH', *Amazon*, (877).

Briney, A. (2018) *The Pressure Gradient Force and Other Effect on Wind*, *ThoughtCo*. Available at: https://www.thoughtco.com/winds-and-the-pressure-gradient-force-1434440.

Burnett, J., Bojić, M. and Yik, F. (2005) 'Wind-induced pressure at external surfaces of a high-rise residential building in Hong Kong', *Building and Environment*, 40(6), pp. 765–777. doi: 10.1016/j.buildenv.2004.08.019.

Charisi, S., Thiis, T. K. and Aurlien, T. (2019) 'Full-scale measurements of wind-pressure coefficients in twin medium-rise buildings', *Buildings*, 9(3). doi: 10.3390/buildings9030063.

Charisi, S., Waszczuk, M. and Thiis, T. K. (2017) 'Investigation of the pressure coefficient impact on the air infiltration in buildings with respect to microclimate', *Energy Procedia*. Elsevier B.V., 122, pp. 637–642. doi: 10.1016/j.egypro.2017.07.362.

Cheung, J. O. P. and Liu, C. H. (2011) 'CFD simulations of natural ventilation behaviour in high-rise buildings in regular and staggered arrangements at various spacings', *Energy and Buildings*. Elsevier B.V., 43(5), pp. 1149–1158. doi: 10.1016/j.enbuild.2010.11.024.

Daniela Ciancio Senior Lecturer. (2015, April 30). Cheap, tough and green: Why aren't more buildings made of rammed earth? Retrieved March 06, 2020, from https://theconversation.com/cheap-tough-and-green-why-arent-more-buildings-made-of rammed-earth-38040

D.M. Hargreaves, N.G. Wright / J. Wind Eng. Ind. Aerodyn. 95 (2007) 355–369, doi:10.1016/j.jweia.2006.08.002

Endo, T. *et al.* (2006) 'Development of a simulator for indoor airflow distribution in a cross-ventilated building using the local dynamic similarity model', *International Journal of Ventilation*, 5(1), pp. 31–42. doi: 10.1080/14733315.2006.11683722.

Epifanov, V. m. (2011) *Boundary Layer*, *Thermopedia*. doi: 10.1615/AtoZ.b.boundary_layer.

Hall, N. (2015) *Boundary Layer*, *National Aeronautics and Space Administration*. Available at: https://www.grc.nasa.gov/WWW/K-12/airplane/boundlay.html.

Harold, G. (2019) *Laminar Flow vs. Turbulent Flow*, *Diffzi*. Available at: https://diffzi.com/laminar-flow-vs-turbulent-flow/.

Jiru, T. E. and Bitsuamlak, G. T. (2010) 'Application of CFD in modelling wind-induced natural ventilation of buildings - A review', *International Journal of Ventilation*, 9(2), pp. 131–147. doi: 10.1080/14733315.2010.11683875.

Karava, P. (2008) *Airflow Prediction in Buildings for Natural Ventilation Design: Wind Tunnel Measurements and Simulation*.

Karava, P. and Stathopoulos, T. (2012) ‘Wind-induced internal pressures in buildings with large façade openings’, *Journal of Engineering Mechanics*, 138(4), pp. 358–370. doi: 10.1061/(ASCE)EM.1943-7889.0000296.

Kurabuchi, T., Ohba, M. and Nonaka, T. (2009) ‘Domain decomposition technique applied to the evaluation of cross-ventilation performance of opening positions of a building’, *International Journal of Ventilation*, 8(3), pp. 207–217. doi: 10.1080/14733315.2009.11683846.

Li, Y. and Nielsen, P. V. (2011) ‘Commemorating 20 years of Indoor Air: CFD and ventilation research’, *Indoor Air*, 21(6), pp. 442–453. doi: 10.1111/j.1600-0668.2011.00723.x.

Meroney, R. N. (2009) ‘CFD prediction of airflow in buildings for natural ventilation’, *11th Americas Conference on Wind Engineering*.

Mishra, P. (2016) *Difference Between Laminar and Turbulent Flow*, *Mechanical Booster*. Available at: https://www.mechanicalbooster.com/2016/08/difference-between-laminar-and-turbulent-flow.html.

Norhidayah, A. *et al.* (2013) ‘Indoor air quality and sick building syndrome in three selected buildings’, *Procedia Engineering*. Elsevier B.V., 53(2010), pp. 93–98. doi: 10.1016/j.proeng.2013.02.014.

Oliveira, P. J. and Younis, B. A. (2000) ‘On the prediction of turbulent flows around full-scale buildings’, *Journal of Wind Engineering and Industrial Aerodynamics*, 86(2–3), pp. 203–220. doi: 10.1016/S0167-6105(00)00011-8.

Peles, Y. (2014) ‘Computional Domain’, *Encyclopedia of Microfluidics and Nanofluidics*, pp. 1–7. doi: 10.1007/978-3-642-27758-0_170-2.

Ramponi, R. and Blocken, B. (2012) ‘CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters’, *Building and Environment*. Elsevier Ltd, 53, pp. 34–48. doi: 10.1016/j.buildenv.2012.01.004.

Sampaolo, M. (2018) *Boundary layer*, *Encyclopedia Britannica*. Available at: https://www.britannica.com/science/boundary-layer.

Sanz, W. (2017) *Computational Fluid Mechanics*. Fifth Edit, *Fluid Mechanics*. Fifth Edit. Elsevier. doi: 10.1016/B978-0-12-382100-3.10010-1.

Straube, J. (2008) 'Building Science Digest 014 Air Flow Control in Buildings', *Ashrae Handbook*, pp. 1–18.

Tiwari, P. (2018) *Atmospheric Pressure: Measurement, Distribution and Controlling Factors*, *geographynotes*. Available at: http://www.geographynotes.com/atmosphere/atmospheric-pressure-measurement-distribution-and-controlling-factors/854.

Wang, E., Wang, H., Deng, M., Wang, K., & Wang, Y. (2017). Simulation of the ventilation and energy performance of a PV-integrated breathing window. *Procedia Engineering, 205*, 2779-2784. doi:10.1016/j.proeng.2017.09.880

Yuan, C. S. (2007) 'The effect of building shape modification on wind pressure differences for cross-ventilation of a low-rise building', *International Journal of Ventilation*, 6(2), pp. 167–176. doi: 10.1080/14733315.2007.11683775.

Zuo, W. (1981) 'Introduction of Computational Fluid Dynamics', *Inc., September*, pp. 81--1.