CERTIFICATION OF APPROVAL

A CFD INVESTIGATION INTO THE THERMAL COMFORT OF BUILDING ENVELOPE

By

Muhammad Asyiq bin Ansorudin

A project dissertation submitted to the

Mechanical Engineering Program

Universiti Teknologi PETRONAS

In partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved by,

____

(Dr Ir Mohd Shiraz b Aris)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

December 2010

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgments, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

MUHAMMAD ASYIQ BIN ANSORUDIN

ABSTRACT

Heating, ventilating and air-conditioning (HVAC) is a type of system that seek to provide thermal comfort for occupants in a conditioned building envelope, or supplying a set of environmental condition (temperature, humidity, etc.) for a process within a space. The system heavily involves with transfer of heat, mass by means of distribution.

One of the primary functions of HVAC design is to establish comfort of occupants by regulating indoor environment whilst maintaining indoor air quality, room air distribution and operational cost.

Improper usage of HVAC system is usually to met the desired environmental condition, but at the cost of the excess power and energy and may have a higher operational cost. A good energy audit must be carried out to assess a specific building envelope for its energy management.

Using the computational fluid dynamic (CFD) software, engineers can observe and approximate heat transfer and fluid flow within the building envelope, effectively enabling them to analyze further on the thermal comfort required and the energy consumption efficiency. Using American Society of Heating, Refrigeration and Air Conditioning Engineers (AHSRAE) standards, engineers can determine the proper range for controlled environmental conditions required inside a building envelope to achieve a comfortable indoor environment. The energy consumption that lead to operation cost can also be reduce if the analysis is successful in maintaining a balance of energy required for ventilation, thus no excess usage of energy will happen.

ACKNOWLEDGEMENT

First and foremost, I would like to take this opportunity to extend my highest gratitude to my God for his blessing and helpfulness in doing this project. I would also like to extend my gratitude to everyone who's directly and indirectly responsible for lending me assistance and guidance throughout the whole period of completing my final year project.

I would also like to extend my thanks to my cherished supervisors, for without his everlasting supports, guidance and time allocation throughout the whole completion period; I would never have finished the project successfully.

Next, many thanks to Prof Vijay Raghavan and Ms Chin Yee Sing as the examiners for their tireless supports, comments and assistance that further enhance my project.

I would also like to thank Dr Saravanan a/l Karuppanan as the Final Year Project coordinator that had made essential arrangements for this course.

Finally, thanks to the campus, staff, technician, peers and family members for the unrelenting supports, ideas and inspiration throughout the completion of this study. Thank you.

TABLE OF CONTENT

LIST OF FIGURE

LIST OF TABLE

CHAPTER 1

INTRODUCTION

1. INTRODUCTION

1.1. PROJECT BACKGROUND

A building envelope refers to enclosed conditioned space which thermal energy is transferred to or from the outdoor environment, usually by means of HVAC system. The building envelope selected for the study is the UTP mosque which operates a regulated air conditioned system. This study requires collection of data from the actual environment and numerical simulation of a building envelope model. A sufficient collection of data must be gathered that includes building size, insulation material and the designed conditions of ventilation that serve as a boundary condition for modeling purposes.

A site study must be performed to gather the actual environmental data of the building envelope using auditing equipment. This data is required for comparison with the numerical simulation data obtained for validation purposes. The entire data gathered will be utilized further for the assessment of the thermal comfort performance of the building envelope.

1.2. PROBLEM STATEMENT

Thermal comfort analysis inside a building envelope is essential nowadays to ensure the thermal comfort condition is satisfied. By doing CFD investigation of thermal comfort inside a building envelope, better understanding of fluid flow and heat transfer can be achieved within that enclosed space.

1.3. OBJECTIVE AND SCOPE OF STUDY

The objectives of this work are to carry out a numerical simulation on thermal comfort inside the UTP mosque by using ANSYS FLUENT software.

The scope of study involved would be to develop a model of the UTP mosque and simulating the indoor environment condition by using the boundary condition to observe the pattern of temperature distribution and the air flow. The boundary condition is gathered by doing measurement on the actual indoor environment of the building envelope. This study will then analyze the thermal comfort pattern inside the building envelope. The result of the study will then be assessed to compare with the thermal comfort condition given by AHSRAE standards.

1.4. SIGNIFICANCE OF PROJECT

The significance of this project is that in the future, UTP and the mosque administration would be able to refer to this project as a benchmark and be able to implement energy management with the data that is found in this project. UTP mosque operates a regulated air conditioning system, and susceptible for excessive energy supply for its HVAC system due to many factors. The mosque administration as well as universities would be able to use this research to update the environment condition required for thermal comfort whilst maintaining low operational cost and energy consumption.

CHAPTER 2

LITERATURE REVIEW

2. LITERATURE REVIEW

2.1. INTRODUCTION

This chapter will emphasize on the thermal comfort, building envelope, CFD modeling and simulation.

2.2. HISTORY

During the 1990's, attempts were made to establish and specify conditions to provide a comfortable environment. During the period of 1913 to 1923, Professor John Sheppard, at Teacher's Normal College in Chicago introduced the term comfort zone [1]. The New York State Commission on ventilation began its experiments [2] and the American Society of Heating and Ventilating Engineers published the first work of Houghten and Yaglou which established lines of equal comfort, defined effective temperature and determined a comfort zone.

The comfort zone was defined as including these effective temperatures (ET) over 50% of the people are comfortable. On this basis, the zone limit were found to be 16.7^{o} C and 20.6^{o} C. with a comfort line at 17.8^{o}C (ET). The effective temperature is defined as an arbitrary index which combines into a single value, the effect of dry bulb temperature, humidity and air motion on the sensation of warmth or cold by human body. The first comfort chart was published in 1924.

Modification at comfort chart resulted from work published in 1929 by Yaglou and Drinker [4], the summer comfort zone was found to be 18.9^{o}C and 23.8^{o}C ET, votes indicating comfort not just 50% or more.

2.3. THERMAL COMFORT

The primary function of most of air conditioning systems is to provide and maintain a good thermal comfort for the occupants in a building envelope. Human thermal comfort is “the state of mind that expresses satisfaction with the surrounding environment” (ASHRAE Standard 55). The human thermal comfort is influenced by the rate of body heat loss, which is affected by its surrounding environment.

The rate of heat loss is primarily affected by five major conditions:

- Air temperature
- Relative humidity
- Air motion
- Temperature of surrounding objects
- Clothing and insulation

The air conditioning systems can manage thermal comfort by adjusting three conditions: temperature, humidity, and air motion. By setting lower temperature, the rate of heat loss will increase by convection, thus ensue a good thermal comfort. It also applies to humidity, where it can be lowered so that perspiration process can easily evaporate into surrounding air effectively taking heat with it. Air motion can also be increased to allow more air movement, thus increase the potential of heat loss by means of convection.

By the process metabolism, heat is constantly generated within the body while on other hand, loss of heat is constantly occurring from the surface of body by convection, radiation, evaporation and conduction. The human body maintains the heat balance with its environment through minor physiological change (i.e. increasing and decreasing flow of blood to the skin).

Human body is likely more sensitive to the change of temperature rather than relative humidity and there is less conclusive evidence to show that either high or low humidity is detrimental to the health of normal people. A study conducted by University of California Berkeley, [6] on thermal comfort at high relative humidity shows that there are no significant physiological or psychological difference in human response to exposure of between 60% to 90% relative humidity for the temperature range of 20^{o}C to 26^{o}C while sedentary.

There are other factor which influenced the thermal comfort such as reciprocating factor which include diet, clothing, exposure and activity; physical factors such as light, area-volume, radiation, inspired gas, atmospheric pressure, air motion, temperature, relative humidity, body type, sensory process and genetic.

There are three basic conditions for optimal thermal comfort. The first one for a person under long exposure is the existence of a heat balance which is usually not sufficient under extreme condition. Man's thermoregulatory system is quite effective and will therefore create heat balance within wide limits of the environment variables even if comfort does not exist. With the establishment of a double heat balance, an equation is obtained:

$$f = \left(\frac{H}{A_{du}}, I_{cl}, t_{db}, t_{mrt}, RH, V, t_s, \frac{E_{sw}}{A_{du}}\right) = 0 \ldots\ldots\ldots\ldots (1)$$

Where;

F = Function of thermal comfort criteria

H/A_{du} = Internal heat production per unit body dubious area

I_{cl} = Thermal resistance of clothing

T_{db} = Dry bulb temperature

T_{mrt} = Mean radiant heat temperature

RH = Relative humidity

V= Air velocity

T_s = Mean skin temperature

E_{sw}/A_{du} = Heat loss per unit body surface area by evaporation sweat discretion

For a given activity level, the t_s and E_{sw} are seen to be the only physiological variables influencing the heat balance in equation (1). The sensation of thermal comfort has been relation to the magnitude of these two variables.

There is study involving a group of people at different activity level of skin temperature and sweat discretion as functions of activity level, for persons in thermal comfort. The result has the following term.

$$t_S = f\left(\frac{H}{A_{du}}\right) \ldots\ldots\ldots\ldots (2)$$

$$E_{sw} = A_{du} f\left(\frac{H}{A_{du}}\right) \ldots\ldots\ldots\ldots. (3)$$

Equation (2) and (3) are presented as the second and third basic condition for thermal comfort. By substituting (2) and (3) in (1), the desired comfort equation takes the following term:

$$f = \left(\frac{H}{A_{du}}, I_{cl}, t_{db}, t_{mrt}, RH, V\right) = 0 \dots \dots \dots \dots (4)$$

Using comfort equation (4), it is possible for any activity level and any clothing insulation to calculate all condition at t_{db}, t_{mrt}, RH and V which will create optimal thermal comfort.

The recommendation are made by AHSRAE Standard 55 [7] are shown in Table 1. These environmental conditions should guarantee at least 90% of occupants feel thermally satisfied.

Table 1: ASHRAE Standard recommendations.

	Operating Temperature	Acceptable Range
Winter (0.9 clo)	22^{o}C	20-23^{o}C
Summer (0.5 clo)	24.5^{o}C	23-26^{o}C

These conditions were assumed for a relative humidity 50%, a mean relative velocity lower than 0.15m/s, a mean radiant temperature equal to air temperature and a metabolic rate of 1.2 met.

The acceptable range of operative temperature and humidity for the winter and summer is further defined on the psychrometric chart in figure below. This study will determine the temperature and velocity inside the mosque and plot the particular values into the summer condition. If it is within the summer condition region, then the numerical simulation result is thermally comfort (within thermal comfort region).

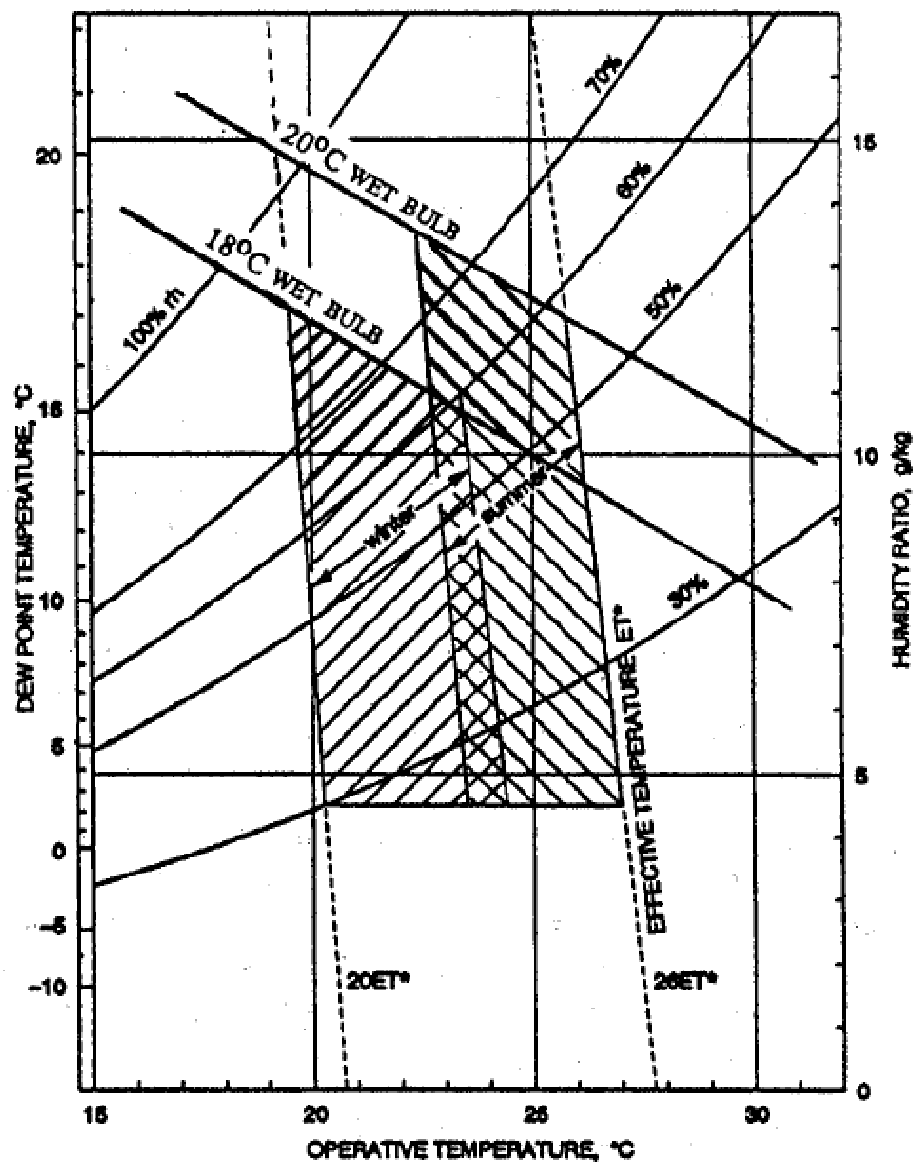


Figure 1: Acceptable range of operative temperature and humidity for people in typical summer and winter clothing during light, primarily sedentary activities. The operative temperature ranges are based on a 10% dissatisfaction criterion. [5]

2.4. BUILDING ENVELOPE

Building envelope is an enclosure of a conditioned building component that allows thermal energy transfer to or from the surrounding. For this study, the thermal energy transfer rate is referred to heat loss. The building envelope selected for this project is the UTP mosque. The details of the building envelope should be considered to design a proper air conditioning system, and it is also required for CFD modeling purposes.

The following details must be taken into account to define the model of the building envelope:

- Building size (wall area, roof area, glass area, floor area)
- Type of material and insulation (wood, glass, metal, bricks)
- Type of fenestration (light transmitting partition)
- Types of doors (amount of air infiltration)
- Types and location of lighting
- Inlet velocity, temperature
- Location of ventilation and air conditions

These parameters will not only serve as a boundary condition for the model, but also assist in determining thermal comfort performance and characteristic of building envelope energy consumption.

2.5. CFD MODELING AND SIMULATION

A finite volume technique is a process which a model of physical configuration is developed. It permits computer modeling prior to prototype building. In this study, the software used for CFD modeling and simulation is FLUENT. There are five requirements to be considered for finite volume modeling:

- Model geometry
- Material properties
- Meshing
- Grid independence
- Load cases and boundary conditions

This study requires finite elements analysis to calculate temperature distribution and air flow. The model of building envelope will predict every boundary condition such as floor area, wall insulation, roof area, fenestration area, cooling loads and so forth. This data consist of constant data (such as material properties, insulation R-value) and varied data (cooling loads, inlet temperature).

The effect of external outdoor must also be considered. This includes fenestration area which contributes to radiation heat transfer. The effect of daylight also influences the fenestration rate, depending on the time of the day. The simulation must also vary the data for the effect of fenestration.

CHAPTER 3

METHODOLOGY

3. METHODOLOGY

3.1. PROJECT PLANNING

This section explains the work that has been done so far for the project. For reader information, the explanation is will be more on data gathering and CFD simulation.

Firstly, select a building envelope to be investigated and gathered boundary condition data for the envelope for further modeling. Secondly, by using ANSYS FLUENT, generate a model of the building envelope using the boundary conditions and load gathered earlier. After the model has been generated, simulate the model according to the specified loads and external effect (air infiltration, fenestration effect). Thirdly, perform a site study on the building envelope to measure actual indoors conditions for validation. Lastly, investigate the thermal comfort characteristic of the building envelope using both data. Based on the data, conclude all the finding and contribute ways on how to reduce energy consumption and improve energy efficiency of the building envelope.

3.2. TOOLS/EQUIPMENT REQUIRED

The tools and equipment which are required in this project are Microsoft Office and FLUENT which is used to model and simulate the data obtained from the site; equipment needed basically would be data from on site results as well as other references, notably AHSRAE standards.

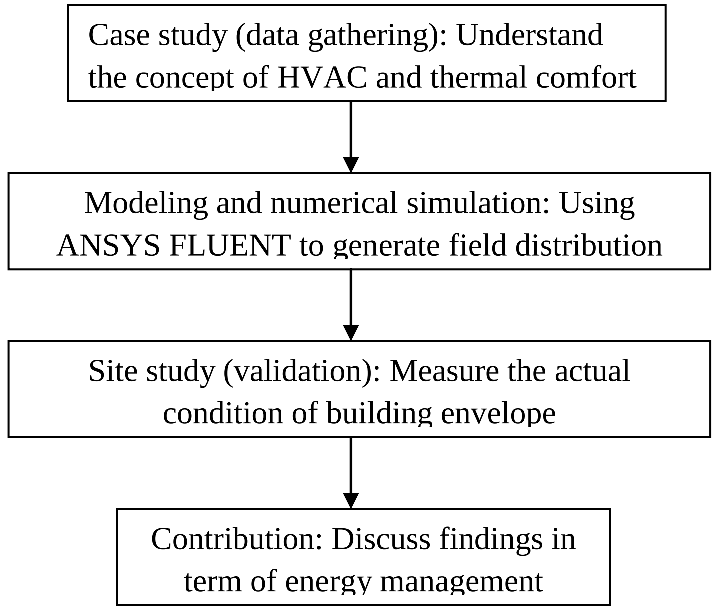


Figure 2: Methodology Flow Chart

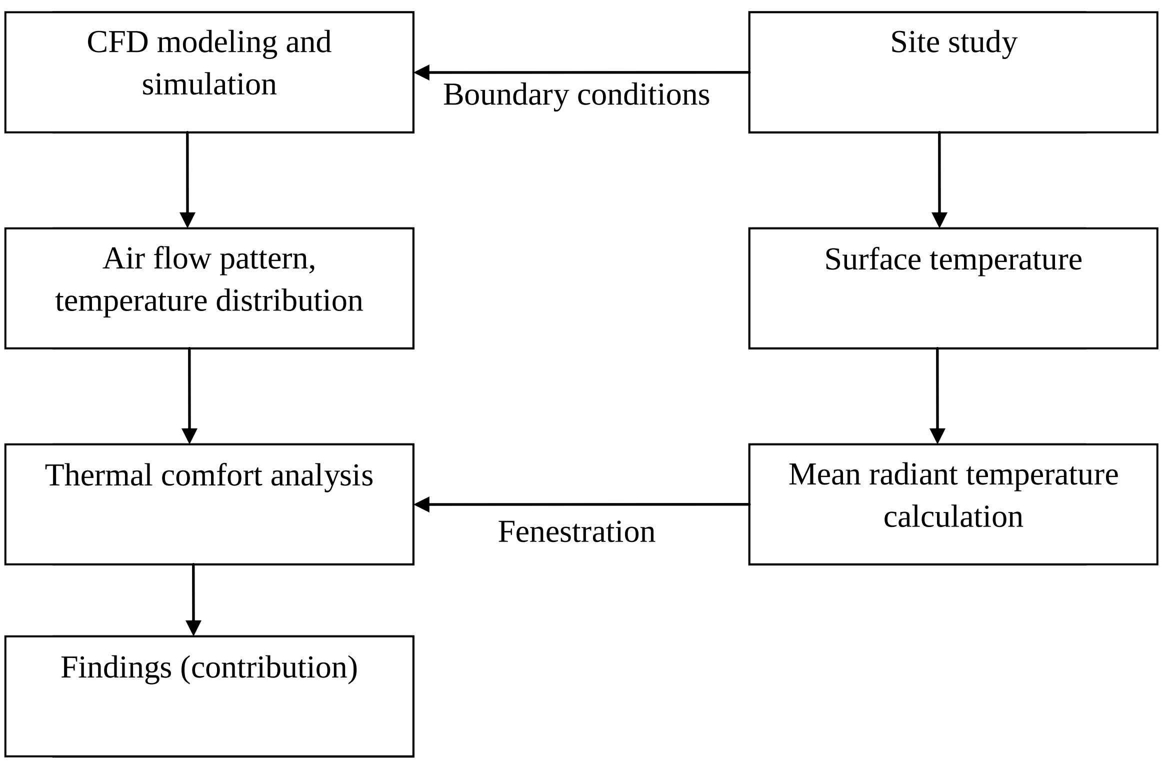


Figure 3: Overall Methodology

3.3. PROJECT MILESTONE

Table 2: Project Gantt chart

No.	Detail/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Project Work Continue		■	■											
	-3D model simulation of building envelope sample														
2	Submission of Progress Report 1					●									
3	Project Work Continue				■	■	■	■							
	-Modeling and meshing of mosque by importing from CATIA - Finding environmental data either by site study or use of TRNSYS software														
4	Submission of Progress Report 2								●						
5	Project work continue								■	■	■	■			
	-Validating thermal conditions inside mosque														

	- Analyzing findings														
6	Submission of Dissertation Final Draft												●		
7	Oral Presentation													●	
8	Submission of Project Dissertation														●

● Suggested milestone

Process

CHAPTER 4

MATHEMATICAL MODEL

4. MATHEMATICAL MODEL

4.1. INTRODUCTION

Mathematical model used in CFD will be discussed and the governing equation, solution technique used in ANSYS FLUENT related to the study will be describe in this chapter.

4.2. TRANSPORT EQUATION

The equation that describes the flow of fluid and heat within an envelope are based on the conservation of mass, momentum and thermal energy (Hazim, 2003.). The equations take the form of partial differential equations. Each equation describes the conservation of one dependent variable within the field. Two dimensional (2D) models are used in this study. The two dimensional (2D) incompressible, steady and turbulent flows in the x-y-plane with heat generation are modeled. The equation used in this study is in the form of Cartesian coordinate system. The velocity and temperature are predicted by solving the equations.

The conversation of mass of a steady and incompressible flow within the control volume dx and dy is given as:

- Continuity equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

By applying the law of conservation of momentum (i.e. the net force on the control volume in any direction equals the efflux of momentum minus the influx of momentum in the same direction.) in the x and y directions:

- Momentum equation (Navier-Stokes equations)

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

$$u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

The conservation of thermal energy in the control volume dx and dy describes that the net increase in internal energy in the control volume equals the net flow of energy by convection plus the net inflow by diffusion and also heat generation:

- Energy equation

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}=\alpha\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial^2 y}+\frac{\partial^2 T}{\partial^2 z}\right)+\frac{q}{\rho c}$$

Where u, v, p and T are the average velocities in x and y directions, average pressure and temperature respectively; while ρ, ν, α, c, q and g are density, kinematic viscosity, thermal diffusion coefficient, specific heat, rate of heat generation of occupants/unit volume and acceleration of gravity, respectively.

4.3. NUMERICAL METHOD

Examples of CFD programs that can be used to predict the air flow pattern inside the building envelope are FLUENT, PHOENICS, VORTEX. In this study, FLUENT software was used to solve the above equations.

For the testing purposes, a simple building envelope was constructed in tetrahedral mesh using GAMBIT software. The model is made up of simple rectangular box to represent a room with one occupant inside it. An inlet velocity of 1 m/s and wall functions at all surfaces were used in all the simulation. The effect of turbulence was taken into account by k-ε turbulence model. This is widely used and the dimensional model based on transport equations for turbulent kinetic energy and the dissipation of turbulence kinetic energy.

In FLUENT, there are some important parameters to be considered. For instances, the model must enable heat transfer energy due to heat generation of occupants. The fluid flow inside the model was incompressible flow with constant air properties. Boundary condition is important parameters that need to be carefully selected. Since different part of the room envelope experienced different sun irradiance, the heat coefficient of outside wall vary with time. Floor surfaces are concrete with a constant temperature which is 2^{o}C lower than the ambient temperature. The ambient temperature was determined to be 30^{o}C and the inlet air speed is assumed to be 1 m/s.

GAMBIT is used to help analysts and designers build and mesh models for FLUENT. GAMBIT receives user input by means of its graphical user interface (GUI). The GAMBIT GUI makes the basic steps of building, meshing and assigning zone types to a model. It is simple and intuitive, yet is versatile enough to accommodate a wide range of modeling applications. For the project, the model room will be modeled and meshed for different location, size and number of inlets and outlets and the boundary conditions need to be defined before export to FLUENT.

4.4. FLOW CHART OF CFD ANALYSIS

Figure 3 presented the flow chart of the CFD model analysis. First, GAMBIT software was used to model and to mesh the building envelope. The meshed model was the export to FLUENT.

Input parameters in FLUENT need to be determined carefully to provide accurate and reliable results. The initial values of the simulation model will influence the convergence of the simulation.

At each iterative step, the convergence of the numerical solution was checked. An iterative process is converged when further iteration will not produce any change in the values of dependent variables. The relaxation factors should be considered before determining whether or not a solution has converged. FLUENT provides guidance on how to determine the convergence of the solution via monitoring the residual value and the histories residual and solution variables.

For this study, the residual set for convergence used is the default values in FLUENT. The residual set for convergence is shown in the table below, and is compared with the residual of an equation at iteration. If the residual is less than the specified value in the table, that equation is deemed to have converged.

Table 3: The residual set for convergence.

Residual	Criteria
Continuity	0.001
X velocity	0.001
Y velocity	0.001
Z velocity	0.001
Energy	1e-06

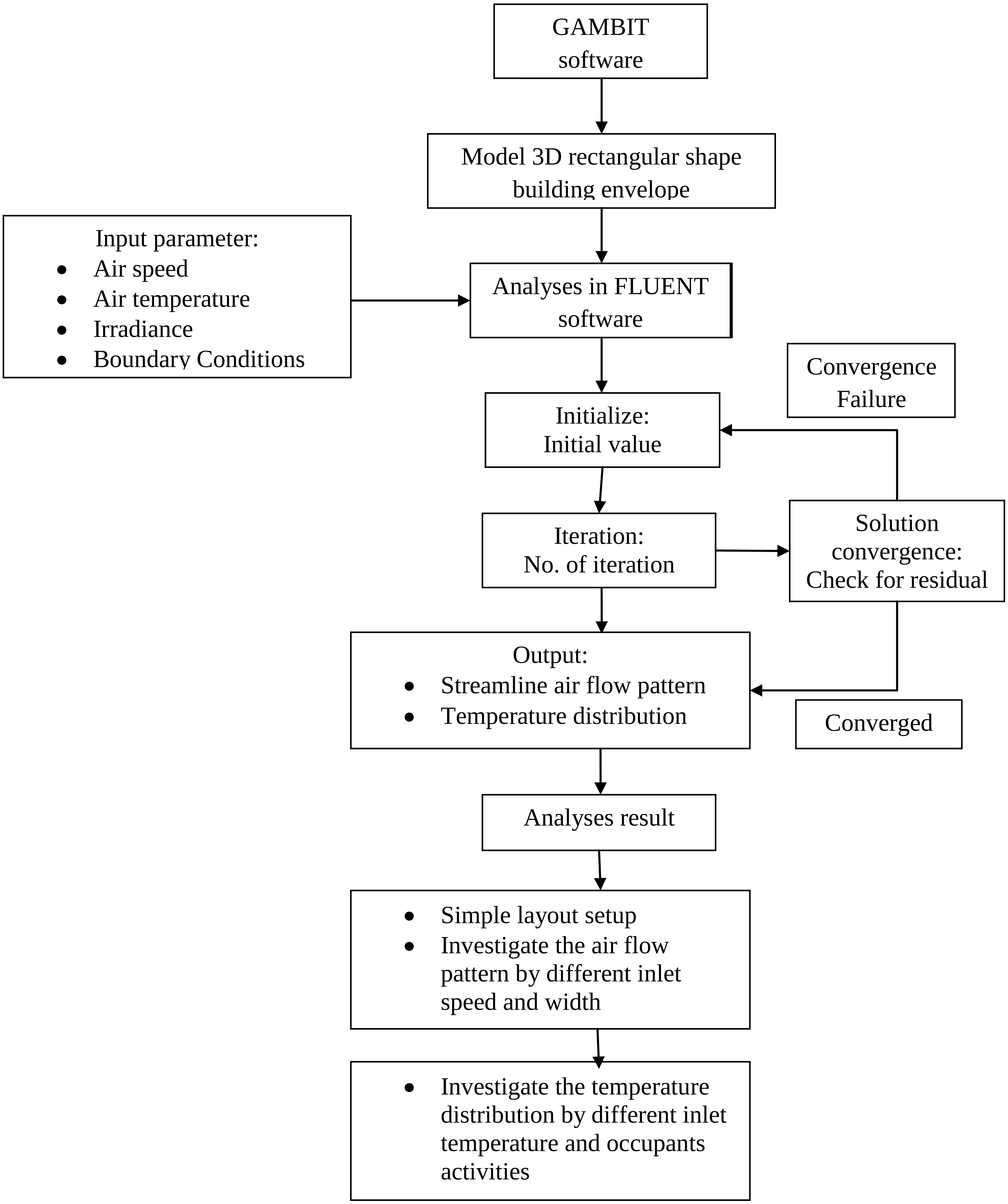


Figure 4: Flow chart of the CFD model analysis

CHAPTER 5

NUMERICAL SIMULATION

5. NUMERICAL SIMULATION

5.1. BUILDING ENVELOPE MODEL

The In this chapter, the mosque interior shown in Figure 4 was modeled by using GAMBIT 2.4.6. The mosque consists of two main praying areas, separating male and female. The male area encompasses the lower area which can be accessed using 9 main doors. The female area is situated upstairs which can be accessed by stairs. The size of the male area is 19.3 m length, 10 m height (excluding the dome) and 18.9 m width. The size of the female area is 24.3 m length, 3.75 m height and 13.4 m width. The ventilation system is a water cooler air conditioning type of ventilation, which directly supplied from Gas District Cooling UTP. Figure 5 showed the isometric view of the mosque.

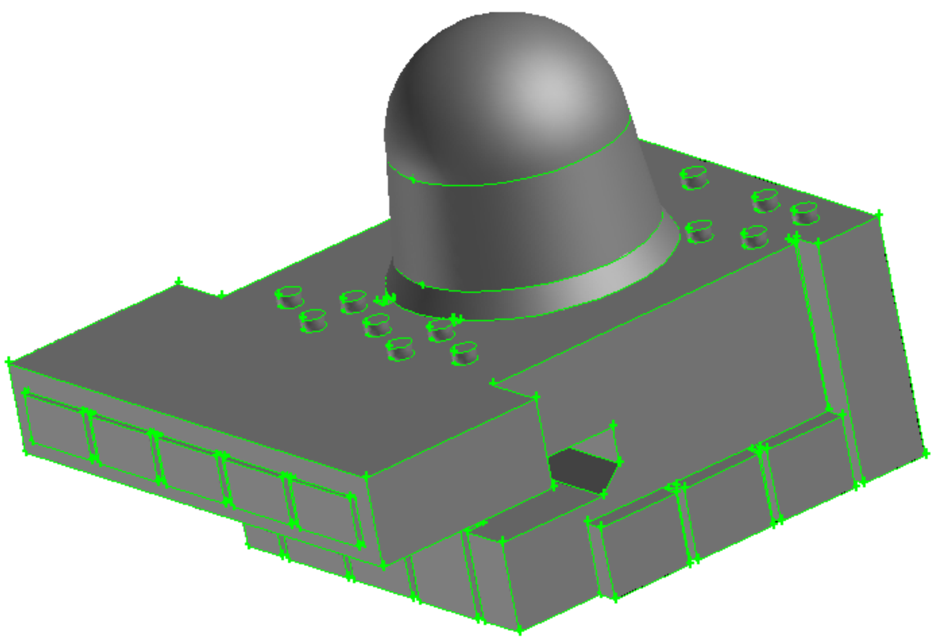
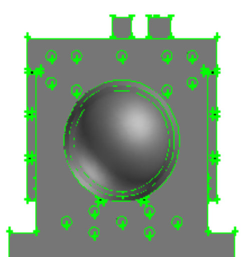


Figure 5: UTP mosque interior



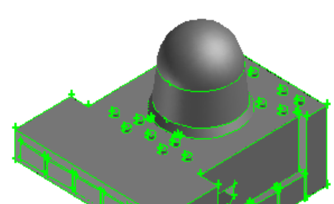


Figure 6: Isometric view of UTP mosque

The model of mosque was meshed using GAMBIT. The size function was applied with the bottommost of the mosque as the source. Two different viscous models was examined and compared, which is the Standard k – ε and Realizable k – ε turbulence model.

5.2. HUMAN PARAMETERS

The method for estimating personal parameter was based on AHSRAE Standard 55 [7]. The clothing and metabolic rate were estimated based on the observation made during a survey. It was observed that all occupants clothing were short sleeves shirt or baju melayu or T shirt and jeans/straight trousers. The clothing that the occupants wore could be categorized as light summer clothing (0.5 clo).

Metabolic rate was estimated to be 1.2 met (70 W/m^2), which likely to correspond to the sedentary activities inside the envelope. The activities of the occupants were praying, ranging from standing, bowing and sitting which requires them remain at fixed position.

5.3. DISCRETIZATION

The mesh is designed using GAMBIT 2.4.6 and the discretization of the computational domain is achieved by means of an unstructured mesh. In the boundary layer next to a nonslip wall especially the focused area (floor area), there are high gradients within a small region, so to capture these gradients accurately it was necessary to have fine mesh spacing normal to the wall.

The volume mesh process consisted of several steps, including applying size function, surface improvement, subsurface generation and then the actual interior meshing. The grid contains tetrahedral elements, the final mesh composed of ---- cells. A grid dependency analysis could be conducted to ensure that the resolution of the mesh was not influencing the result.

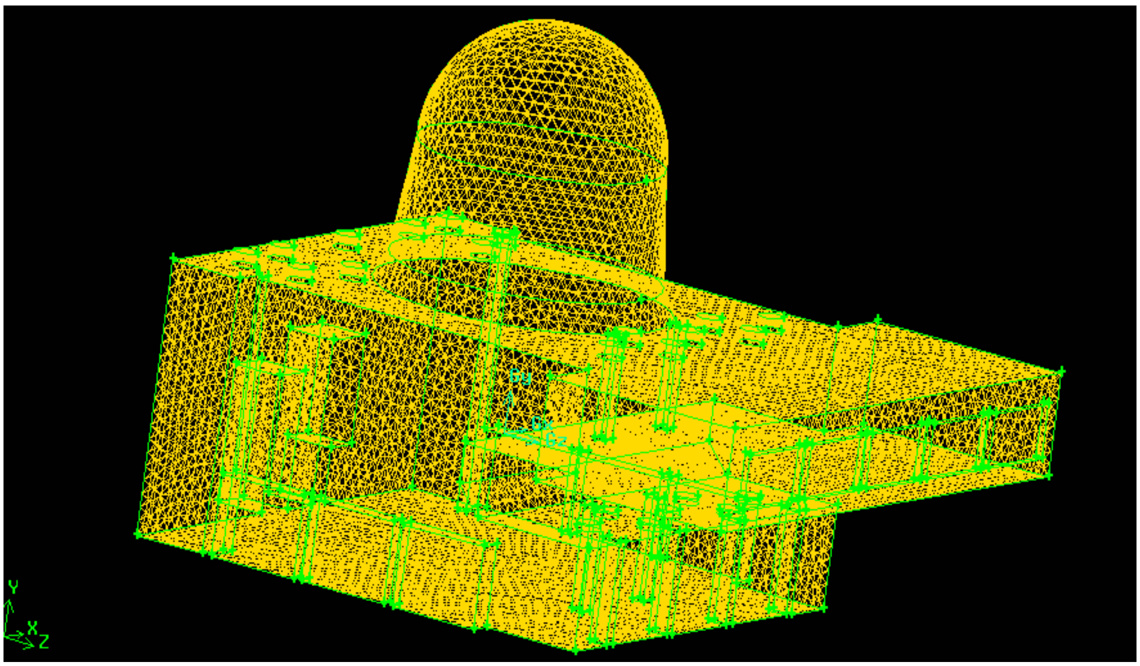


Figure 7: UTP mosque discretization

5.4. CFD SIMULATION

The thermal boundary conditions on the doors were taken to be heat flux due to radiation with heat flux ranging from 10.105 W/m^2 to 0.91W/m^2 for 9 doors. Air velocities and temperature at 10 m from floor were also taken to be 11 m/s and 22.1^{o}C respectively. At the velocity inlets the turbulence specification method was applied to intensity and hydraulic diameter and 10% assume the air is coming with some turbulence from the ducting. Streamline of air velocities and temperature distribution were examined to investigate the thermal condition inside the envelope.

The Realizable k – ε model was used as it will provide more accurate prediction as there is lots of detail missing in the standard k – ε model. The realizable model satisfies certain mathematical constraints on the normal stresses, consistent with the physics of turbulence flow. Figure 8 displays the velocity vector for chosen cross sectional area. This sectional area represents the most densely occupied by people, which is directly under the velocity inlet, -8.118 m away from the origin point.

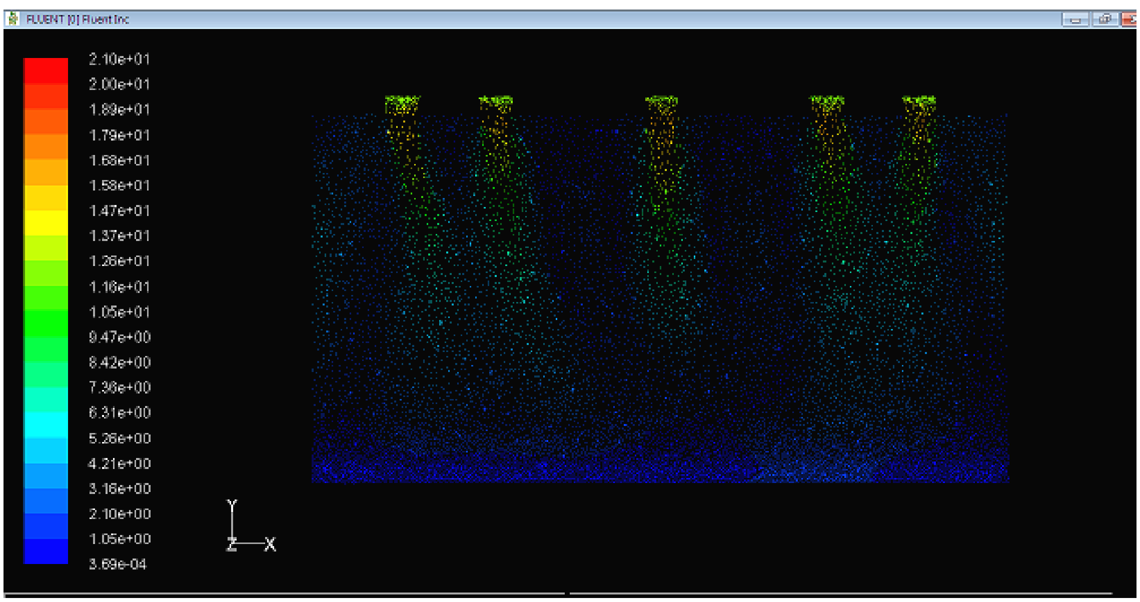


Figure 8: Vector of velocity magnitude of cross sectional area of building envelope

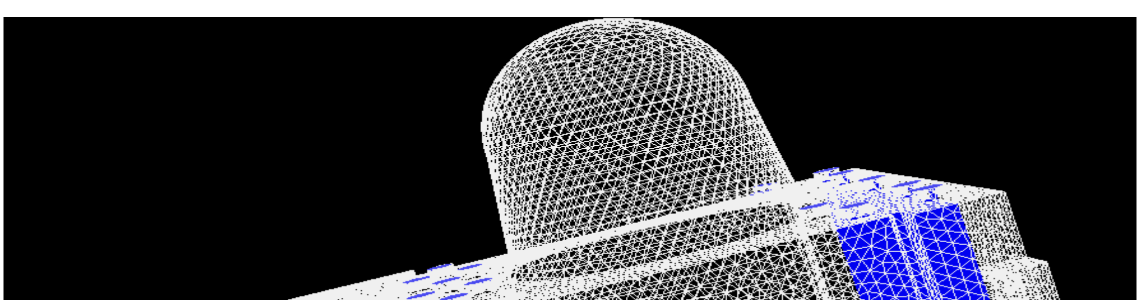


Figure 9: The cross sectional area region

Important objective of the air conditioning system is to create a comfortable thermal environment with proper combination of comfort variables such as air velocity and temperature. As what figure 8 shows, the circulation of air is nearly small as such no drastic gradient of velocity happen inside the envelope. The maximum velocity of the air is found to be 15.73m/s, while the minimum velocity is found to be around 0.03 m/s which are nearly equal to 0. Mean air speed should be less than 0.15 m/s in occupied zone to prevent a feeling of discomfort and sense of draughtiness.

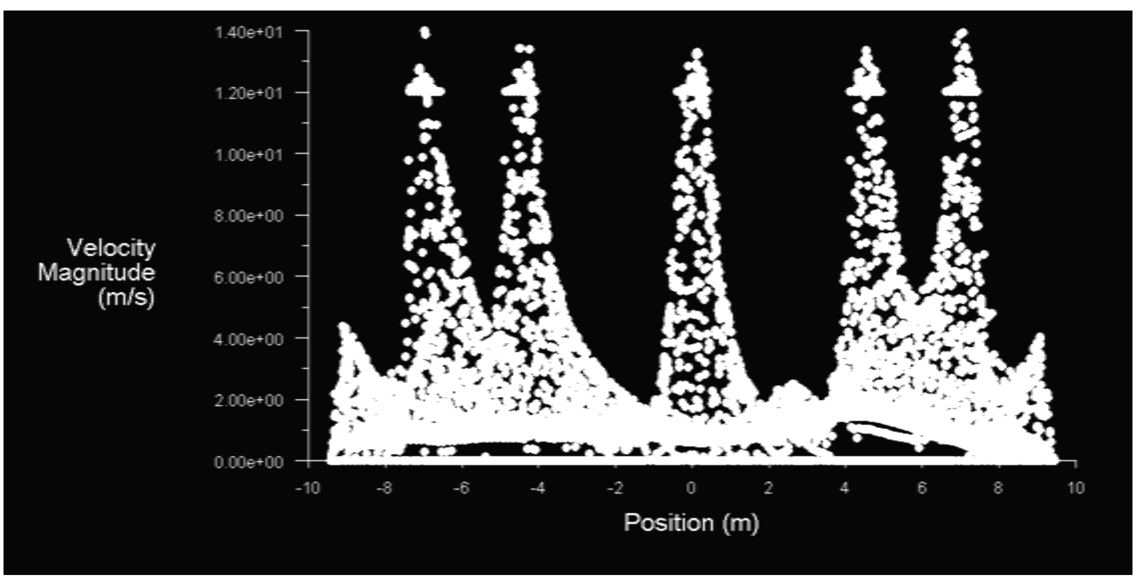


Figure 10: Plot X-Y graph of velocity magnitude

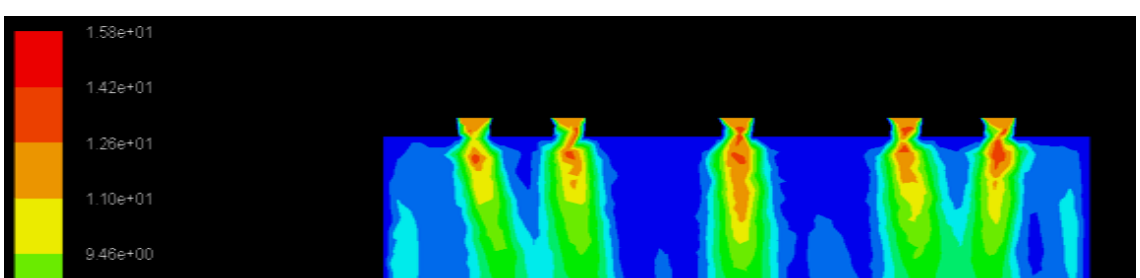


Figure 11: Contours of velocity magnitude

For the same layout, it can be seen that the temperature varied from 22.1^{o}C to 25.55^{o}C. The reason for that seems to be due to little mixing of entering air with the existing air. The increase in temperature is probably due to irradiance effect, which is supplied by the nearby door (heat flux source). Overall, the range of this area is desirable as it satisfies the AHSRAE comfort condition.

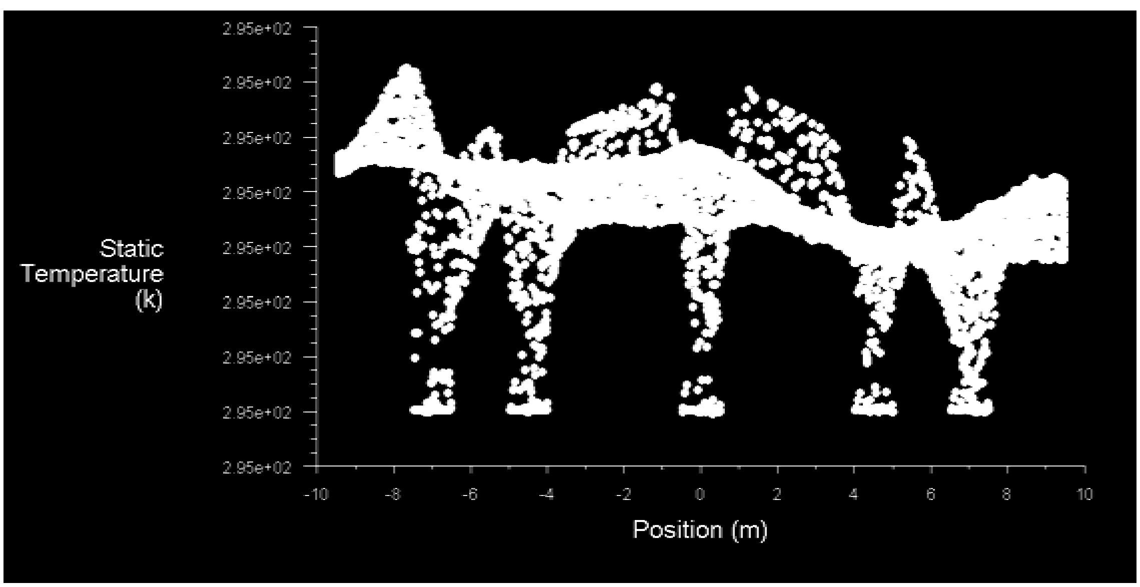


Figure 12: Plot X-Y graph of static temperature

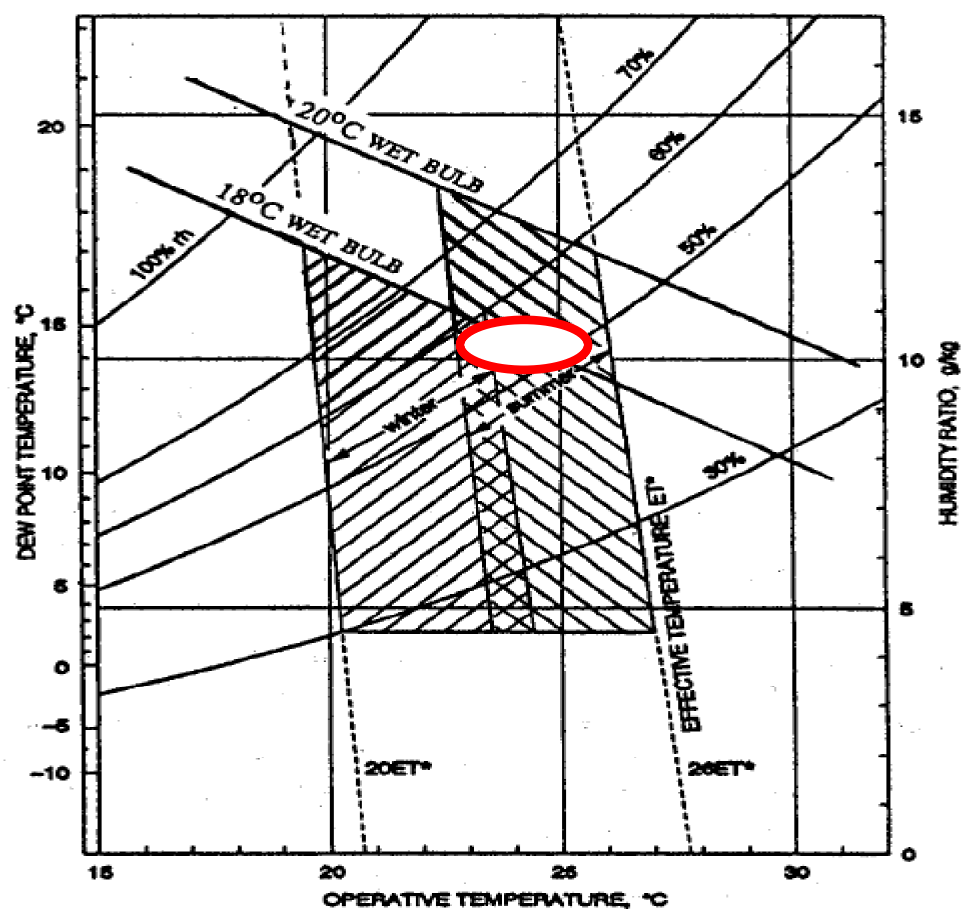
CHAPTER 6

CONCLUSION AND RECOMMENDATION

6. CONCLUSION

6.1. AHSRAE RECOMMENDATION

CFD is used to assess the effectiveness of the air conditioning inside the envelope. Using the observation for occupants clothing and activity, the CFD result are found to be in good agreement with the ASHRAE Standard recommendation. It is found that the temperature range is 22.1 to 25.55^{o}C, which is fall under summer condition acceptable range.



6.2. RECOMMENDATION

The scope of study could be widened if given ample of time to the extent of infiltration situation where some of the door is opened, permitting outside air to enter the envelope and vice versa. Perhaps, the study could also consider the air change per hour to get more realistic result.

The study could also attempt to improve the model by implementing grid dependency analysis, which will make the result more accurate and free from variation due to mesh size. The study should refine the mesh further to get the optimum outcome expected from CFD simulation.

REFERENCES

The format of references for the respective sources is as follows:

1. Journal refers to Hazim (2003)
2. Standards refers to ASHRAE Standard 55

(1) Houghten,F.C., Yaglou,C.P., 1923, *Determining lines of Equal Comfort and Determination of Comfort Zone*, ASHRAE Transaction, Vol 29, Page 361

(2) Yaglou,C.P., Drinker,1929, *The Summer Comfort Zone Climate and Clothing*, ASHRAE Transaction, Vol 3, Page 269

(3) Yaglou, C.P., Millness,W.E, 1925, *Effective Temperature With Clothing*, ASHVE Transaction, Vol 31, Page 89

(4) Arens e., Bauman F., Huizenga C., 2002, *ASHRAE Investigation of Thermal Comfort at High Humidity*, Building Science at University of California at Berkeley

(5) ASHRAE 55-2004 ASHRAE Standard 55-2004 *Thermal Environmental Conditions for Human Occupancy.*