

# CHAPTER 1

## INTRODUCTION

### 1.1 BACKGROUND OF STUDY

The Academic Complex of Universiti Teknologi Petronas (UTP) is a high technology building for educational institution. With such a great scale infrastructure, the air conditioning systems is one of the essential components of the building which should be carefully analyzed to ensure efficiency . The air conditioning system of the whole academic is designed to be a centralized system which can be divided into two separate systems, i.e upstream and downstream . The upstream consists of chilled water pipe loop from Gas District Cooling (GDC) to the main heat exchanger at the Chancellor Complex. The downstream system is the chilled water piping that circulates chilled water to the whole campus from the main heat exchanger. The chiled water is pumped to unit of Air Handling Unit (AHU) in the new academic building.

The air handling unit (AHU) is a device used to condition or circulate air as part of heating, ventilating and air-conditioning (HVAC) system. The AHU will supply cool air into the lecturer's office space base on demand. If the demand is higher, more cooled air will be supplied and same goes if lower demand then lesser supply. The demand for cooling air based on the cooling load. Building 13 is located at the outer section of Academic Complex. The left side of the building is facing east and the right side is facing west. This indicates that the left side of Building 13 will receive more sun radiation in the morning. The sun radiation generates heat via convection thus the cooling load is higher at the left side of building in the morning. The cooling load is higher on the right side in the evening. This indicates that the cooling load varies during day time thus variable rate of cooling air should be supplied by the AHU. One of the purpose of this study is to investigate by using the AHU response to the cooling load demand at block 13.

## **1.2 PROBLEM STATEMENT**

Cooling load for every building is different and the cooling load changes throughout the day. Different sources of cooling load give different value of cooling load for each building. The AHU supplying the cooled air is based on cooling load and user demand. The system must respond to the cooling load throughout the day.

## **1.3 OBJECTIVE**

The main objectives of this research are:

- To investigate how the system response by analyzing the air flow from the AHU to the space.
- To simulate the air flow profile in the ducting by using software

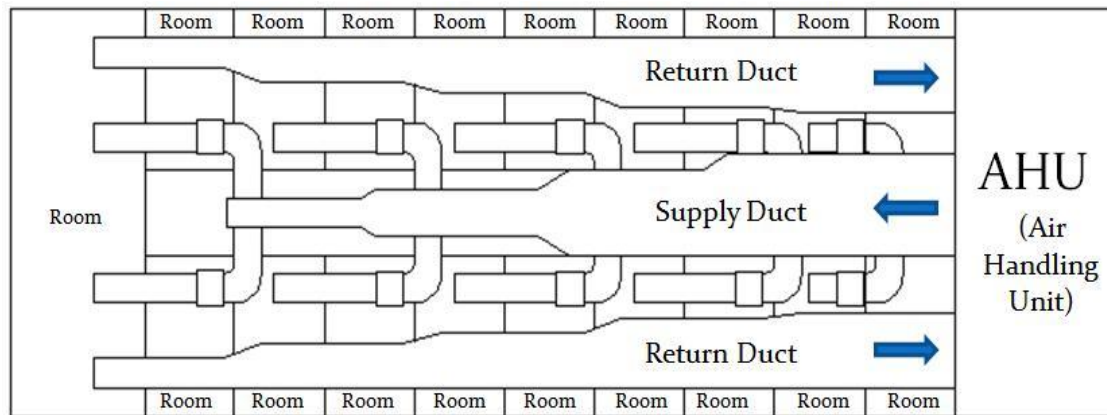
## **1.4 SCOPE OF STUDY**

The scopes of work for this project are divided into two parts. Part 1 which is done during the of first semester is to simulate the cooled air flow in the lecturer's office space by way of measuring the air flow rate from the diffuser and measure the temperature in strategic place in lecturer's office. Then the analysis of the air flow modeled using Fluent. This analysis shall be performed to consider the ventilation efficiency and ventilation effectiveness together with the air change rate. The scope of work of Part 2 is to investigate the changing of cooling load, perfome the measurement of air velocity and temperature with different situation in order to analyses the system responses with the cooling requirement. The simulation using FLUENT simulate the flow profile of the system.

## CHAPTER 2

### LITERATURE REVIEW

#### 2.1 INTRODUCTION



**Figure 1:** AHU and ducting layout in Block 13 at level 3

Air handling unit (AHU), is a device used to condition and circulate air as part of a heating, ventilating, and air-conditioning (HVAC) system. An air handler is usually a large metal box containing a blower, heating or cooling elements filter racks or chambers, and dampers. Air handlers generally connect to ductwork that distributes the cooled air through the building and returns it to the AHU. Sometimes AHU discharge and admit air directly to and from the space served without ductwork [1].

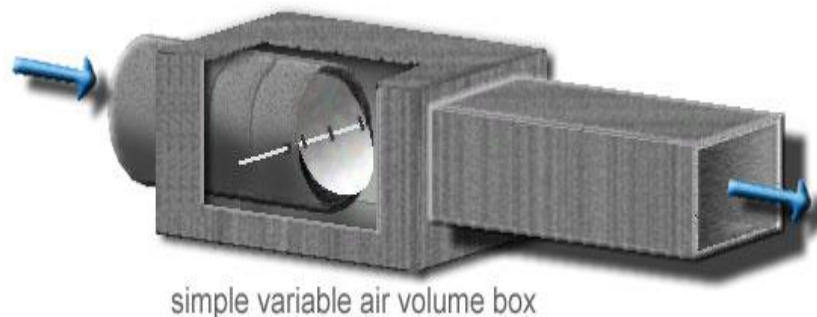
Small air handlers, for local use, are called terminal units, and may only include an air filter, coil, and blower; these simple terminal units are called blower coils or fan coil units [2].

Air handlers typically use a large squirrel cage blower driven by an AC induction electric motor to move the air. The blower may operate at a single speed, offer a variety

of set speeds, or be driven by a Variable Frequency Drive to allow a wide range of air flow rates. Flow rate may also be controlled by inlet vanes or outlet dampers on the fan [3].

Figure 1 show AHU unit the ducting layout in Block 13 at level 3. AHU unit at Block 13 use single blower and typically placed at the beginning of the supply ductwork. Chilled water will be pumped to the AHU from the Gas District Cooling (GDC) which is situated outside the UTP. Then the chilled water normally at  $11^{\circ}\text{C}$  flowed into the cooling coil inside the AHU. The blower will suck fresh air from the outside and blow the air passing through the cooling coil into the ducting. When fresh air ( $30^{\circ}\text{C}$ ) which is high temperature than the chilled water passing through the cooling coil, the heat from fresh air will lost to the chilled water, then the temperature of the fresh air drop around  $15^{\circ}\text{C}$  to  $19^{\circ}\text{C}$  then the fresh air became cooled. This cooled air will flow through supply duct into each rooms . Rate of cooled air coming into the room controlled by controlling device called Variable Air Volume (VAV). Then the cooled air flowing back into the AHU for recirculation through supply duct [3].

## 2.2 DUCTING SYSTEM IN BLOCK 13



**Figure 2:** Variable Air Volume

Ducting system in block 13 is rectangular in shape and at the end of it there are Variable Air Volume (VAV) boxes. Figure 2 show the VAV which is used in the ducting system in Block 13. This VAV distributed the cooled air into two or three rooms. This information was collected during our first meeting with the UTP maintenance engineer, Mr. Hatta and Mr Syed. Two or three rooms share the same VAV. VAV boxes provide

constant or variable air depending on the temperature demands of the space. As the temperature rises the VAV damper opens to send a designed amount of airflow to the room [3].

The VAV boxes can also be classified as pressure independent and pressure dependent. A pressure independent VAV measures cubic feet per meter (cfm) and will maintain the proper airflow regardless of the box inlet static pressure provided by the main air handling unit. A pressure dependent VAV does not measure airflow . But here in UTP we are using pressure independent type which is measure airflow. The cfm will change depending on the inlet static pressure. The inlet static pressure measured inside the ducting. VAV systems are also usually designed with a diversity factor which means that the main air handler design airflow is less than the sum of the total airflow of all the VAV's. This is a common design because not all of the VAV's in a building will be in full cooling or maximum cfm all at once. The VAV damper is opened and closed by a controller sending air pressure to an actuator hooked to the VAV damper [4].

Currently the cooled air from the ducting system flow into the room through fixed diffuser which means no controlling method implemented on the diffuser. The person inside each room can only control the VAV which is shared among three or two rooms.

## **2.3 COMPUTATIONAL FLUID DYNAMICS**

CFD or computational fluid dynamics is predicting what will happen, quantitatively, when fluids flow, often with the complications of simultaneous flow of heat, mass transfer, phase change, mechanical movement, stresses in and displacement of immersed or surrounding solids. Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. Even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. Ongoing research, however, may yield software that improves the accuracy and speed of complex

simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test [4].

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid and Navier- Stokes equations for viscous flow). In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D, or pyramidal solids in 3D) or regular; the distinguishing characteristic of the former is that each cell must be stored separately in memory. Where shocks or discontinuities are present, high resolution schemes such as Total Variation Diminishing (TVD), Flux Corrected Transport (FCT), Essentially Non Oscillatory (ENO), or MUSCL schemes are needed to avoid spurious oscillations (Gibbs phenomenon) in the solution. If one chooses not to proceed with a mesh-based method, a number of alternatives exist, notably Smoothed particle hydrodynamics (SPH), a Lagrangian method of solving fluid problems, Spectral methods, a technique where the equations are projected onto basis functions like the spherical harmonics and Chebyshev polynomials, Lattice Boltzmann methods (LBM), which simulate an equivalent mesoscopic system on a Cartesian grid, instead of solving the macroscopic system . It is possible to directly solve the Navier-Stokes equations for laminar flows and for turbulent flows when all of the relevant length scales can be resolved by the grid (a direct numerical simulation) [6].

In general however, the range of length scales appropriate to the problem is larger than even today's massively parallel computers can model. In these cases, turbulent flow simulations require the introduction of a turbulence model. Large eddy simulations (LES) and the Reynolds-averaged Navier-Stokes equations (RANS) formulation, with the  $k$ - $\epsilon$  model or the Reynolds stress model, are two techniques for dealing with these scales. In many instances, other equations are solved simultaneously with the Navier-

Stokes equations. These other equations can include those describing species concentration (mass transfer), chemical reactions, heat transfer, etc [8] .

## 2.4 COOLING LOAD

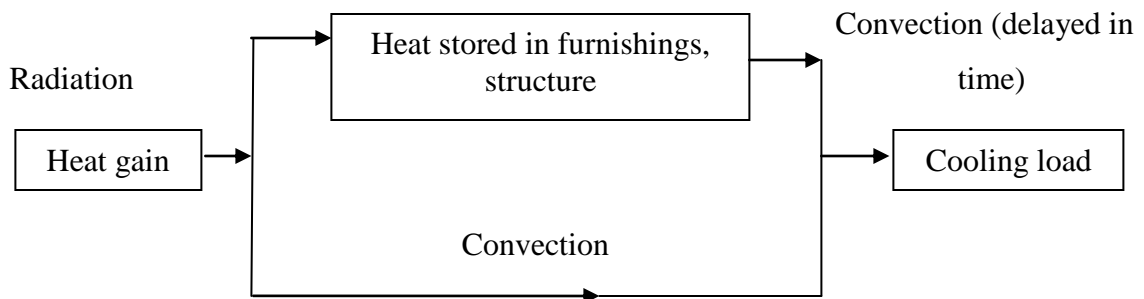
Cooling load is the total amount of heat energy that must be removed from a system by a cooling mechanism in a unit time, equal to rate at which heat is generated by people , machinery and processes, plus the net flow of heat into the system not associated with the cooling machinery [2].

The load analysis is one of the crucial steps in designing of an air conditioning system. There are 2 distinctly different components of air condition load:

1. the sensible load
2. the latent load

Sensible cooling load is a load from a heat gain while latent cooling load is a load that created by moisture in the air, including from outside air. The cooling load is very important to my study because the cooling load effect the fluid dynamic of the system.

The amount of heat that must be removed (the cooling load) is not constantly equal to the amount of heat received at a specific time. This distinction is a result of the heat storage and time lag effects. The thermal storage effect and resulting time lag cause the cooling load to often be different in value from entering heat(called the instantaneous heat gain).



**Figure 3:** Heat flow diagram showing building heat gain, heat storage and cooling load

Figure 3 show that heat gain from sun radiation, some of the heat stored in furnishings, structure and ect. Some of the heat heated the space via convection and rise the

temperature. The heat stored in structure or furnishing will heat the space but delayed in time. The amount of these heat which has to be removed is called cooling load.

Internal heat gains such as people, light and equipment are often a major component of the cooling load in commercial and institutional buildings. In fact, for many large office buildings, the internal heat gains are dominant source of cooling load [7].The heat gain from people has two components : sensible and latent. The total and the proportions of sensible and latent heat vary depending on the level of activity [12].Since lighting is often the major internal load component, an accurate estimate of the space heat gain it imposes is needed. The rate of heat gain at any given moment can be quite different from the heat equivalent of power supplied instantaneously to those lights [12]. Some of the energy emitted by the lights is in the form of radiation that is absorbed by the building and contents. The absorbed energy is later transformed to the air via convection.

## **2.5 DUCTING SYSTEM**

From the review of a research project by Shepherd [10] about the ducting system, the following information about various types of ducting and their performances was obtained.

The efficiency of air distribution systems has been found to be 60-75% or less in many office space because of insufficient and/or poorly installed duct insulation and leaks in the duct system. Properly designed and installed duct systems can have efficiencies of 80% or more for little or no additional cost. Moreover, efficient duct system designs can reduce equipment size, further saving money for new or replacement equipment [10].

Duct systems that leak and/or do not distribute air properly throughout the office space may make some rooms too hot and others too cold. This is cause the uneven air distribution in the office space. Leaky and unbalanced duct systems may force conditioned air outside and unconditioned air into the building. This increases heating

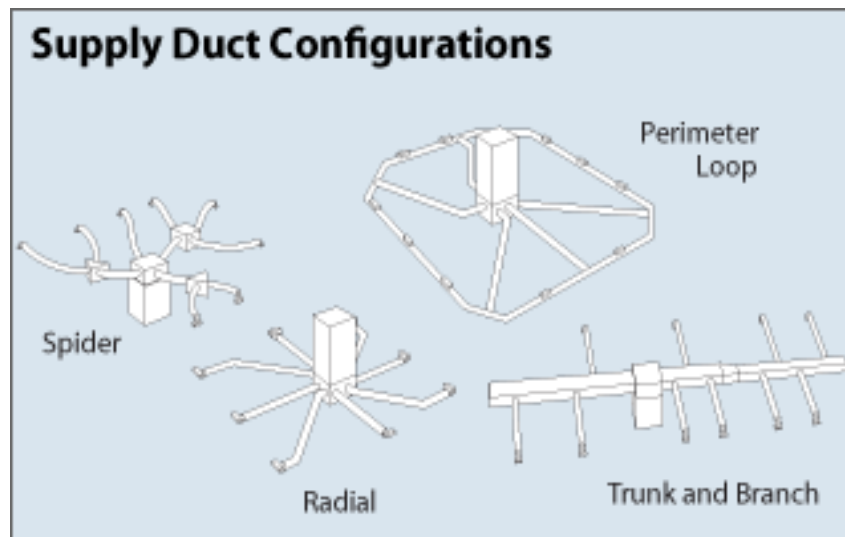


and cooling costs and may also draw humidity, dust, mold spores, and other contaminants into a building from the outside.

Duct systems that are undersized, are pinched, or have numerous bends and turns may lead to low air flow rates and high air velocities. Low air flow rates cause the heating and cooling equipment to operate inefficiently [10].

Figure 4 shows the common types of duct systems implemented in current office building [5].

- trunk and brunch system
- spider system
- radial system
- parameter loop system

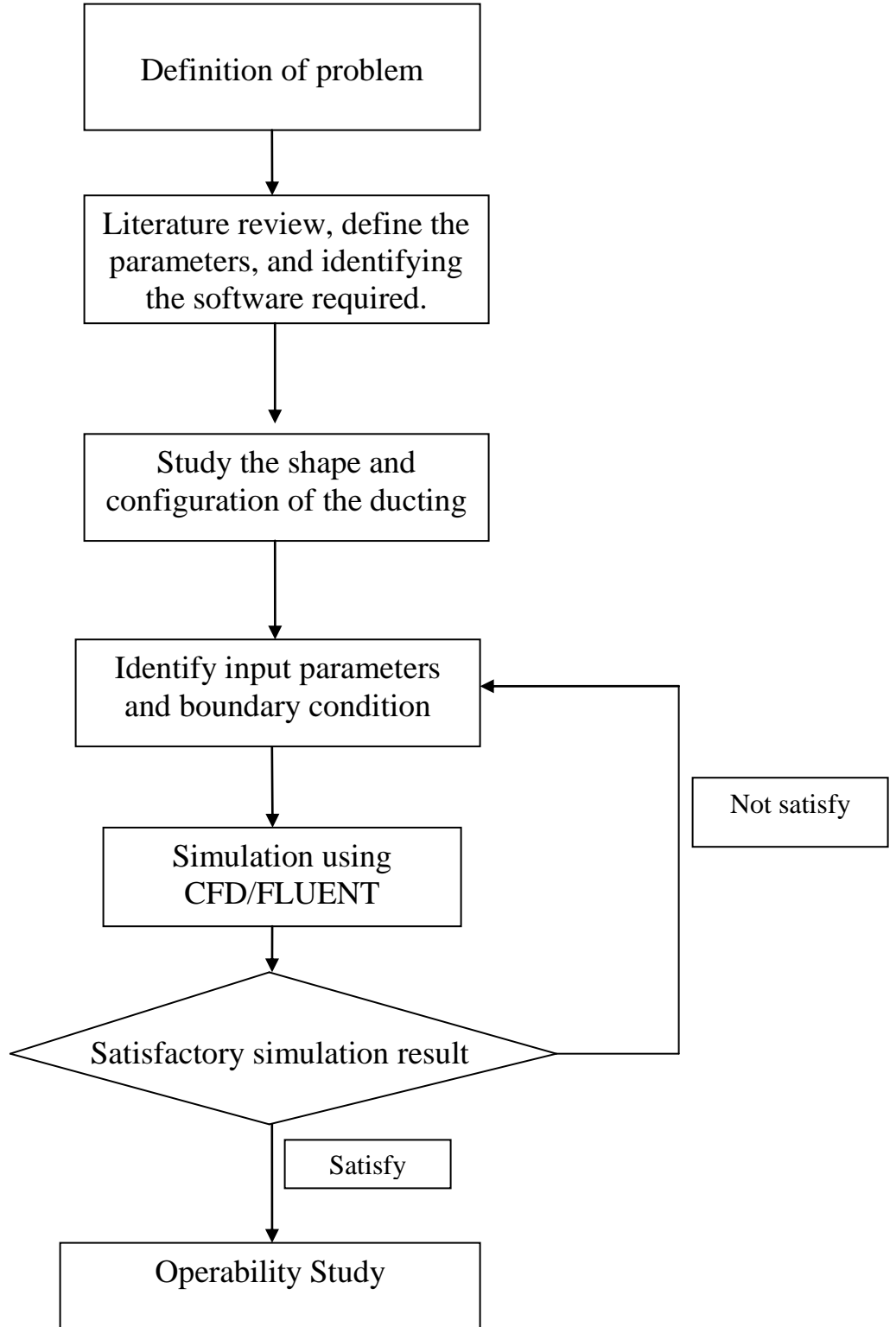


**Figure 4:** Types of Ducting [5]

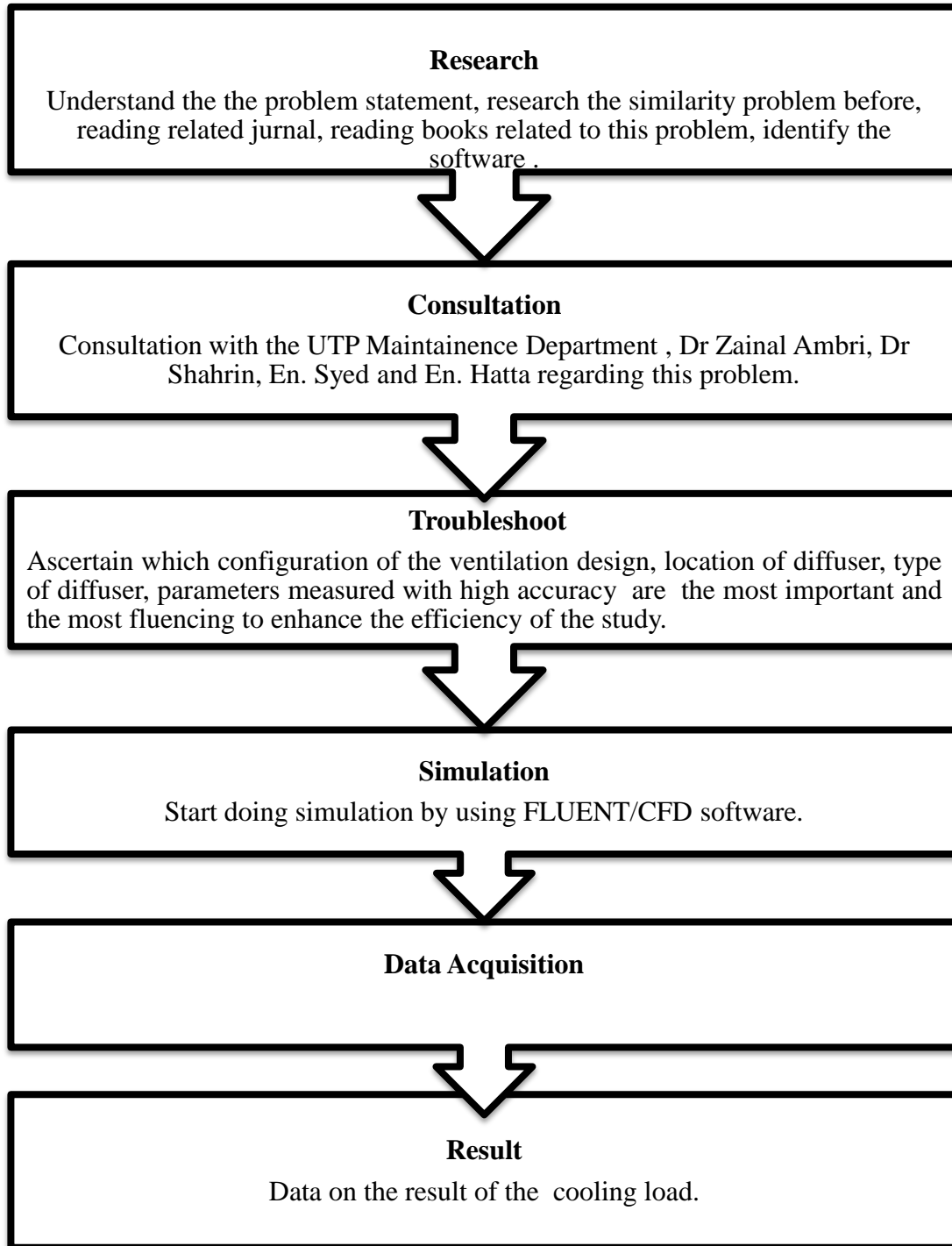
# CHAPTER 3

## METHODOLOGY

### 3.1 Project Execution

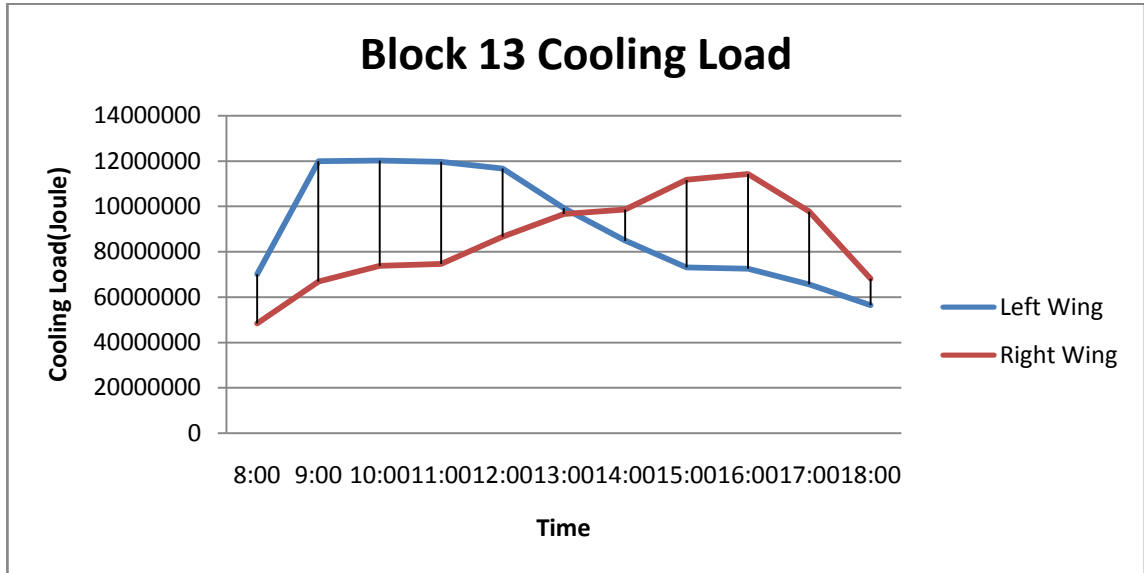


### 3.2 Project Activities



### 3.3 Research Methodology

#### 3.3.1 Introduction



**Figure 5:** Variation of cooling load of Building 13 of UTP's Academic Building

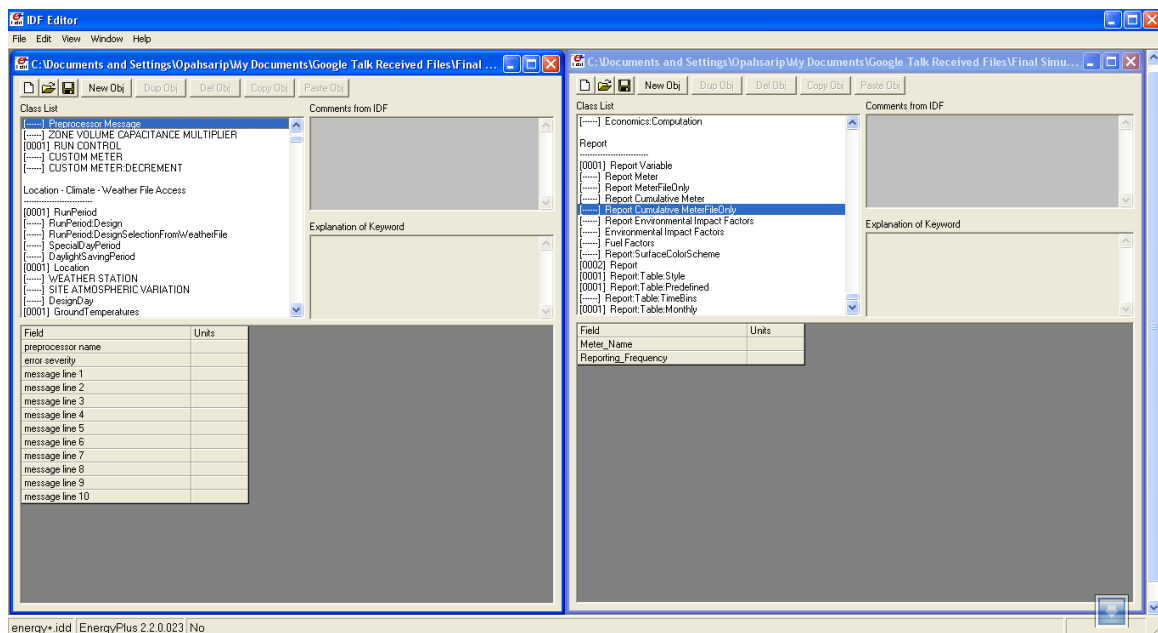
Based on the report of a previous FYP project by Siti Nurfadhilah [11] of Block 13, cooling load was to have a great different between the left and right wing in the morning and evening. The cooling load at left wing was very high in the morning but drop dramatically in the evening. Meanwhile for the right wing it's show the cooling load was very low in the morning but raise dramatically in the evening. This can be related to the correspond velocity of cooled air supply by the AHU. The cooling system for the building should respond accordingly to the cooling load. In order to as certain this activities, temperature and air flow rate into the room need to be recorded. The current study is mainly focus at level 3 in Block 13.

Figure 5 shows variable of cooling load at Block 13 after simulation of the cooling load was done by the previous study. The differences of the cooling load between the two wing due the many factors. The main factors are internal load and external load. The internal load such as lightning, heat from human body and external load such as sun radiation [11].

### 3.3.2 Design parameters

Parameters were collected such as lecturer's room dimension, size of ducting, temperature and velocity of air coming out from diffuser. All these parameters are needed as an input to stimulate the air flow distribution in FLUENT. Temperature and outlet velocity from diffuser must be recorded precisely and in different time. The temperature and velocity recorded using device called anemometer and recorded in morning (10.00 am- 11.00 am) and evening (2.30pm-5.00pm). The design of the ducting system at 3<sup>rd</sup> level in block 13 was designed in Gambit before being transferred into Fluent. All the parameter needed for designing the ducting must be analyse from the blueprint of Air Conditioning and Ventilation Services, Package 3B1, Building 13, 3<sup>rd</sup> Floor Plan. The scale of the drawing is 1:150 mm.

### 3.4 Software for Cooling Load Calculation



**Figure 6:** Energy Plus software

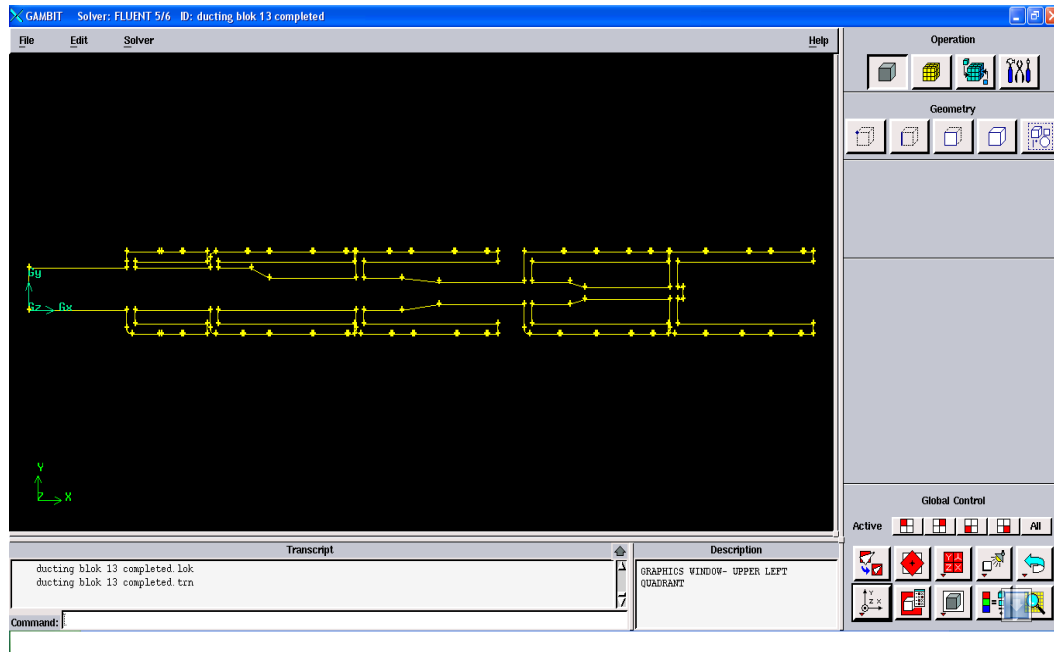
Energy Plus is the official building simulation program by United States Department of Energy. EnergyPlus is originally based on the previously popular energy simulation software BLAST (Building Load Analysis and Systems Thermodynamics)

and DOE-2. The software is created in order to model heating, cooling, heating, ventilating and other energy related systems of a building. Based on a user's description of a building from the perspective of the building's physical make-up, associated mechanical systems, etc., Energy Plus will calculate the heating and cooling loads necessary to maintain thermal control set points, conditions throughout secondary HVAC system and coil loads, and the energy consumption of primary plant equipment as well as many other simulation details that are necessary to verify that the simulation is performing as the actual building would.

The simulation program based on the heat balance method which allows for simultaneously calculation of radiant and convective effect at both interior and exterior surface during each time step. The heat balance method takes into account all heat balances on outdoor and indoor surfaces and transient heat conduction through building construction.

Data such as internal heat gain from lightings, ventilation , infiltration, electrical equipment as well as from the occupants are defined in the software. Usage scheduling can also be incorporated for all internal gains. Finally, user should define the kind of output that is desired as the simulation report . The report comes in two ways whether in Microsoft Excel format or HTML format for ease of viewing.

### 3.5 Drawing of the main ducting

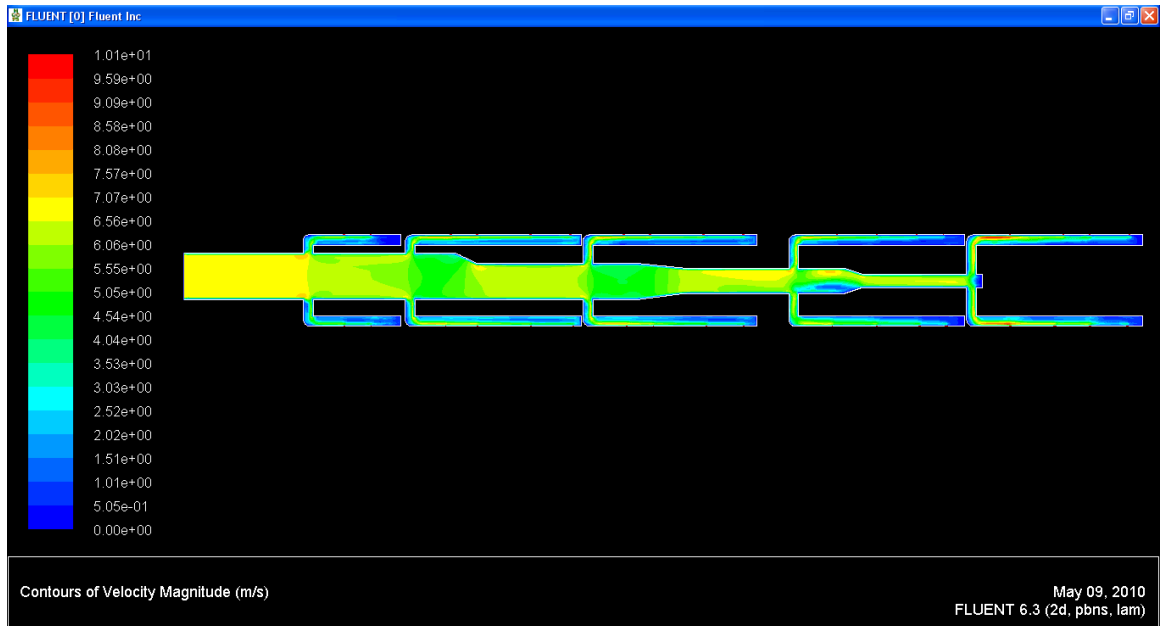


**Figure 7:** Designing the ducting in GAMBIT

The model development is carried out on GAMBIT. It is the pre processing process where the model of ducting in Block 13 development is performed and meshing of model is followed for further analysis. GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. 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 simple and intuitive, yet it is versatile enough to accommodate a wide range of modeling applications.

The model of the ducting was designed in 2 dimension. The profiles are generated with the help of coordinates available which have been generated. The assumption when modeling the ducting is there are no frictions at the surface of the ducting. The output of this modeling is grid generated

### 3.6 Simulation using Fluent



**Figure 8:** Simulation using FLUENT

FLUENT is the software used for modeling fluid flow and heat transfer in complex geometries. It provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease. It is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The basic procedural steps used for the model geometry solving a problem in FLUENT of Block 13 include:

- 1) Define the modeling goals.
- 2) Create the model geometry and grid.
- 3) Set up the solver and physical models.
- 4) Compute and monitor the solution.
- 5) Examine and save the results
- 6) Consider revisions to the numerical or physical model parameters, if necessary



### 3.6.1 Analysis

The analysis is carried in FLUENT by importing the meshed file saved in gambit. The steps that are followed are given below which include all the conditions and the boundaries values for the problem statement.

#### 3.6.1.1 Checking of mesh and Scaling:

The fluent solver is opened where 2D is selected and then meshed file from model generated using gambit is importing. The meshed file undergoes a checking where 62452 number of grids are found. After this grid check is done following which smoothing and swapping of grid is done. For the skewness method is selected and minimum skewness is set as 0.8 and the iterations are set as 8. Following this the scaling is done. Scale is scaled to mm. Grid created was changed to mm. After this defining of various parameters are done.

#### 3.6.1.2 Solver and Material Selection and Operating Condition Defining:

The solver is defined first. Solver is taken as pressure based and formulation as implicit, space as 2D and time as steady. Velocity formulation as absolute and gradient options as Green-Gauss Cell based are taken. Energy equation is taken into consideration. The viscous medium is also taken. First the analysis is carried using laminar flow.

The selection of material is done. Material selected is air. The properties of air is taken as follows:

$$\text{Density} = 1.229 \text{ kg/m}^3$$

$$\text{Cp (specific heat capacity)} = 1020 \text{ J/kg K}$$

$$\text{Viscosity} = 1.73 \times 10^{-5} \text{ kg/m s}$$

The analysis is carried out under operating conditions of 101325 Pascal. Gravity is not taken into consideration.

### 3.6.1.3 Boundary Conditions:

#### Nozzle Inlet:

Velocity flow inlet was taken for the nozzle inlet and the value of velocity flow was taken as 6.933 m/s. The velocity inlet was stated in the design plan of the AHU. Initial gauge pressure was taken as 250000 Pascal.

#### Outlet:

The diffuser was set as pressure outlet and the pressure was set to 101325 Pascal.

#### Initialization:

Solution initialization is done. Initial values of velocity are taken as zero along x, y and z direction. Temperature is taken as 300K.

Residual Monitorization is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity- 0.001

X-Velocity- 0.001

Y-Velocity- 0.001

Z-Velocity- 0.001

Energy- 1e-06

The number of iterations is then set up and iterations starts. The iteration continues till the convergence is reached.

### 3.7 Measurements of airflow.

An anemometer was used to measure the air velocity in one point at a time. Mapping this quantity this requires multiple measurements at point distributed in a two or three dimension grid.



**Figure 9:** Anemometer

The measurement is taken during day time which is at morning between 9.30 am till 11.30 am and evening at 2.3 pm till 5.30 pm. This measurement is repeated 3 times in 3 different days. The measurement taken at the diffuser in each room. When the measurement is taken the door of the room is closed, light is on, computer is on and there is occupant in the room which is me and the lecturer.

### 3.8 Gantt Chart and Key Milestone

NO.	DETAILS/WEEK	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Selection of project topic	■	■												
	-Confirmation of project topic														
2	Preliminary of research works			■	■	■									
	- Data collection														
	- Identifying software and tools required														
	- Literature review on ducting and HVAC systems			■	■	■	■	■	■	■	■	■	■	■	
3	Submission of Preliminary Report						■								
4	Data reduction and presentation							■	■						
5	Submission of Progress Report									■					
	-Literature Review														
	-Project Description														
											■	■			
6	Project work continue														
	-Analysis on original flow of air distribution in office space														
7	Submission of Interim Report Final Draft											■			
8	Oral Presentation												■		
9	Submission of Interim Report														■

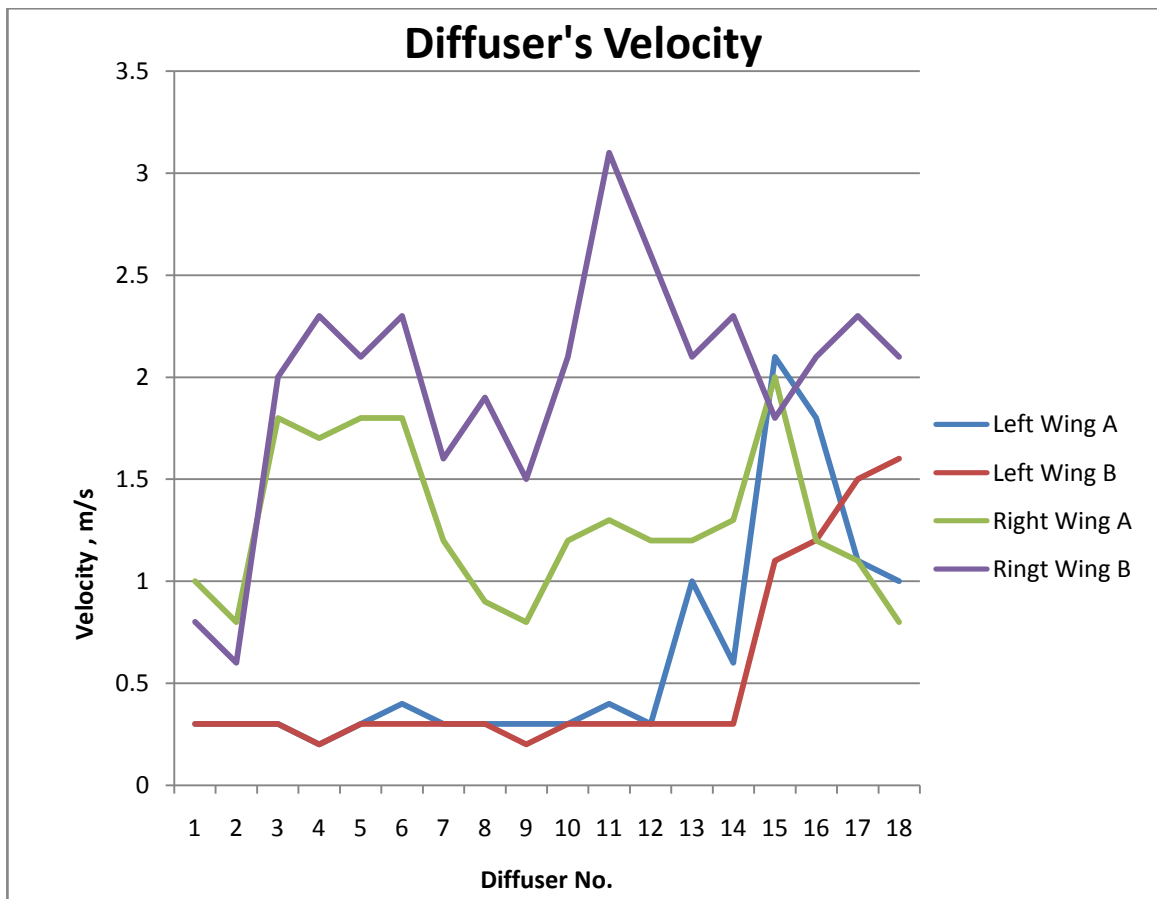
Figure 10: Gantt chart

## CHAPTER 4

### RESULT AND DISCUSSIONS

#### 4.1 Average Velocity at Left Wing and Right Wing at Block 13

Velocity measurement was taken during the period of 2.30 pm – 5.30 pm of observe the AHU respond when the cooling load during this period is high. Figure 11 shown the air velocity at each diffusers between 2.30 pm and 5.30 pm .



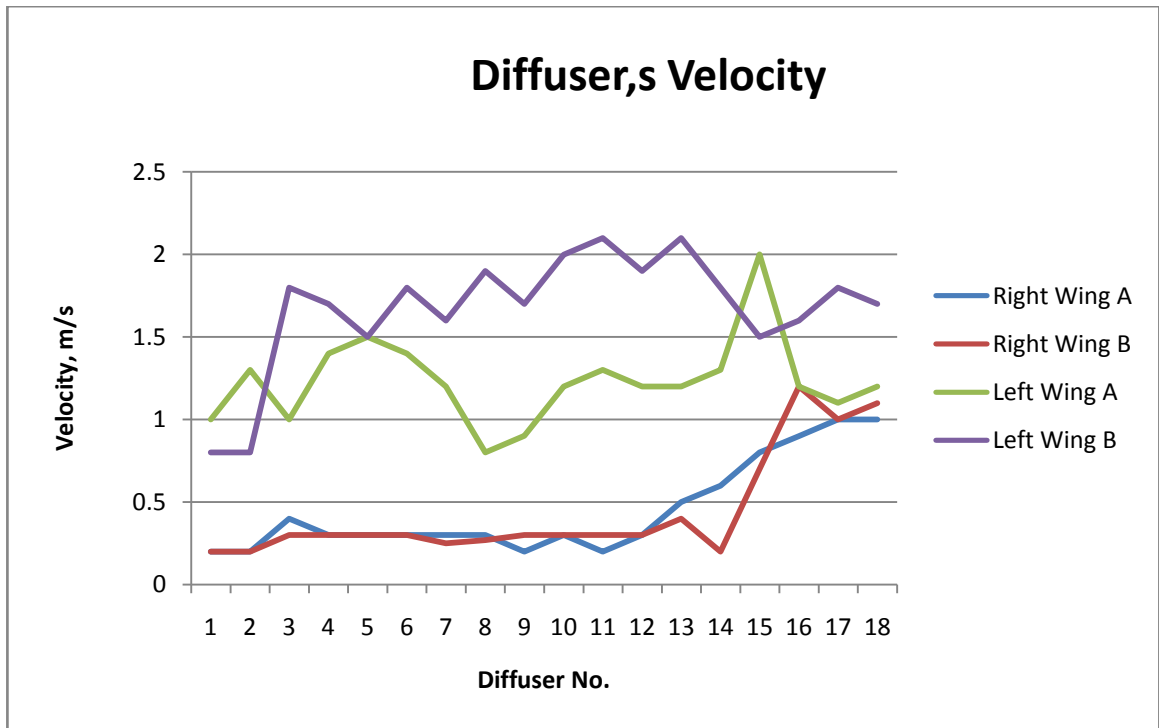
**Figure 11:** Variation of velocity coming out from the diffuser on the Left Wing and Right Wing in evening (2.30pm – 5.30pm).

This data were collected in the afternoon at block 13 between 2.30 pm – 5.30 pm on the right wing and left wing. It's show that the average velocity coming out from the diffusers at the right wing is higher than the left wing. The average velocity of the right

wing is 2.32 m/s meanwhile the average velocity for left wing is 0.58 m/s. From this measurement we can see the result is parallel with the previous study regarding the cooling load in Block 13.

**4.2 Average Velocity at Left Wing and Right Wing at Block 13 in the morning ( 9.00 am – 11.30 am ).**

Velocity measurement was taken during the period of 9.30 am – 11.30 am of observe the AHU respond when the cooling load during this period is high at left wing and low at right wing. Figure 12 shown the air velocity at each diffusers between 9.30 am and 11.30 am .



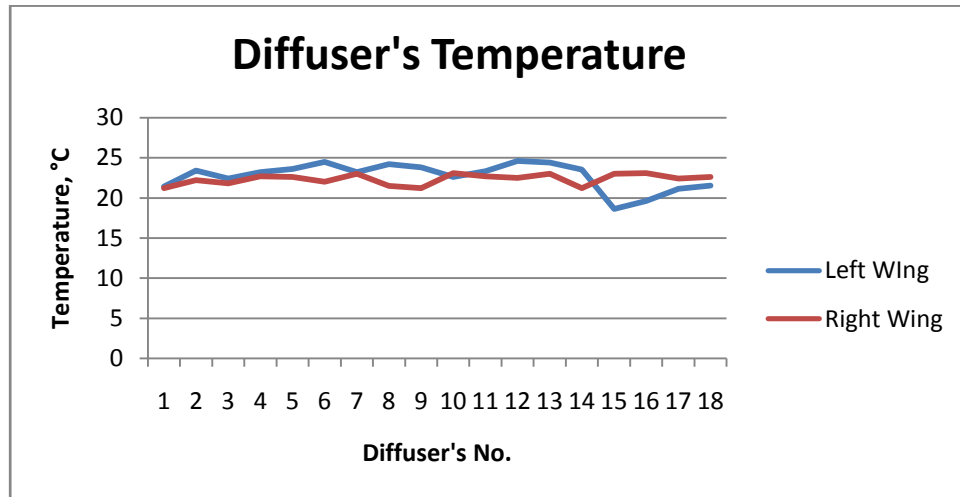
**Figure 12 :** Variation of velocity coming out from the diffuser on the Left Wing and Right Wing in morning ( 9.30am – 11.30am).

This data were collected in the morning at Block 13 between 9am - 11am on the right wing and left wing. Figure 12 show that the average velocity coming out from the diffusers at the left wing is higher than the left wing. The average velocity of the left wing is 1.31 m/s meanwhile the average velocity for right wing is 0.43 m/s. From this

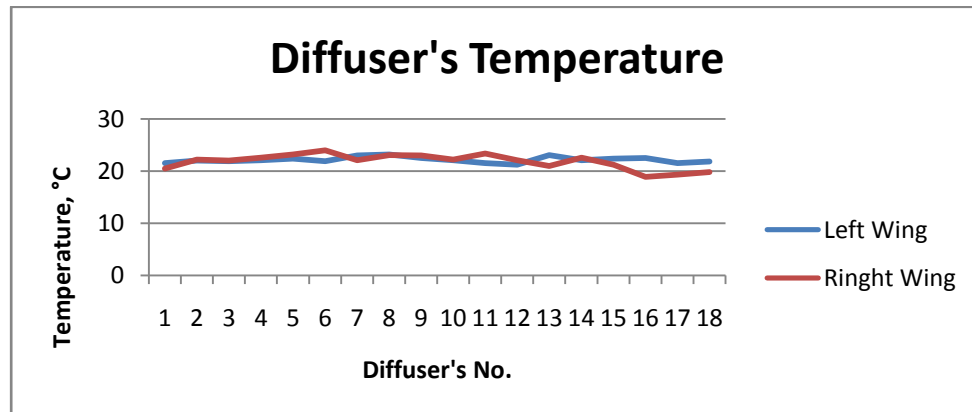
measurement we can see the result is parallel with the previous study regarding the cooling load in Block 13. From the previous study the cooling load was high in the morning at left wing but very low value at right wing [11]. When cooling load is high it's mean more cooled air is required for cooling the space. The AHU supposedly supply more cooled air to the leftt wing rather than right wing. When AHU supply more cooled air , the velocity and pressure in the ducting will be increased. Velocity of air at the left wing increased rapidly and the value is higher than the right wing , this result show that the AHU response with the need. At evening the cooling load is high at right wing and low at left wing. The AHU supposedly supply more cooled air to the right wing rather than left wing. When AHU supply more cooled air , the velocity and pressure in the ducting will be increased. Velocity of air at the right wing increased rapidly and the value is higher than the left wing , this result show that the AHU response with the need.

### 4.3 Average Temperature at Left Wing and Right Wing at Block 13

Temperature measurement was taken during the same time with velocity measurement.



**Figure 13 :** Variation of temperature at the diffuser on the Left Wing and Right Wing in evening ( 2.30pm – 5.30pm).



**Figure 14 :** Variation of temperature at the diffuser on the Left Wing and Right Wing in morning ( 9.30am – 11.30am).

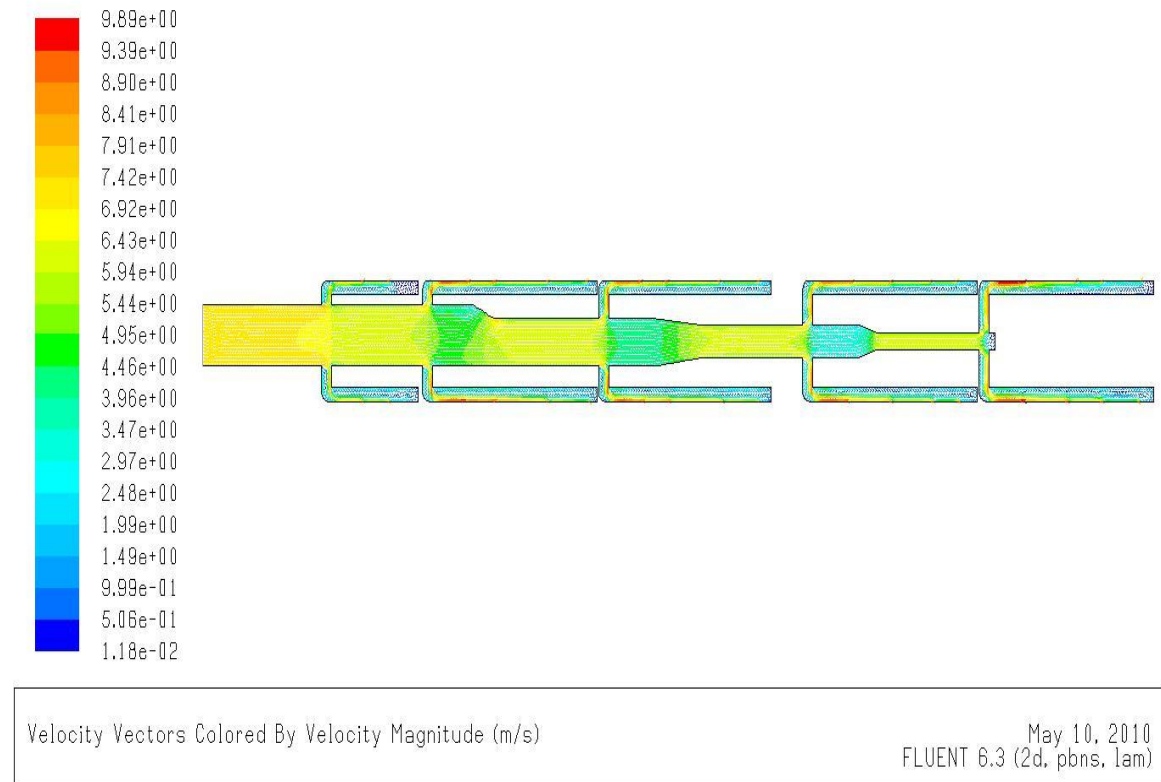
Figure 14 and 15 show the value of temperature inside the lecturer's office room. The temperature are quite stable and quite similar in each room.



## 4.4 Result from Simulation

### 4.4.1 Velocity of air in the ducting at Right Wing in the evening.

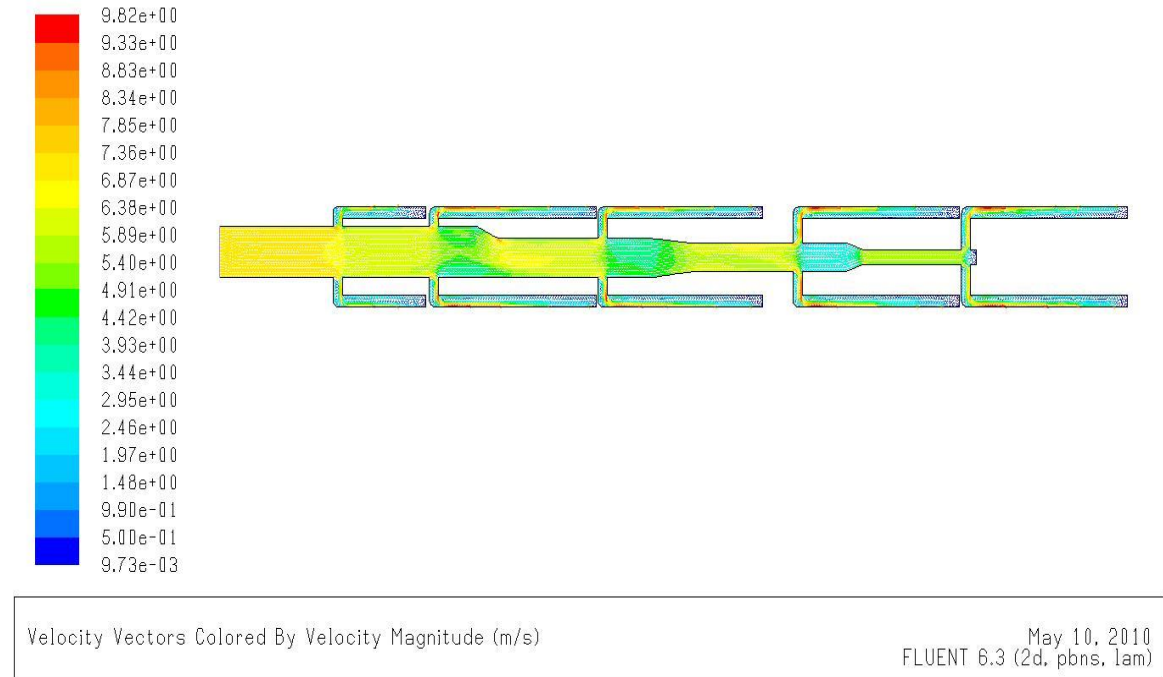
Figure 15 show the velocity profile in the ducting at Block 13. The velocity is higher at the inlet which is about 8.92 m/s and start to decrease when its goes through the ducting till to the end of the duct and at the diffuser. We can see the value drop in range of 2.48 m/s to 4.95 m/s when its its flow approaching the diffuser and at the end of the ducting.



**Figure 15:** Velocity profile of air in the ducting at Right Wing in the evening

Result from the simulation show that the average velocity coming out from diffuser in between 2.48 m/s to 6.92 m/s . These velocities are higher than the actual velocity due to the system in the ducting. This result is parallel to the actual result in Figure 11. From Figure 11, the velocity in evening should be high. The assumption by doing this simulation is the damper in the ducting was open 100%.

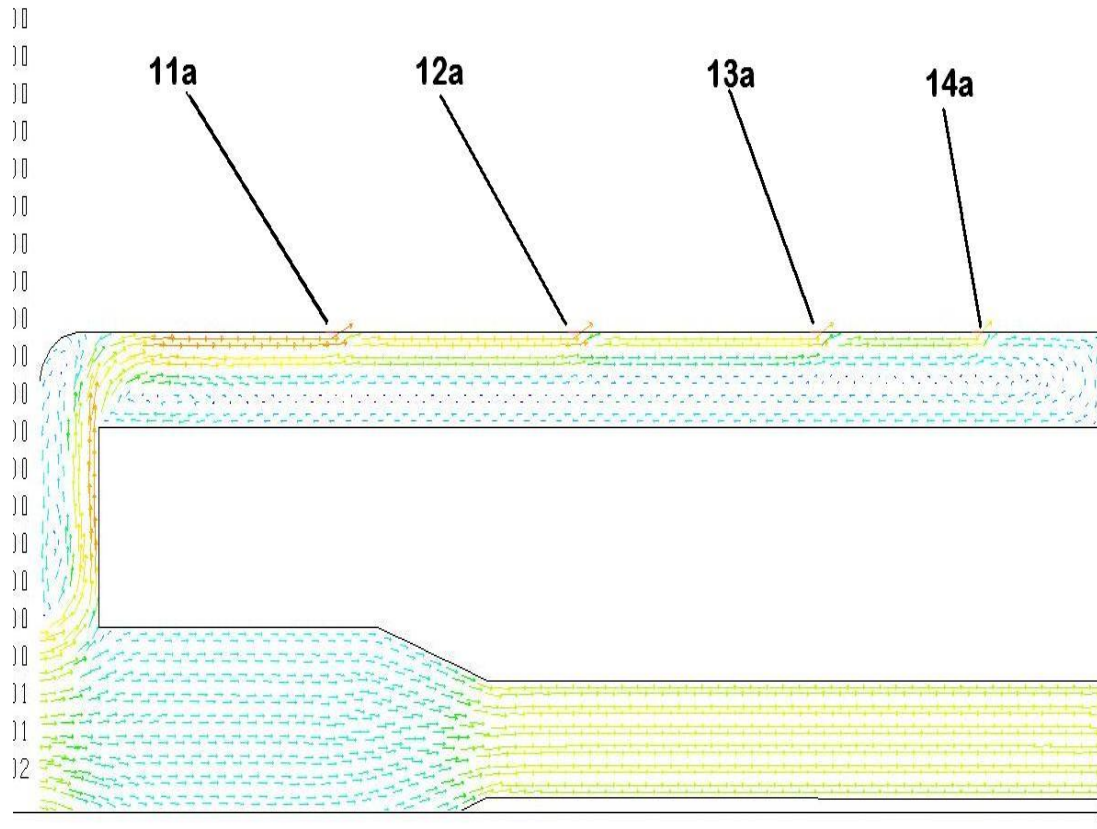
#### 4.4.2 Velocity of air in the ducting at Left Wing in the morning.



**Figure 16:** Velocity of air in the ducting at Left Wing in the morning.

Result from the simulation for the left wing in the morning show that the average velocity coming out from diffuser in between 2.95 m/s to 6.38 m/s . These velocities are higher than the actual velocity due to the system in the ducting. This result is parallel to the actual result in Figure 12. From Figure 12, the velocity in morning should be high. The assumption by doing this simulation is the damper in the ducting was open 100%. The different of the value also due to the pressure loss in the real condition thus make the velocity value decrease.

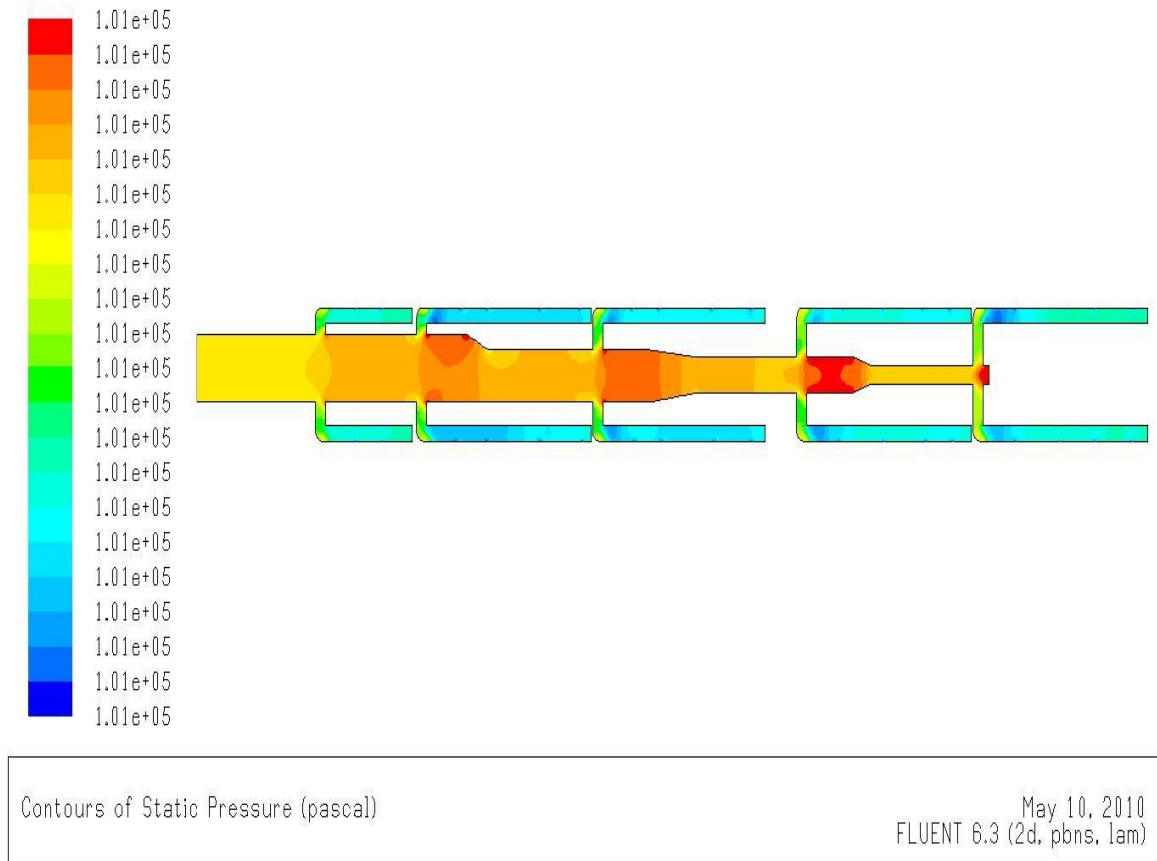
#### 4.4.3 Velocity vector in the ducting at Left Wing in the morning.



**Figure 17:** Velocity vector in the ducting at Left Wing in the morning.

This picture show velocity profile in the main ducting and velocity profile at 4 outlets which is at 11a , 12a ,13a and 14a. There are backflows at the end of the ducting branch. The backflows effect the efficiency of the system by decreasing the air velocity in the ducting.

#### 4.4.4 Pressure profile in the ducting at Left Wing and Right Wing



**Figure 18:** Pressure profile in the ducting at Left Wing and Right Wing

Both wing show quite similar pressure profile because of the inlet of the ducting supplied the same pressure by the AHU. The AHU also control the static pressure in the ducting to be 1000 Pa at all time. The pressure distribution as shown in Figure 18 indicate that the pressure inside the ducting is not changing.

## **CHAPTER 5**

### **CONCLUSION**

In the current research, the operational practices of the UTP air conditioning system were analyzed. The AHU performance will be analyzed using simulation and site survey. The objective of this research is to study the effectiveness and response of the air conditioning systems to the need in variable condition. The data of the cooling load were taken from the previous study, from the data we can estimate how much cold air needed in order to meet the need by the occupant of that building. This research had covered that most lecturers office were occupied during office hour.

Anemometer was used to measure the velocity and temperature at the diffuser during morning and the evening at both wings. Result from this measurement use to set as indicator of how much does the AHU supplied to the building during different condition.

FLUENT use to simulate the flow in the ducting in order to know the velocity profile, velocity magnitude, temperature and pressure in the ducting. Result from the simulation is use to compared with the actual result. After comparing the result, it shows that result from simulation is higher than the actual result. This is due to the system in the AHU and the ducting itself. This assumption is the damper is 100% open.

Both result show that the AHU supply enough cooled air to the office space thus meet the need. This project is useful to determine the efficiency of the AHU unit in all blocks in UTP.

## REFERENCES

- [1] . Roonak Daghieh and Kamaruzzaman Sopian, (2009), “ Effective Ventilation Parameters and Thermal Comfort Study of Air-conditioned”, Offices Solar Energy Research Institute, University Kebangsaan Malaysia (UKM).
- [2] S.C Sekhar, K.W. Tham, and David Cheong , (2001), “*Ventilation Characteristics of an air-conditioned office building in Singapore*”, Department of Building , National University of Singapore .
- [3] University Teknologi Petronas (UTP) Maintenance Department
- [4] William J. Fisk, (1999), “*Physical Characterization, Air leakage and Heat Conductions Gain in Building*” ,Indoor Environment Department , Lawrence Barkeley National Laboratory.
- [5] H. Yoshida, T. Iwami, H. Yuzawa, M. Suzuki, (1996) , “*Typical faults of airconditioning systems and fault detection by ARX model and extended Kalman Filter*”, ASHRAE Transactions 102 (Part 1) 557–564.
- [6] W.Y. Lee, J.M. House, D.R. Shin, (1997), “*Fault diagnosis and temperature sensor recovery for an air-handling unit*”, ASHRAE Transactions 103 (Part 1) 621–633.
- [7] A.L. Dexter, M. Benouatets, (1996),” *A generic approach to identifying faults in HVAC plants*”, ASHRAE Transactions 102 (Part 1) 550–556.
- [8] Loomans M., (1998), “*The measurement and Simulation of Indoor Airfow*”. PhD Eindhoven University of Technology, Netherland.

- [9] Hazim B. Awbi, (2008), "*Ventilation System (Design and Performance)*"  
Professor of Building Environmental Science, University of Reading, UK .
- [10] Keith Shepherd, (1999) "*VAV Air Conditioning Systems*"
- [11] Siti Nurfadhilah Ahmad Mujor. (2009) "*Reduction of Air Conditioning Load Within New Academic Building in Universiti Teknologi Petronas*".
- [12] McQuiston / Parker /Spittle,(2002) "*Heating, Ventilating and Air Conditioning*"
- [13] <http://www.lutron.com.tw>