MEASUREMENT AND NUMERICAL SIMULATION OF INTERNAL AIR FLOW IN FACULTY OF MECHANICAL ENGINEERING ADMIN OFFICE VENTILATION SYSTEM

MOHD AMIRUL HUSAINI BIN SAAT

Report submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

2012

ABSTRACT

Ventilation unit is critical for closed room such in Faculty of Mechanical administration office to recirculate air to entire room and maintain the surrounding temperature. Air is cycled to the inlet office space from inlet diffuser and outlet vent. However, the geometry position of the inlet diffuser and the furniture in the office effect the flow of the air cannot be supply directly to occupants. For this study, the objective is to investigate and observe the flow pattern in the office area due to obstruction by some partitions. A commercial CFD software was used which is ANSYS-Fluent to run the simulation. The real office dimension is 40 meters in length, 19 meters wide and 3 meters height. For the simulation RNG k-ε is used as the mathematical model for the flow in CFD. Results show that the obstruction occurred and effect the air distribution which is in agreement to previous researches. Analysis has been done after the simulation and the results show that the partitions in the office in good position which is the occupants are still in comfort while in working condition.

ABSTRAK

Unit pengudaraan adalah penting untuk bilik tertutup seperti di Pejabat Pentadbiran Fakulti Kejuruteraan Mekanikal untuk meyebarkan udara ke seluruh ruang dan mengekalkan suhu persekitaran. Udara yang dialirkan oleh peresap masuk akan mengair di dalam ruang pejabat dan akan disedut keluar ke peresap keluar. Namun, kedudukan geometri peresap masuk dan susun atur perabot memberi kesan kepada aliran udara di dalam ruang tersebut. Untuk kajian ini, objektifnya ialah untuk menyiasat dan melihat corak aliran di dalam kawasan pejabat kerana terhalang oleh pembahagi. Pelantar seperti ANSYS-Fluent digunakan untuk melakukan simulasi. Dimensi sebenar ruang pejabat ialah 40 meter panjang, 19 meter lebar and 3 meter tinggi. Untuk simulasi, RNG k-ε telah digunakan sebagai model matematik untuk aliran dalam CFD. Analisis telah dilakukan selepas simulasi dan hasil kajian mendapati pembahagi disusun dalam keadaan baik, dimana penghuninya masih dalam keadaan selesa semasa melakukan kerja.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF ABBREVIATIONS

IAQ	Indoor Air Quality
HVAC	Heating, Ventilation And Air Conditioning
CAD	Computer Aided Drawing
CFD	Computational Fluid Dynamic
FKM	Fakulti Kejuruteraan Mekanikal
LDA	Laser Doppler anemometry
PIV	Particle Image Velocimetry
DNS	Direct Numerical Simulation
TCM	Thermal Comfort Measurement
2D	Two Dimension
3D	Three Dimension
ASHRAE	American Society of Heating, Refrigerating and Air Conditioning Engineers
ANSYS	Analysis System
^{0}C	Celsius degree
m	Meter
cm	centimeter
ft	Feet
m/s	Meter per second
u_i	Velocity

H	Enthalpy
µ	Kinematic viscosity
k	Thermal Conductivity
c_p	Specific Heat
ρ	Density
p	Pressure
g_i	Gravitational Acceleration
clo	Clothing Insulation

CHAPTER 1

INTRODUCTION

1.1 Project background

Mechanical ventilation in buildings is common practice nowadays, due to the need to provide thermal comfort and good indoor air quality (IAQ) in enclosed spaces. Ventilation is a process of replacing or circulating the air in a space to provide better IAQ. The process includes remove dust, air bacteria, carbon dioxide and unpleasant smells, and replenish oxygen inside the area. The cooling and heating operation of air used in buildings is done by air conditioning system. In the other hand, air conditioning refers to the treatment of air to control humidity as well as temperature to create an environment which is comfortable to the occupant of the conditional space. When those systems are combined, it is called Heating, Ventilation and Air-Conditioning (HVAC). The functions of heating, ventilating, and air-conditioning are interrelated, especially with the need to provide thermal comfort and acceptable IAQ within reasonable installation, operation, and maintenance costs. HVAC systems can provide ventilation, reduce air infiltration, and maintain pressure relationships between spaces. This project focuses on the internal air flow simulation of ventilation system inside the Faculty of Mechanical Engineering administration office area.

1.2 Problem statement

The geometry and location of the inlet and outlet vent is important in HVAC system design. This is because it will affect the air distribution and air flow pattern in the office area. The air surrounds the whole area but the characteristic is different depend on the location. In order to observe the flow pattern, a simulation using Computational Fluid Dynamics (CFD) is important. From the perspective of thermal comfort and indoor air quality, the existing HVAC system should provide acceptable indoor environmental conditions occupants of the building to maintain the good indoor quality. This indoor air quality is important to occupants because it help them to do work in comfortable condition. Hence, they can increase their job performance. The experimental result should be similar in simulation result. The effective boundary parameters in this study should be considered are air temperature, air velocity, and relative humidity.

1.3 Objectives

The objectives of this project are:

i. To investigate the air flow pattern in FKM admin office.
ii. To check and measure air temperature and velocity in FKM admin office.
iii. To use simulation in CFD to check the flow pattern.

1.4 Project scopes

This project is focusing on numerical simulation of administration office room, which includes parameters such as air flow pattern, air velocity and temperature. The scopes are:

i. Setup the initial condition for the room temperature and air velocity.
ii. Setup and control the related boundary condition.
iii. Compare the simulation result and experimental result for temperature.

CHAPTER 2

LITERATURE REVIEW

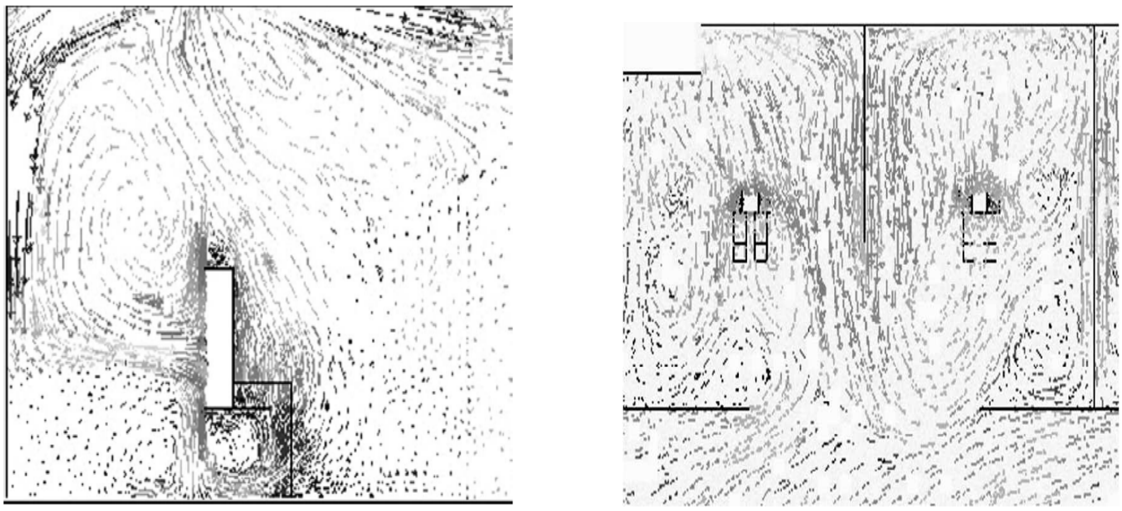
2.1 Introduction

The basic idea of ventilation systems started from natural type of ventilation which is control by nature and controlled by the building's orientation and placement of windows to circulate the air. Modern types of ventilation started in 1880s when the use of steam and electricity had spread. While electrical industries are growing, Tesla's invention of the electric fan in 1882 was a major innovation in helping people feels more comfortable during hot weather.

The development of air conditioning started with refrigeration. Due the cooling limitation by the fan, Charles Tellier from France invented vapour compression refrigeration machines. By 1911, air conditioning was to be great economic value while at that time control of indoor air and humidity started to be major concern to several branches to industries (Kuhnl, 2002). Thus resulting the expansion of science as part of air conditioning engineering. Nowadays, air conditioning is a method of mechanical ventilation system especially in closed room.

2.2 Air flow pattern

Nowadays, with growing concern over indoor air quality issues, the airflow pattern as a result of ventilation design has been considered very substantial in the assessment of air quality exposed to the occupants and comfort level due to air movement. The characteristic features of air propagation in a space results from the interaction among factors such as air inlets and outlets/exhausts, space design or layout, occupants and equipment as heat sources, the space enclosure, and activities or movements (Sekhar and Wilem, 2003). From the research by Sekhar and Wilem (figure 2), the airflow profile forms recirculation patterns due to obstructions by partitions, furniture and occupants. This is substantiated by an observation of a recirculation pattern moving upwards that results in dissipative flow towards the back of the occupant model, whilst there circulating flow moving downwards creates small turbulence below the seats.



(a) (b)

Figure 2.1: (a)Simulated vertical air flow around human model. (b)Simulated horizontal air flow around human model.

Source: Sekhar and Wilem, 2003.

Simple three dimensional numerical simulations (CFD) with laser Doppler anemometry (LDA) and particle image velocimetry (PIV) experimental measurements of indoor air flows has been perform by J.D. Posner. Geometry design at the boundaries of rooms (including walls, partitions, and furniture) can have a large effect on flow motion (Posner et al., 2002). Figure 2.2 shows the model room design with single partition in the middle and the ventilation has single inlet vent and single outlet vent.

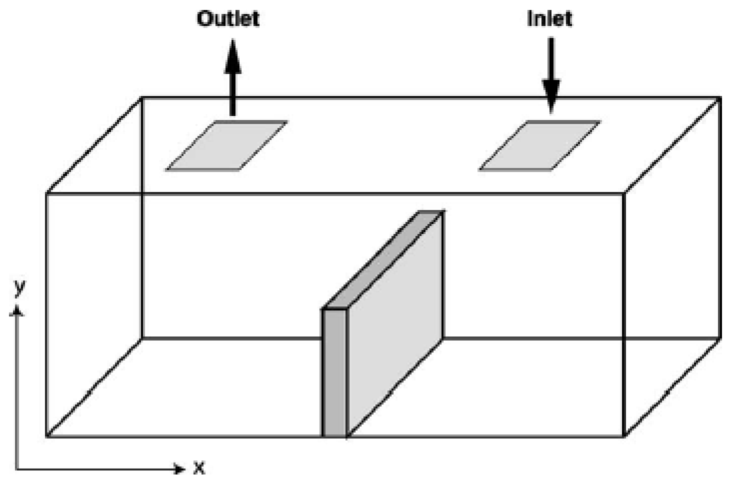


Figure 2.2: Drawing of model room showing the partition and its location.

Source: Posner et al., 2002.

Figure 2.3 shows the schematic diagram for LDA system, used a laser beam for measuring the velocity of particle in transparent or semitransparent fluid flows. It is mounted on an optical table perpendicular to the table that supports the model. This model utilizes a 1-W continuous argon-ion laser at 488 nm as the coherent light source.

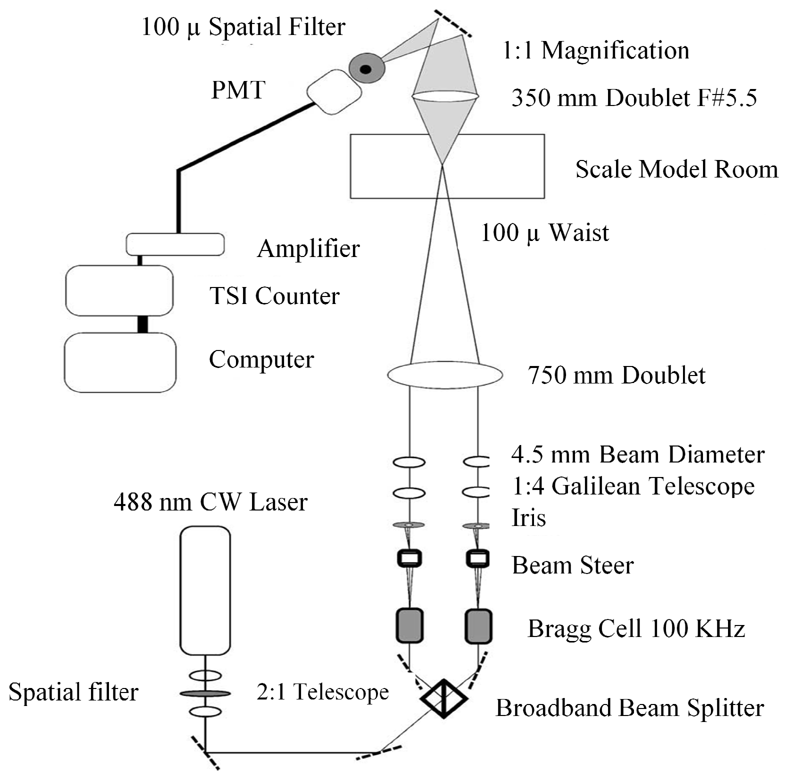


Figure 2.3: Schematic of laser Doppler anemometry system.

Source: Posner et al., 2002.

Result show that the flow moves up across the partition the get to the other half of the room in Figure 2.4. This shows that geometry design of a room effect the air flow.

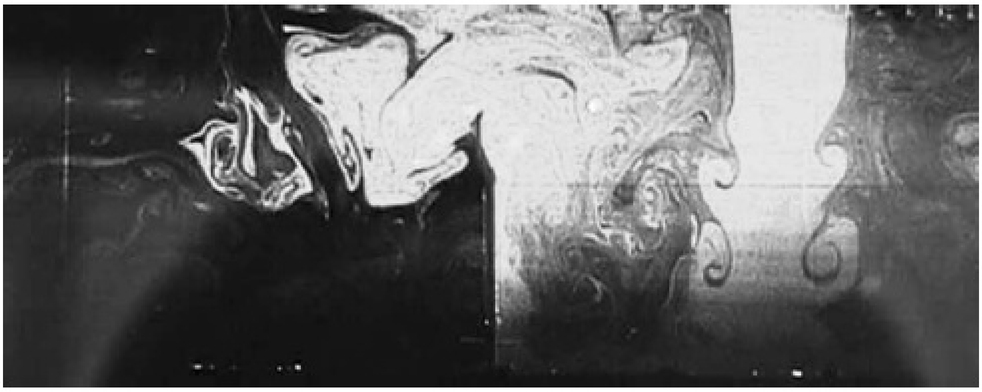


Figure 2.4: Flow visualization in the model room with partition in the middle.

Source: Posner et al., 2002.

2.3 Air Conditioning

The basic of air conditioning is based on characteristic as temperature, heat and pressure. Heat is energy, which can be transfer from one space to other. In air conditioning process, the mechanical concept is use to remove heat from a space to other with two conditions:

i. Fluids absorb heat when change from liquid form in to gases condition.
ii. Fluids reject heat when change from gases form in to liquid condition.

It is a derivation of refrigeration concept. Air conditioning can maintain low temperatures and humidity under adverse weather conditions. The recommended air conditioned outer limit parameters for comfort in summer is 24^0C dry bulb and relative humidity from 40% to 60% with research indicates that 80% of people will feel comfortable under these conditions.

In the office, it is ducted air conditioning system. It is a system that provides central air conditioning to all selected area that which is air need to be distributed by square outlet vent.

2.3.1 **Air distribution method**

The air distribution methods are refers to ways of controlling the air distribution within a ventilated room, the air with mechanical ventilation system means that the flow rate of air is controlled but the resulting distribution of air within the room is also dependent on the other factor. The air distribution can be divide into two types which is momentum (jet) controlled air and buoyancy (thermally) controlled air distribution (Etheridge & Sandberg, 1996). In ventilation, supply of momentum occurs in the form of jets and this can assumed to consider jet-controller air distribution.

2.3.2 ASHRAE 55-92 Standard

ASHRAE 55-92 standard is based on research in lab (indoor) with under control boundary condition. this standard is most to the relation between boundary thermal and indoor light activities like sitting and reading. The scope of this standard is no limit to a type of building only. So, it can use in various type of building with difference condition. In ASHRAE 55-92, the effective parameters are considered as temperature, moisture, air velocity and internal parameter.

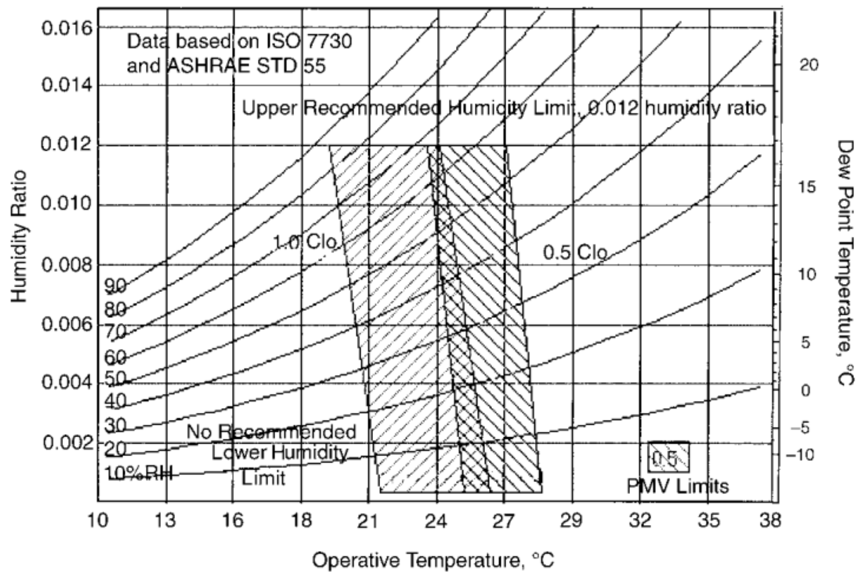


Figure 2.5: The comfort zone for environments that meet the above criteria.

From the Figure 2.5, two zones are shown, one for 0.5 clo of clothing insulation and one for 1.0 clo of insulation. These insulation levels are typical of clothing worn when the outdoor environment is warm and cool respectively. The operative temperature range allowed for intermediate values of clothing insulation may be determined by linear interpolation between the limits for 0.5 clo and 1.0 clo using the following relationships. The operative temperature is about 22 ^{0}C until 27 ^{0}C and 60% humidity.

The operative temperature meets the study study of the environmental comfort in a closed air conditioned work space in Malaysia (A.R.Ismail et al, 2010). The study had been carried out by the use of custom made Thermal Comfort Measurement (TCM) equipment in order to investigate the level of thermal in this laboratory, level of CO2, pressure, humidity, wind speed, ambient temperature and globe temperature. It shows that in this case, the operation temperature was set to 25 degree celcius and the mean humidity ratio was taken as 38.75 %, resulting good thermal comfort in the room.

2.4 Computational fluid dynamic (CFD)

Computational fluid dynamics or CFD is the analysis of the system involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. There is several unique advantage of CFD over experimented based approaches to fluid system analysis. Substantial reduction of lead times and costs of new designs, ability to study system where controlled experiments are difficult or impossible to perform and unlimited level of details or results are a few advantages of CFD techniques.

All CFD codes contain three elements, which is the pre-processor, a solver and a post processor. A pre-processor consists of the inputs of a flow problem to a CFD program by means of an operator friendly interface and the subsequent transformation of this input into a suitable form for use by the solver. After a problem is defined in the pre-processing stage, a solver will solve the problems numerically by using numerical methods such as finite difference method, finite volume method, and finite element spectral method. After the solution and solving process, the post-processing stage will interpret the outcome into various type of data output and visualization system, such as vector plots, particle tracking, 2D or 3D surface plots and contour plots (Lim, 2005).

2.4.1 Mathematical Model

A flow field may be described by the conversation of mass, energy and momentum. For given boundary conditions, the resulting flow pattern is determined by solving the combined Navier-Stokes and energy or any other scalar equations as in equations 2.1 to 2.3 (Loomans M, 1998).

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0\ , \qquad (2.1)$$

$$\frac{\partial}{\partial t}\left(\rho u_i\right)+\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]+\rho g_i, \qquad (2.2)$$

$$\frac{\partial}{\partial t}(\rho H)+\frac{\partial}{\partial x_i}\left(\rho u_i H\right)=\frac{\partial}{\partial x_i}\left[\frac{k}{c_p}\frac{\partial H}{\partial x_i}\right]+S_H, \qquad (2.3)$$

where, u_i is the velocity component (u,v,w), p is the pressure, H the enthalpy and S a source term. The diffusion term is indicated by the kinetic viscosity μ, the thermal conductivity k and the specific heat c_p. The time is indicated with t, x_i is the coordinate axis (x,y,z), ρ is the density and g_i is the gravitational acceleration. Direct Numerical Simulation (DNS) calculates the turbulent motion by solving the Navier-Stokes equations. A fine grid and a small time step are required to determine the flow field up to the smallest length scale, but the limitations in computer capacity restrict the application DNS to flows with moderate Reynolds number (Loomans M, 1998).

2.4.2 RNG $k - \varepsilon$ Turbulence Model

The RNG $k - \varepsilon$ turbulence model is obtained via a statistical mechanics approach, in which the small-scale motions are systematically removed from the governing equation expressing their effects in term of larger scale motion and a modified viscosity. The advantage of this model is that the constants in the equations are calculated explicitly.

Furthermore, a new term appears in the ε-equation, which account for anisotropy strained turbulence flow. This term is incorporated through a modeled constant in the production term, based on the equilibrium assumption that production equals dissipation, restricting the RNG k-ε turbulence model to a coarse grid approach near walls. The same default wall functions as in the standard *k*-*ε* turbulence model are valid in this case.

$$\frac{\partial \rho\varepsilon}{\partial t} + div(\rho U\varepsilon) - div\left[\left(\mu + \frac{\mu_\tau}{\mu_\varepsilon}\right) grad(\varepsilon)\right] = \left(C_{1\varepsilon} - C_{1RNG}\right)\frac{\varepsilon}{k}P - C_{2\varepsilon}\rho\frac{\varepsilon^3}{k}, \quad (2.4)$$

$$C_{1RNG} = \frac{\left(1 - \frac{\eta}{\eta_0}\right)}{(1+\beta\eta^3)}, \quad (2.5)$$

$$\eta = \left(\frac{P_s}{\mu_T}\right)^{0.5}\frac{k}{\varepsilon}, \quad (2.6)$$

$$C_\mu = 0.085;\ \sigma_k = 0.7179;\ C_{1\varepsilon} = 1.42;\ C_{2\varepsilon} = 1.68, \quad (2.7)$$

where η_0 and β additional model constants and which are equal to 4,38 and 0.015, respectively and P_s is the part of production. The standard values of the other constants as in equation (2.7) are regarded s suitable for this application. The *k* equation is the same format as in the standard *k*-*ε* turbulence model, and the same wall profiles are applied.

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter includes the step from beginning until the end process of simulation method. This chapter also explains how this method applied in this project.

3.2 Project Flow Chart

After understanding the objectives, it comes to set up data collecting and simulation to final stage. It is necessary because to ensure there are no steps in the flow left behind or undone. The execution of the project will be done smoothly without any discrepancies. The overall flow of this project is shown in Figure 3.1.

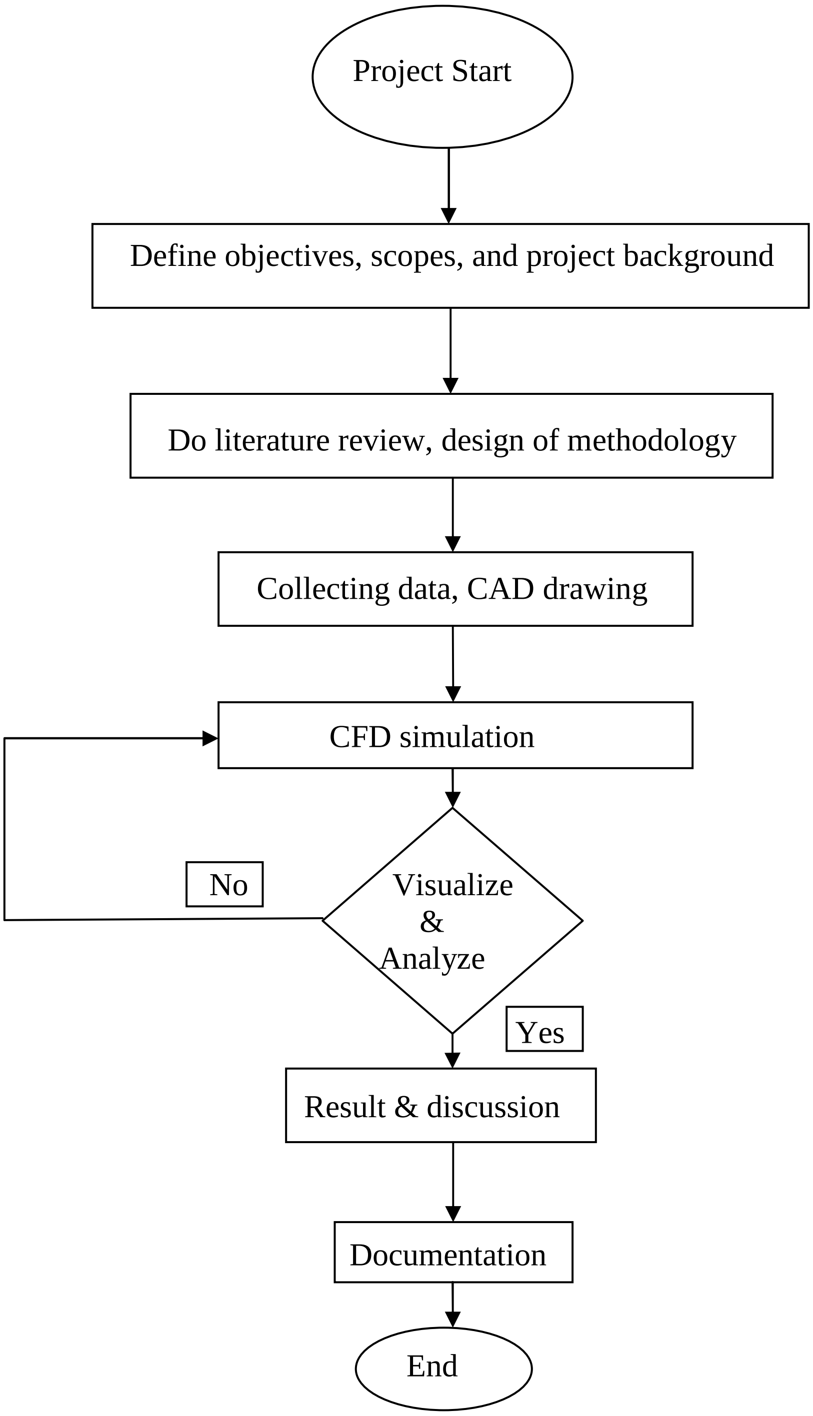


Figure 3.1: Project Flow Chart

3.3 Case Study

The scopes of study in this research are involving the measurement of indoor air parameters and the air flow pattern simulation inside office room using CFD software to find out between the effect of air distribution and the geometry location of inlet and diffuser in the room area. The physical conditions of lecture room are maintained without any modification during the parameters measurement. The main procedures of this study are as listed below:

a) Measurement the size of FKM administration office room.
b) Measurement of indoor air parameter.
c) Drawing the office area using CAD.
d) Performing CFD analysis on indoor air flow in simulation.

3.3.1 Office Room Boundary Condition

In this study, the office room is the boundary of the experiment and simulation. The air conditioning system is central air conditioning type, where air conditioning system that uses ducts to distribute cooled or dehumidified air to more than one room. For this simulation, the initial air velocity was set to 3.0 m/s at steady state with room temperature of 23^0 C. The other criteria of the boundary condition that has been chosen are as follow:

a) The office room should be mechanically ventilated and the air inlet vent and air outlet vent are well function and in good condition.
b) The office room should be fully closed, well sealed and without gap for outside ventilation.
c) The flow is steady flow.

3.3.2 Instrumentation

There are 3 types of instrumentation in order to obtain raw data for this study which is as follow:

a) Room Dimension Data Measurement.
b) Indoor Air Parameter measurement.
c) CFD Simulation.

3.4 3D Modeling Designs

Figure 3.2 is the technical drawing of administration office that has been build and occupy by workers daily. The total length of the room is 40 meters, 19.4 meters width, and 3 meters height. There are 17 rooms around the office area, including 1 meeting room. The dimension of the inlet diffuser and outlet vent is same, which is 121.92 cm^2 (4 ft^2). The partition is about 1.6 meter. There are about 42 inlet vents and 17 outlet vents at the ceiling of the office.

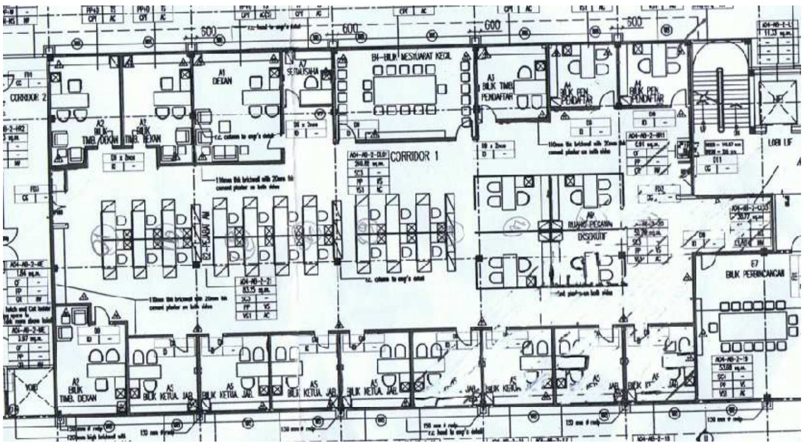


Figure 3.2: Administration Office Technical Drawing