Airflow study of a Split-Type outdoor unit subjected to near wall effect using Computational Fluid Dynamics (CFD) simulation

by

Kueh Seow Hian

16501

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical)

JANUARY 2016

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Airflow study of a Split-Type outdoor unit subjected to near wall effect using Computational Fluid Dynamics (CFD) simulation

By

Kueh Seow Hian

16501

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

In partial fulfilment of the requirements for the

Bachelor of Engineering (Hons)

Mechanical Engineering

Approved by,

____

(AP. Dr Morteza Khalaji Assadi)

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

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 acknowledgements, and that the original work contained herein have not been undertaken o done by unspecified sources or persons.

Kueh Seow Hian

Abstract

The performance of an air conditioner depends on the refrigeration cycle that consists of compressor, condenser, evaporator and expansion valve. The temperature and pressure of each component are crucial to the performance of the system. Each design has different operation condition (temperature and pressure) depending on the refrigerant that is used in the refrigeration cycle and depending on the climate condition on that particular places. Once the temperature and pressure are out of the range of the standard operating condition, the power consumption will increase for the component to reach the requirement and hence the overall performance and energy efficiency will drop. There are many factors that are affecting the performance of air conditioning such as an overheated compressor or dirty coil. Besides that, the distance of the wall gap between the wall and outdoor unit is one of the factor that affect the performance of air conditioning unit. The distance of wall gap will create static pressure around the wall gap which will affect the air flow towards the outdoor unit. The shorter the distance of wall gap, the higher the static pressure, hence prevent air from going through the outdoor unit. The objective of this project is to study the airflow of a split type outdoor unit when it is installed near the wall and to determine the distance between outdoor unit and wall that gives optimum performance. For the first stage, CAD (Computer Aided Design) drawing of the actual model of DAIKIN outdoor unit of 5SLY15F SERIES is needed to simplify by using a CAD software called ANSYS SPACECLAIM and mesh it before simulation process. The outcome that is obtained from the CFD simulation that will be done in OPENFOAM is the air flow rate (CFM) that are coming out from the outdoor unit. Based on the results that was obtained, the minimum installation space for outdoor unit is 60mm in x-axis and 60mm in y-axis.

ACKNOWLEDGEMENT

In completion of this Final Year Project, I would like to express my gratitude to all parties for all the guidance and useful advises which helped me to complete my Final Year Project successfully.

First, I would like to extend my appreciation to my Final Year Project supervisor, AP. Dr Morteza Khalaji Assadi, for his guidance and concerns in the project and helped me through all the problems which I faced in this project and to my Host Company supervisor, Dr. Chin Wai Meng, who give me the chance of having this project and ask two CAE engineer, Mr Mohd Anuar Bin Abd Aziz and Mr Low Lee Leong to guide me and help me in the simulation especially on the OPENFOAM software and their time for solving the problems together.

Last but not least, the acknowledgement also goes to my beloved family and peers who showed continuous support and motivation during the process of project completion.

Table of Content

List of Figures

List of Tables

CHAPTER 1
INTRODUCTION

1.1 BACKGROUND

Malaysia is a tropical country where the temperatures are around 25°C to 35°C throughout the year and the weather are very hot and humid especially in the major cities. The weather on those islands surrounding Malaysia are less compared to the weather around cities due to cool breezes. Besides that, many highlands of Malaysia such as Cameron Highland, Genting Highland and so on have cooler temperature compare to cities where the temperatures are lower than 25°C. Due to the hot and humid weather condition in Malaysia, the number of air conditioning installed in residential area and industrial area are increasing. In residential area, most of the residents installed split type air conditioner unit which is more energy saving and good cooling effect [1].

A split type air conditioner consists of an indoor unit and an outdoor unit. The indoor unit is the device that supplies the cold air inside the surrounding of the room which is linked with an outdoor unit that is installed outside of the cooling/heating space [2]. The indoor unit consists of evaporator while condenser and compressor are installed in the outdoor unit. There are two types of split type air conditioner which are single split unit and multi-split unit. Single split air conditioner consists of one indoor unit that connected to one outdoor unit while multiple indoor units that connect to single outdoor unit is called multi-split air conditioner. Figure 1.1.1 and Figure 1.1.2 show the types of split type air conditioner.

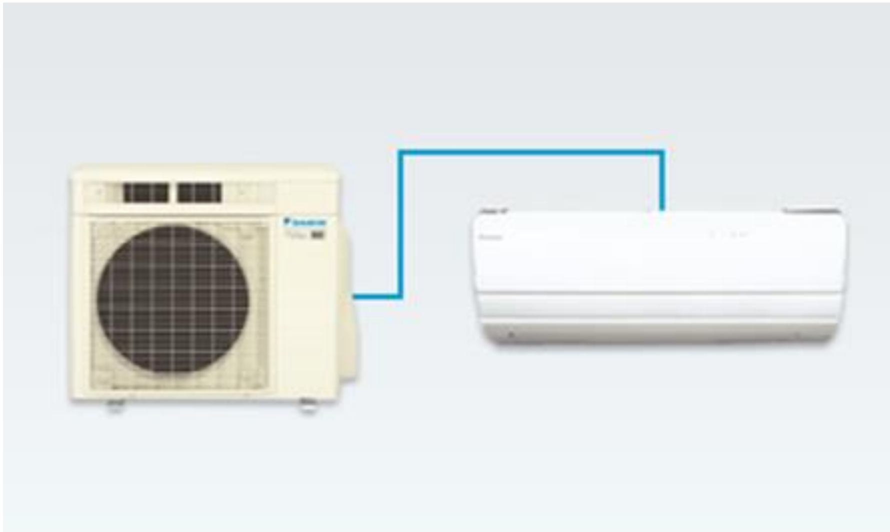


Figure 1.1.1: Single Split Air Conditioner

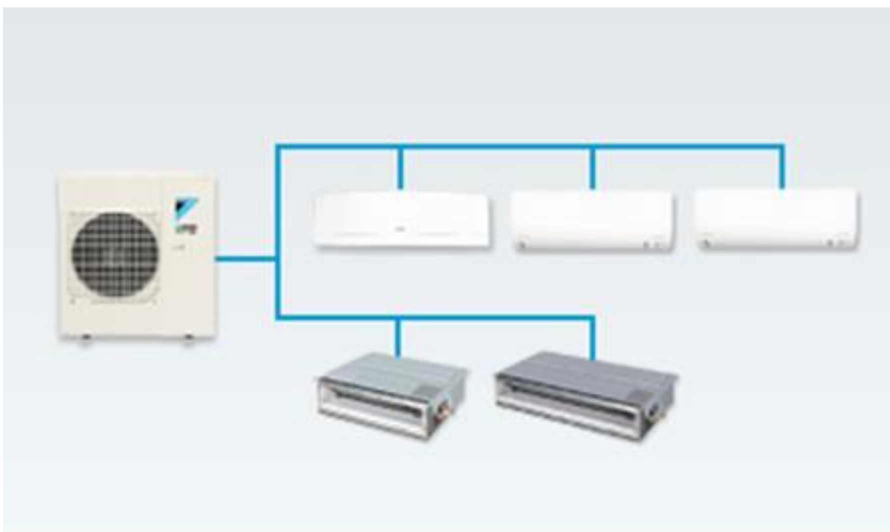


Figure 1.1.2: Multi Split Air Conditioner

Figure 1.1.3 shows the path of the refrigeration cycle. The whole air conditioning system cycle is supported by 4 main components which are the condenser, evaporator, expansion valve and compressor. The refrigeration cycle starts with a cool, low pressure mixture of liquid and vapor refrigerant that enter the evaporator where this fluid absorbs heat from the surrounding fluid. This process will heat up fluid inside the evaporator and produce low pressure superheated vapor refrigerant which then flows through the compressor. The compressor then absorbs the vapor refrigerant from the suction line and compresses it into a high pressure and temperature superheated vapor refrigerant. This forces the vapor to flow towards the condenser where heat exchange takes place. During condensation process, heat is transferred from hot vapor refrigerant to cool ambient air or cold water. This process changes the phase of superheated vapor refrigerant into a subcool liquid refrigerant before it leaves the condenser without changing the temperature. Once the subcool liquid refrigerant pass through condenser, it flows to expansion valve where this device restricts the amount of subcool liquid refrigerant flowing into the evaporator. At the same time, the pressure of the liquid refrigerant drops and it changes phase into a mixture of liquid and vapor refrigerant. From here onwards, the whole cycle starts again from the evaporator.

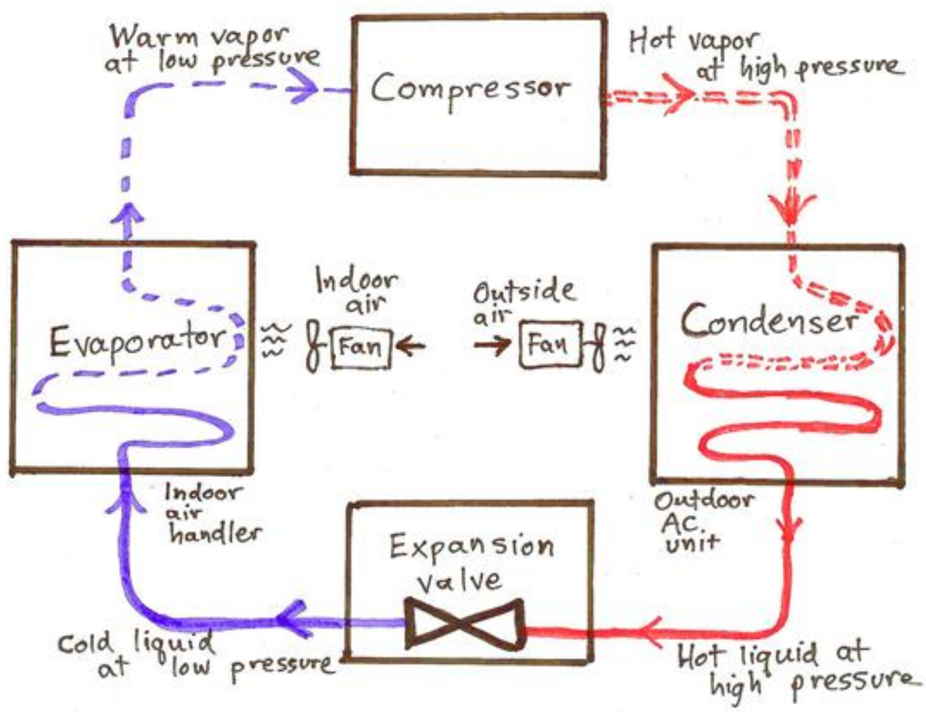


Figure 1.1.3: Air Conditioning System

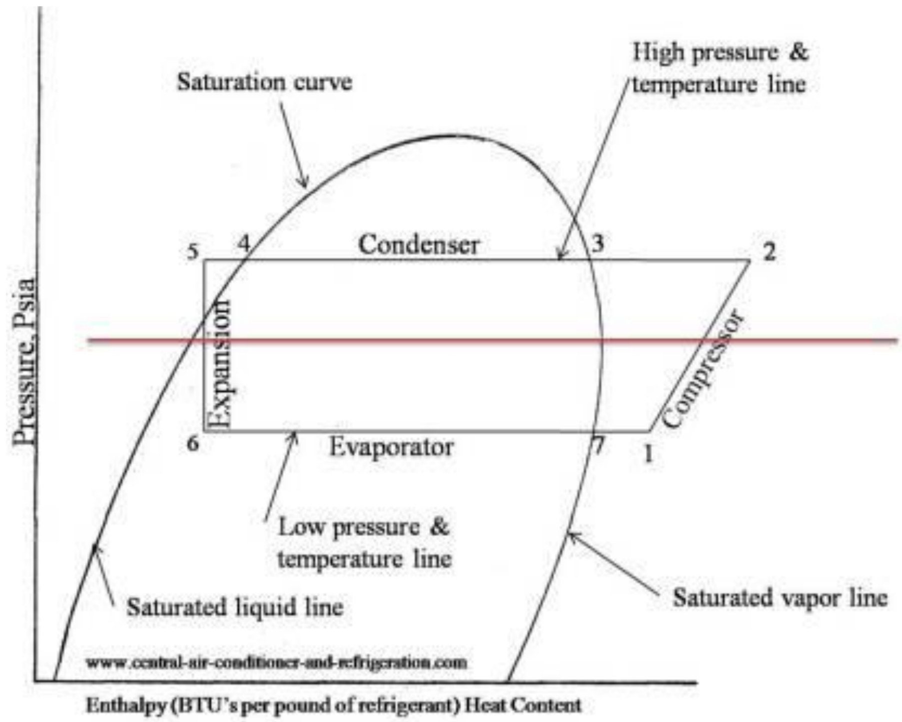


Figure 1.1.4: P-h Diagram

There are many factors that influence the performance of split unit air conditioning system. One of the factors that affect the performance of air conditioning system is wall gap between the wall and outdoor unit [3]. The distance that separates the walls of particular building with the outdoor unit can influence the whole performance of the air conditioning where the performance can be calculated using COP (Coefficient of Performance) and EER (Energy Efficiency Ratio) [4]. Based on Daikin Installation Manual, they set the standard for the installation space between the outdoor unit and the wall to be more than 100mm in y-axis and 50mm in x-axis [5]. With the standard distance that provided, this is to give space for the outdoor unit to run at optimum operating condition.

1.2 PROBLEM STATEMENT

The outdoor unit has been designed under a condition of unobstructed air flow through the fin-tube heat exchanger (free-throw conditions). Air is drawn in from the surrounding and exhausted out by the propeller into the same atmospheric space. The rated air volumetric flow rate flows through the heat exchanger.

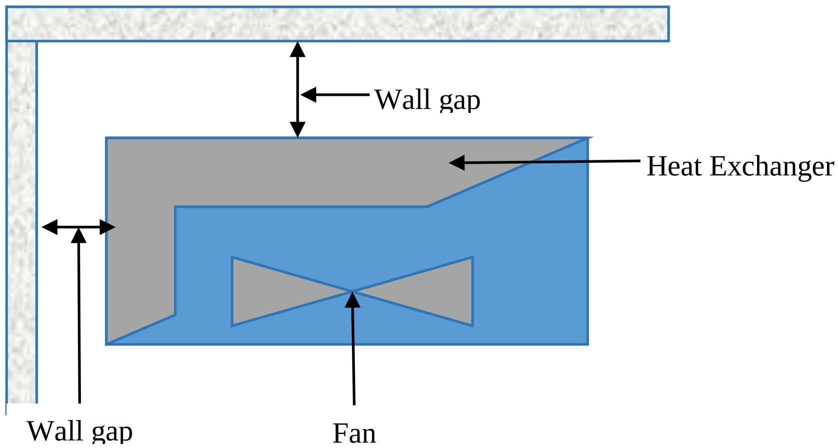


Figure 1.2.1 Air Gap between wall and outdoor unit

Based on figure 1.2.1 that is shown above, when the distance of the wall gap between the walls and outdoor units is decrease, the resulting obstruction will increase the static pressure, reducing the air flow rate that passes through the condenser heat exchanger. Consequently, the heat rejection within the condenser will reduce causing an inadequate cooling of the refrigerant. Because of that, the performance of air conditioning in total will decrease [6]. With the reduced air flow rate that flow into the whole system, it will cause the compressor discharge pressure and temperature to increase. Hence, the cooling capacity of the air conditioning system will drop and the energy consumption will increase which bring the performance to drop. In addition, the lifespan of each components inside the system will decrease especially the compressor before it fails. This relationship can be seen in figure 1.2.2 below [7].

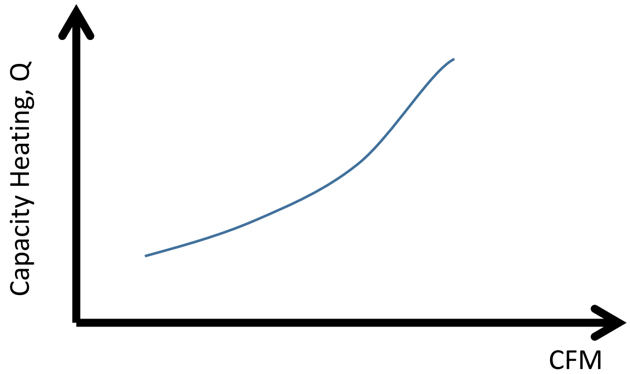


Figure 1.2.2: Relationship between Q and CFM

The closer the gap is, the more adverse is the situation. The limitation for this situation is the maximum allowable compressor discharge and suction pressures. A typical operating pressure for the compressor is at about 400 psig up to 500 psig for high ambient application. This relationship is shown in accompanying figure 1.2.3 below [8].

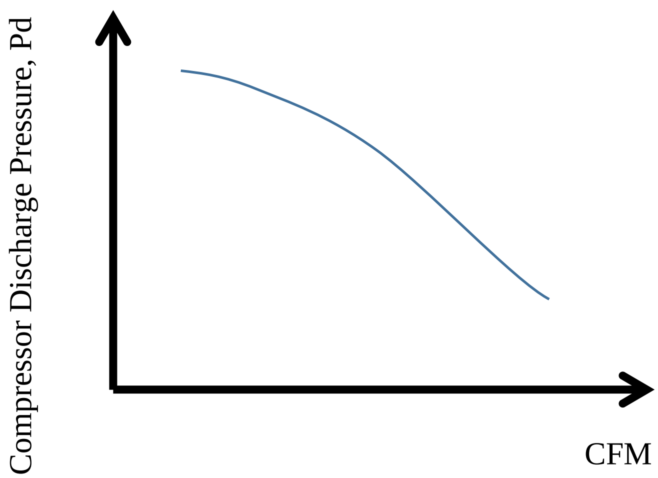


Figure 1.2.3: Relationship between Pd and CFM

At the same time, the static pressure experienced by the propeller fan increases as the gap becomes smaller. This changes the rotational speed of the fan which then affects the air flow rate delivered. For such changes in fan behavior, it is summarized as a fan curve.

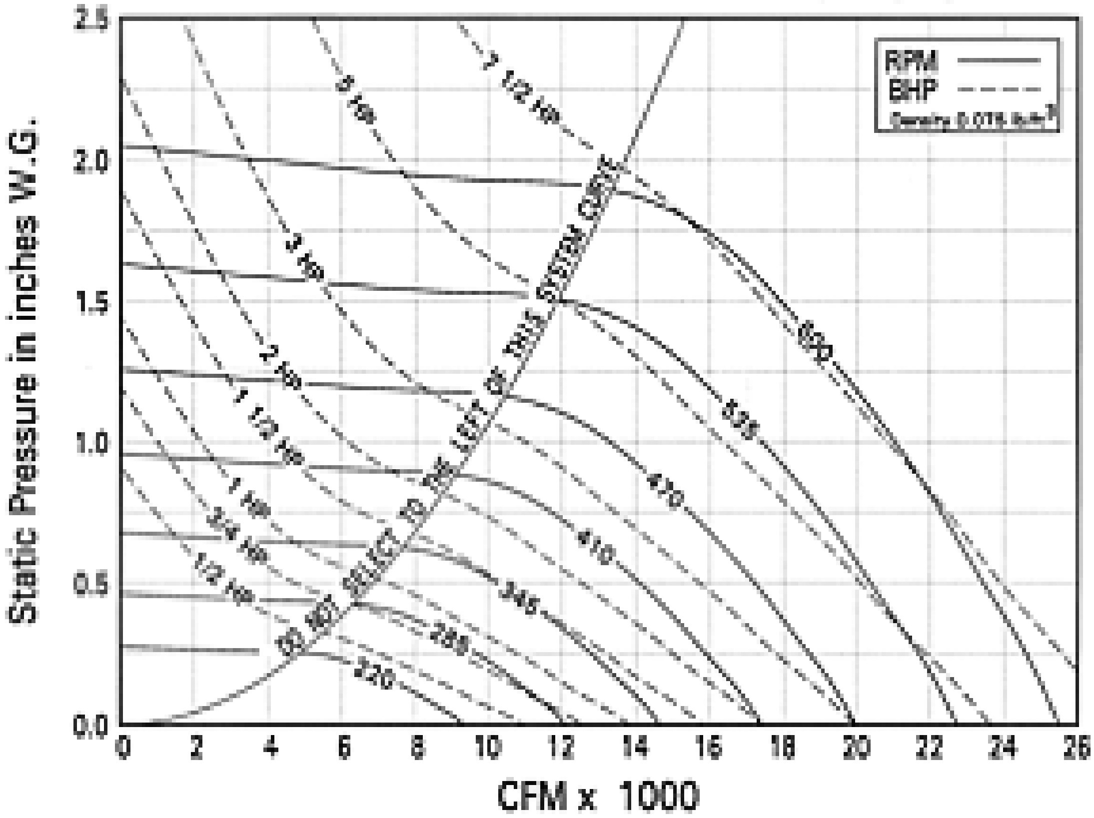


Figure 1.2.4: Relationship between static pressure and CFM

The accompanied figure 1.2.4 [9] shows how the change in static pressure changes the fan delivered volume flow rate (CFM) for a particular fixed RPM. Such relationship can be inputted inside the intended CFD software as a User Defined Function (UDF). On top of that, this could also cause air short-circuit to take place between the air discharge and air intake.

1.3 OBJECTIVES

The objectives of this study are:

- To investigate the relationship between air flow rates (CFM) and the distance of wall gap between wall and outdoor unit.
- To find the minimum installation space of outdoor unit.

1.4 SCOPE OF STUDY

The scope of study are:

a) The air conditioning model that is used in this study is the simplified CAD model of DAIKIN outdoor unit of 5SLY15F SERIES.
b) The distance of wall gap between walls and outdoor unit without considering the surface roughness or the material of the walls.
c) The output of the study is the distance of wall gap between the wall and the outdoor unit versus the air flow rate (CFM) and heat distribution.
d) The simulation only focuses on turbulent flow.
e) The working fluid is air and its properties are constant.

CHAPTER 2

LITERATURE REVIEW

Split type air conditioners are widely installed in residential areas and office buildings because of the simplicity of split type air conditioner and its flexibility [3]. Due to the hot and humid climate in Malaysia all the year where the temperature and humidity of the air are almost constant on every month which measured the average temperature of 20°C - 30°C, the usage of household appliances is assumed to be constant throughout the year [10]. Based on figure 2.1 that shown the surveys that were conducted in one of the states in Malaysia shows that air conditioner is the biggest contributor among the other electrical appliances that used inside household which is 1,167kWh per year [11]. While in office buildings, the majority of power consumption that was used is the air conditioner which consists of 57% of total power consumption [12].

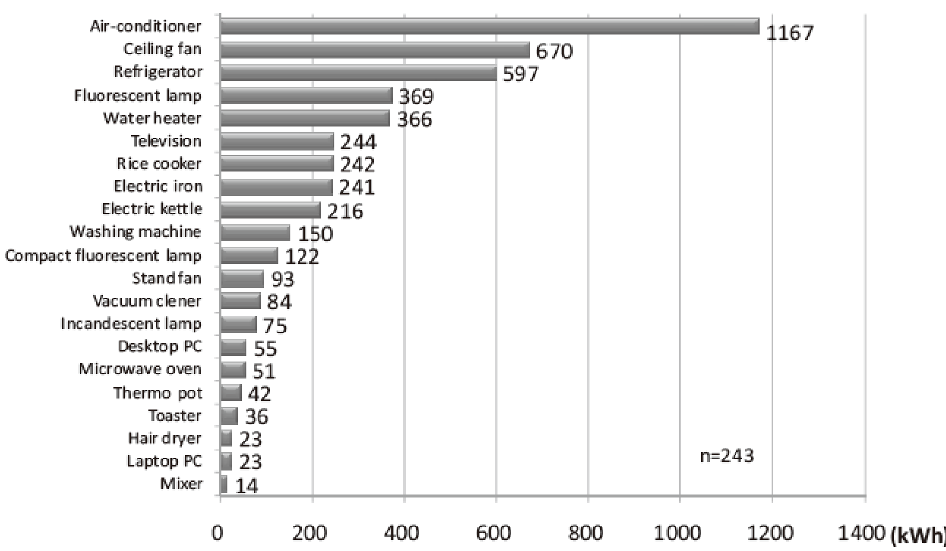


Figure 2.1: Yearly Electricity Consumption [13]

Air temperature and air flow rate have brought significant effect on the performance of an air conditioner. If the coil temperature increase by 1°C, the COP of air conditioning unit will decrease by 3% [3]. Once the temperature increase more than 45°C, the whole air conditioning system will trip and fail to function due to excessive working pressure that occurs in condenser and compressor [14]. The distance of the wall gap between wall and outdoor unit is one of the factors that influences the air flow rate. A restriction will cause the temperature to rise and increase the power consumption with a reduction in low cooling capacity, and this can eventually cause the whole system to trip.

There is a few research paper that studied about the placement of outdoor unit that influences the performance of air conditioning. Chow et al. [14, 15] had studied the effect of the placement and re- entry on outdoor unit of split-type air conditioning unit. He claims that the position of outdoor unit for each re-entrant affect the performance of the air conditioner which the number of condensing units at tall buildings increase, the heat energy will accumulate and can cause substantial energy waste and deteriorate the operation of the condensing unit. According to Avara et al, they used k-ε model to simulate the optimum placement of outdoor unit and the results are good for mean velocity in many cases [3]. Figure 2.2 and figure 2.3 show the results that Avara got from the CFD simulation that they did.

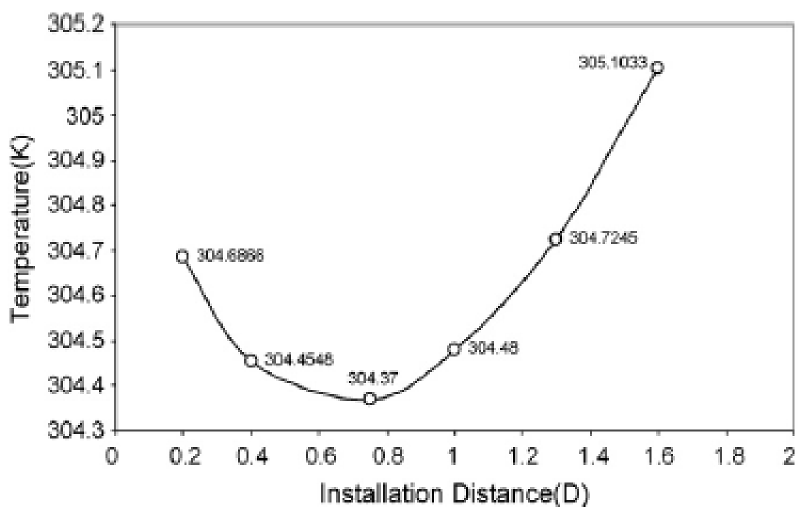


Figure 2.2: Graph of temperature VS installation distance

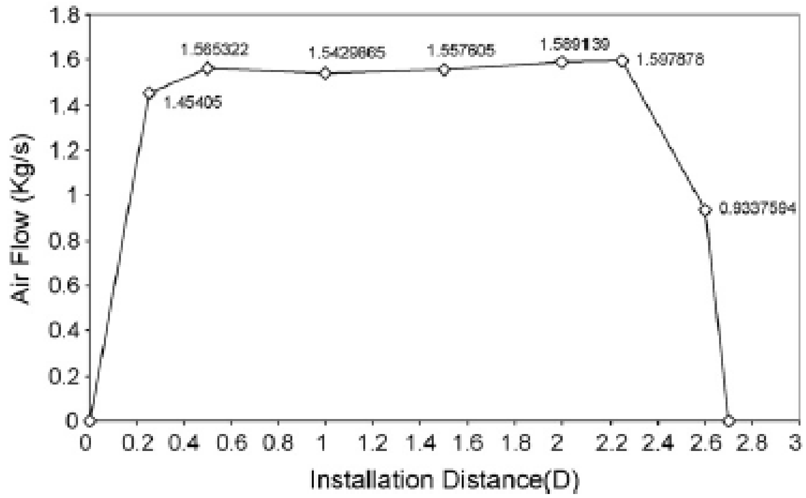


Figure 2.3: Graph of temperature VS installation distance

CFD simulation is a suitable tool to study the complex fluid flow phenomena like air flow in and around buildings, and of course, air conditioner placement can be studied through CFD [3]. There is some assumption need to be done for CFD simulation such as the flow is assumed to be in steady state, incompressible and the air is assumed to be in turbulent flow. Last but not least, natural convection and radiation heat transfer are neglected to simplify the simulation [16]. The k- ε model is used in the simulation which there are five transport equations of k-ε model in a 3D vector space that are used to solve the steady-flow thermal problem [15].

CHAPTER 3

METHODOLOGY

3.1 Project Activities

There are several stages to achieve the objectives of this project. Figure 3.1.1 below shows the flow chart of the whole process of this study.

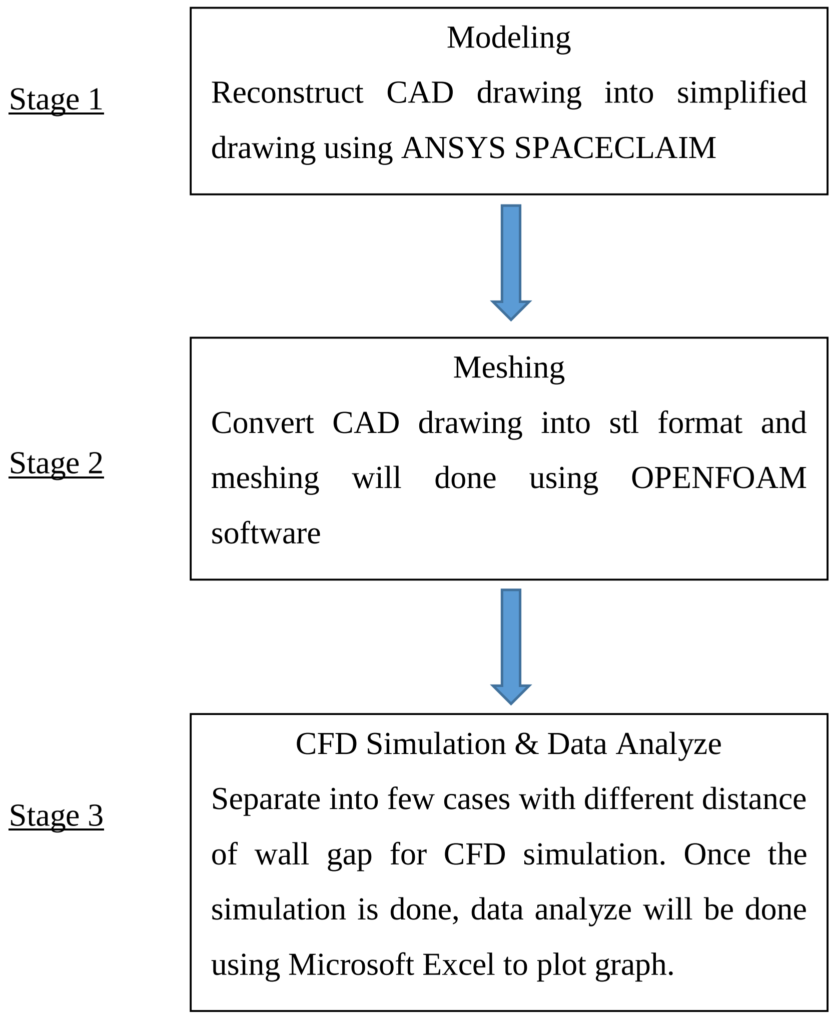


Figure 3.1.1: Methodology Chart

First of all, the first stage of this project is started by modelling and simplifying the CAD drawing of DAIKIN outdoor unit (5SLY15F SERIES) that acquired from Daikin Research & Development (DRDM) Sdn. Bhd. Then, it will proceed to the second stage where meshing process will be done using a software called OPENFOAM. The last stage is to run CFD simulation using the software called OPENFOAM to find the CFM (air flow rate) and the temperature of heat carrier around condenser.

Stage 1

The first stage of this project is started by modelling and simplifying the CAD drawing of DAIKIN outdoor unit (5SLY15F SERIES) that acquired from Daikin Research & Development (DRDM) Sdn. Bhd. The process of simplifying the model is to remove the unnecessary features that are within the model and also to reduce the amount of mesh generated on the model and hence will reduce the time for CFD simulation. In this stage, the casing of the outdoor unit by drawing a simple rectangular block and the components such as fan blade, grilled casing, coil, fan bracket and bellmouth are remain which can be seen in figure 3.1.2.

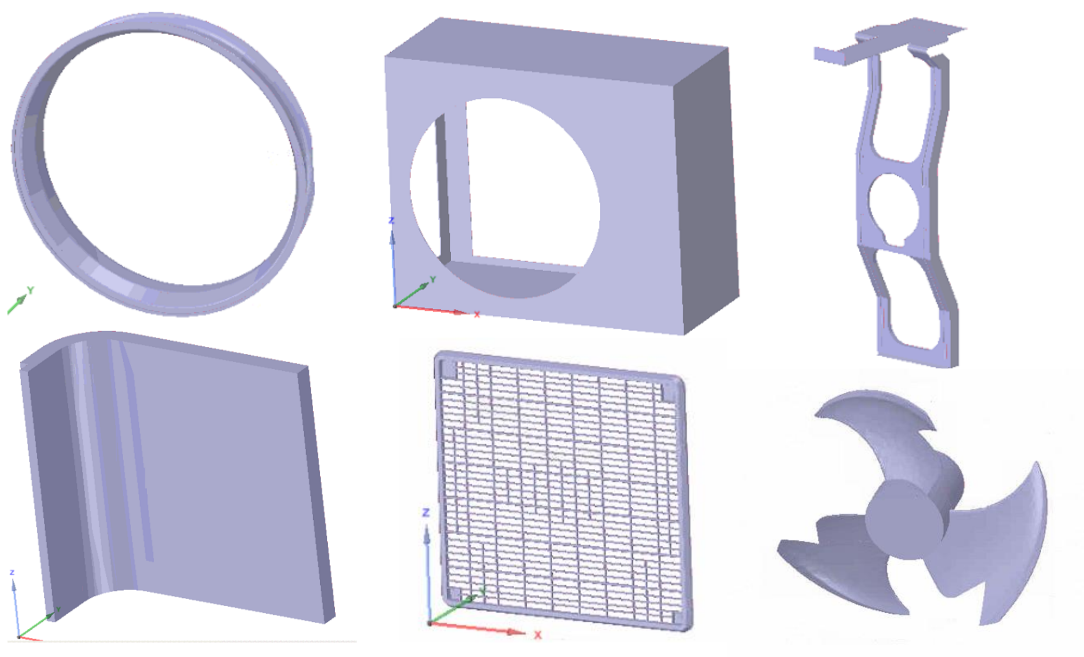


Figure 3.1.2: Components of 5SLY15F

Since the original CAD drawing contain geometry problem on some components, geometry clean-up process will be done. More complex geometry even need to reconstruct. For example, the propeller fan and the grill. Since the CFD simulation is to simulate the actual case of outdoor unit where the system is functioning and consist of a rotating fan blade, a solid cylinder is created around the fan blade. This solid cylinder is called MRF (Multiple Reference Frame) zone where this solid is created to allow OPENFOAM to make the fan blade to rotate within the MRF zone during the simulation. MRF model is a steady state approximation where the cell zones can be assigned on different rotational speed. The flow within each cell zone can be solved using the MRF function which the MRF zone adds in the coriolis force source term into the navier stokes equation. The reason of using MRF model in this project is because of the cell zone of fan blade are rotating during simulation. By applying MRF, OPENFOAM will detect the selected moving cell zone. Figure 3.1.3 shows the MRF zone that was created using ANSYS SPACECLAIM.

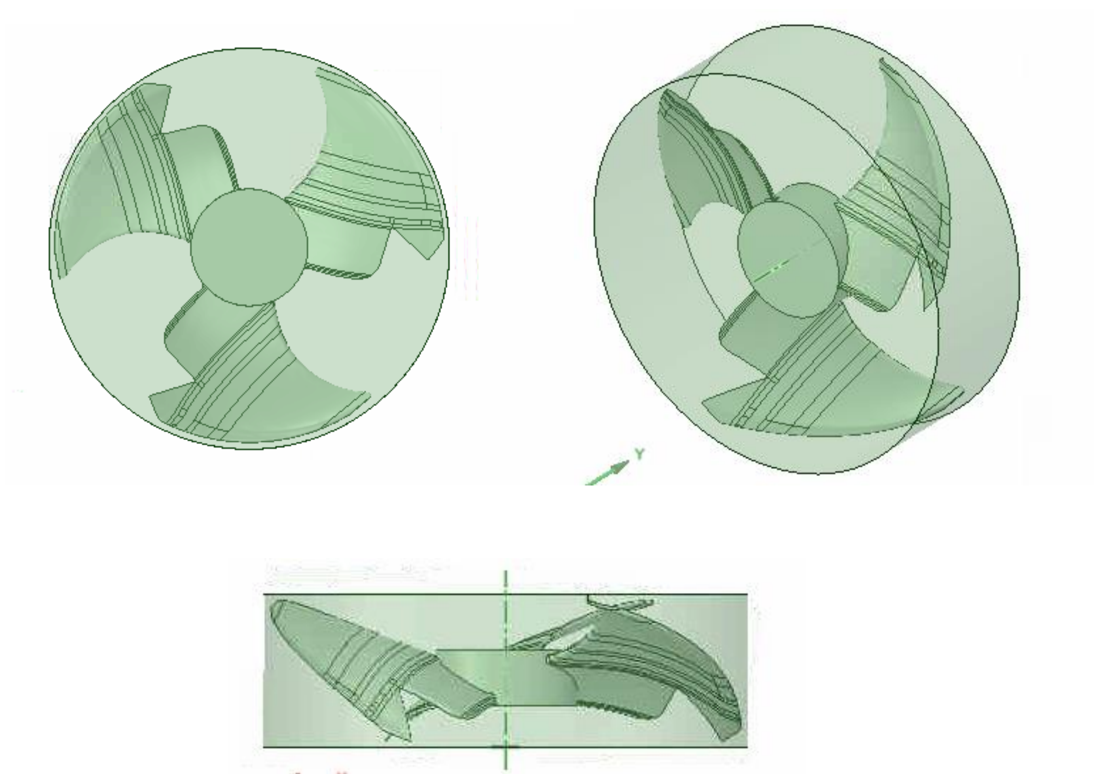
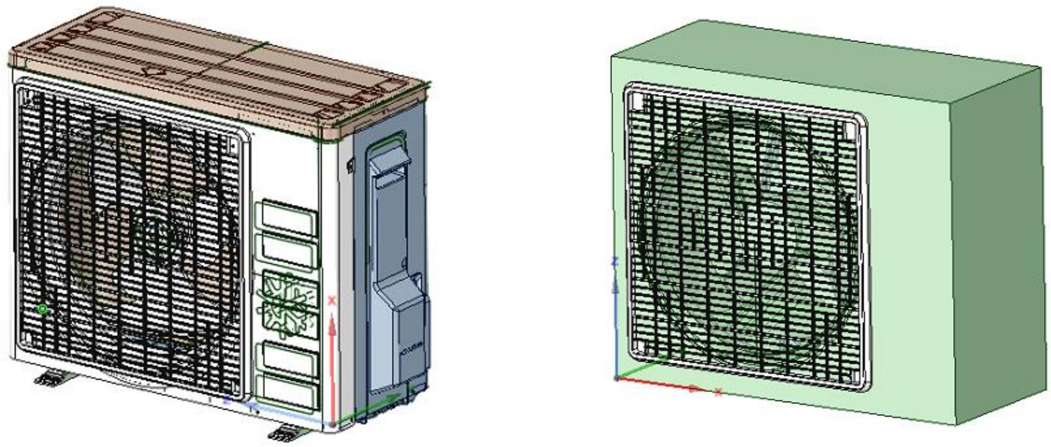


Figure 3.1.3: Multiple Reference Frame (MRF) Zone

Once the simplified CAD geometry is done, it need to convert into STL format using ANSYS SPACECLAIM. Figure 3.1.4 shows the original CAD geometry and simplified CAD geometry.



(a) (b)

Figure 3.1.4: Original CAD geometry and simplified CAD geometry

Stage 2

During second stage, meshing process will be done using SNAPPYHEXMESH from the toolbox of OPENFOAM. OPENFOAM is a free open source CFD software that has object-oriented library for numerical simulations in continuum mechanics written in the C++ programming language. Text files format are used to define the boundary conditions and physical properties of the model that are stored in the folders of OPENFOAM such as blockMesh.dict, snappyHexMesh, decomposePar.dict and so on. To generate mesh using SNAPPYHEXMESH, BLOCKMESH tool are needed to generate a block of mesh that cover the outdoor units. Once BLOCKMESH generate mesh, it will discard the unnecessary feature and produce simple block of mesh and once it completed, SNAPPYHEXMESH will generate the mesh and refine them according to the setting that was set in the snappyHexMesh script file. Figure 3.1.5 below shown the example of text files format used for meshing in OPENFOAM

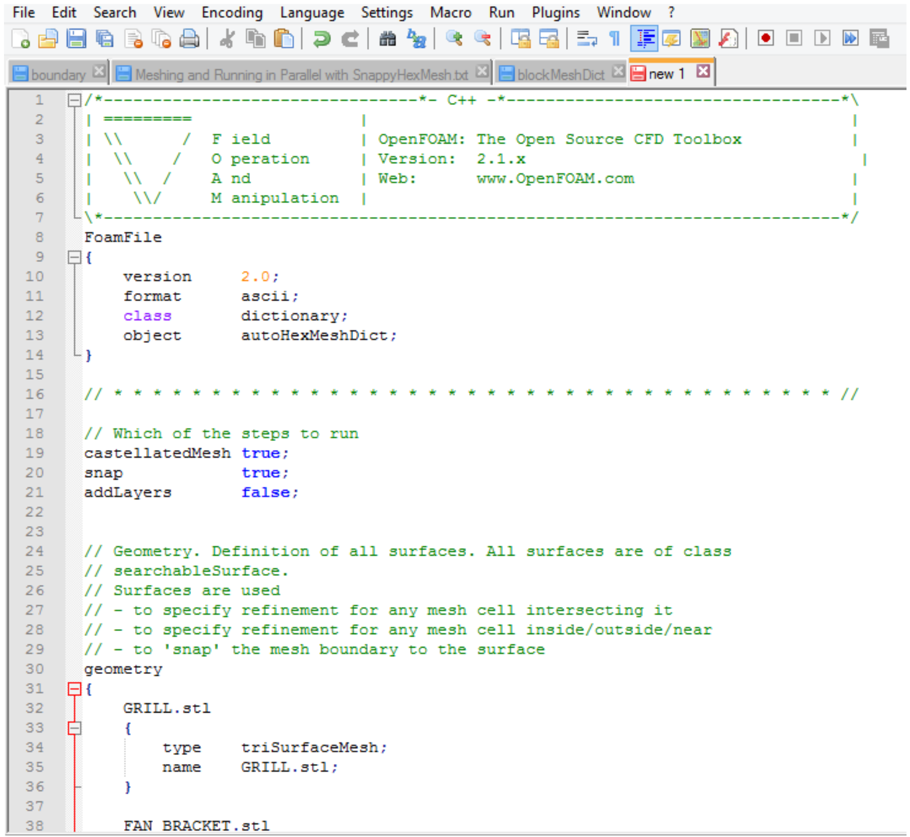


Figure 3.1.5: Script file of OPENFOAM

In OPENFOAM software, there are 3 basic directory structures that are important to run a simulation which are 0 directory, constant directory and system directory. Each directory have important roles and the simulation couldn't proceed without those directory. Figure 3.1.6 shows the basic directory structures that have inside OPENFOAM.

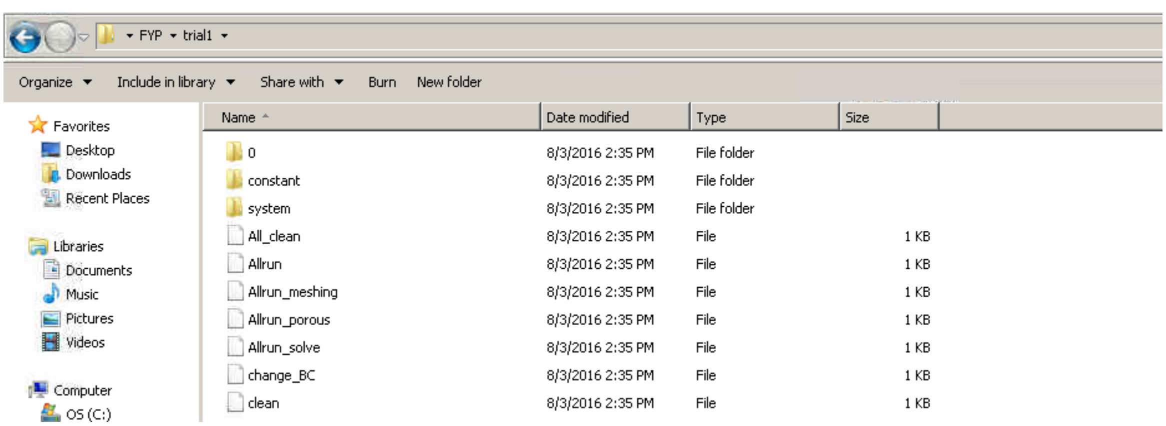


Figure 3.1.6: Directory structures in OPENFOAM

0 directory is to determine the initial boundary condition of the model. For this case, 0 directory consist of U (velocity field file), p (pressure field file), k epsilon (k epsilon field file) and nut (turbulence model) file. For this case study, condenser, grill, propeller, bellmouth, casing, and fan bracket are input inside all of these files and specify the boundary condition of these components. For velocity field file, all the components including the surfaces that were created from BLOCKMESH such as walls_side1, walls_side3, walls_side4, walls_top, walls_bottom and so on will have their own boundary condition. For condenser, casing, fan bracket, propeller, bellmouth, grill, walls_side1, walls_side3, walls_side4, walls_top and walls_bottom will have fixed value type and uniform (0,0,0). This fixed value type means that each components will have constant value while uniform (0,0,0) means the components will have 0 value of velocity during the initial condition. Other components such as atmosphere_side1, atmosphere_side2 and so on that were created in BLOCKMESH are put as zerogradient which means that those components will remain constant throughout the whole simulation. Figure 3.1.7 shows the boundary condition that was set in these files. While in pressure field file, all the components were set as zerogradient except atmosphere_side2 which is put as fixed value type and contain uniform value of 0 in the initial condition. For the other field file such as k-epsilon file and NUT file, all of the components other than atmosphere's surfaces will be input as kqRWallfunction, epsilonWallfunction and nutWallfunction while for atmosphere_side1, atmosphere_side2 and other will be input as zerogradient.

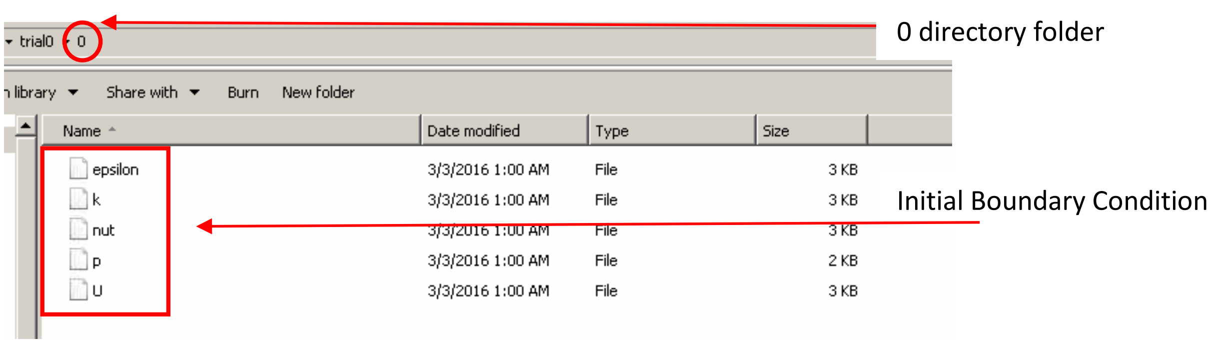


Figure 3.1.7: 0 directory folder

Figure 3.1.8 below shows the example of initial boundary condition script files.

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.x                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    location    "0";
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    #include "${OPENFOAM_INSTALL_PATH}/etc/caseDicts/setConstraintTypes"
    /*
    INLET
    {
        type            cyclicAMI;
        value           uniform ( 0 0 0 );
    }
```

(a)

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.x                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      p;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 2 -2 0 0 0 0];

internalField   uniform 0;

boundaryField
{
       #include "${OPENFOAM_INSTALL_PATH}/etc/caseDicts/setConstraintTypes"
 /*
    INLET
    {
        type            cyclicAMI;
    }
```

(b)

Figure 3.1.8: Example of initial boundary condition script file

Constant directory consist of the CAD drawings in stl format that have already been mesh and put inside one of the folder called triSurface. Figure 3.1.9 and figure 3.1.10 below shows all the files in constant directory where polyMesh consist of boundary condition file, blockMeshDict file and so on. The blockMeshDict script file contain all the information to create a simple block based on fully-structured hexahedral meshes that cover the outdoor unit and wall which this block act as surrounding condition of the case studies. As mention on previous page, BLOCKMESH tool are the first tool to use in mesh generation before it can proceed to SNAPPYHEXMESH. It required information such as the coordinates of the nodes that will form a solid. There can be in rectangular form, square form and pyramid form depends on the coordinates and the sequence that need to input inside blockMeshDict. In this case study, a rectangular block was created as the walls that cover the outdoor unit and the surrounding. Inside vertices, there are the coordinates of each points while the blocks are the place to allow the software to know that which points need to connect on which points to form a solid block. Once the blocks are finish, each boundary will be assigned on each surface.

```
convertToMeters 0.001;

vertices
(
    ( -400 674.299 0) //0
    ( -400 -725.701 0)
    ( 1010 -725.701 0)
    ( 1010 674.299 0)
    ( -400 674.299 1500)
    ( -400 -725.701 1500)
    ( 1010 -725.701 1500)
    ( 1010 674.299 1500)
    ( -800 -725.701 -1000)
    ( -800 -1725.701 -1000)
    ( 1600 -1725.701 -1000)
    ( 1600 -725.701 -1000)
    ( -800 -725.701 2500)
    ( -800 -1725.701 2500)
    ( 1600 -1725.701 2500)
    ( 1600 -725.701 2500)

);

blocks
(
    hex (0 1 2 3 4 5 6 7) stator (60 60 60) simpleGrading (1 1 1)
    hex (8 9 10 11 12 13 14 15) stator (40 96 140) simpleGrading (1 1 1)
);

edges
(
);
```

Figure 3.1.9: Example of BlockMeshDict script file

```
boundary
(
    Walls_side1
    {
        type wall;
        faces
        (
            (0 1 5 4)
        );
    }

    Walls_side2
    {
        type wall;
        faces
        (
            (1 2 6 5)
        );
    }

    Walls_side3
    {
        type wall;
        faces
        (
            (3 7 6 2)
        );
    }

    Walls_side4
    {
        type wall;
        faces
```

Figure 3.1.10: Example of BlockMeshDict script file

System directory consist of all the settings needed to run the simulation including snappyHexMesh, decomposeParDict, surfaceFeatureExtractDict, and so on. SnappyHexMesh is used to create high quality hex-dominant meshes based on geometry that are stored in dictionary constant/trisurface. Inside snappyHexMesh script file, it control all the number of cell, refinement level and mesh quality on every components separately. With snappyHexMesh, OPENFOAM will generate the refine mesh of each components and insert into blockMesh. Besides that, surfaceFeatureExtract function is used to extracts feature edges from the geometry surface using the surfaceFeatureExtract utility and then explicitly specifies the features through the features entry in the snappyHexMeshDict file. There is another function which is called decomposePar which is to run OPENFOAM in parallel on distributed processors. The method of parallel computing used by OPENFOAM is known as domain decomposition, in which the geometry and associated fields are broken into pieces and allocated to separate processors for solution.

Stage 3

Once the mesh element is done, stage 3 will begin with CFD simulation (OPENFOAM). The OPENFOAM solver such as simpleFoam, pimpleFoam, pimpleDyMFoam and so on are used based on the criteria of the case studies. Figure 3.1.11 shows the list of solvers that can be choose based on the criteria of the studies which is under category of incompressible flow.

MRFSimpleFoam	Steady-state solver for incompressible, turbulent flow of non-Newtonian fluids with MRF regions
nonNewtonianIcoFoam	Transient solver for incompressible, laminar flow of non-Newtonian fluids
pimpleDyMFoam	Transient solver for incompressible, flow of Newtonian fluids on a moving mesh using the PIMPLE (merged PISO-SIMPLE) algorithm
pimpleFoam	Large time-step transient solver for incompressible, flow using the PIMPLE (merged PISO-SIMPLE) algorithm
pisoFoam	Transient solver for incompressible flow
porousSimpleFoam	Steady-state solver for incompressible, turbulent flow with implicit or explicit porosity treatment
shallowWaterFoam	Transient solver for inviscid shallow-water equations with rotation
simpleFoam	Steady-state solver for incompressible, turbulent flow
SRFSimpleFoam	Steady-state solver for incompressible, turbulent flow of non-Newtonian fluids in a single rotating frame
windSimpleFoam	Steady-state solver for incompressible, turbulent flow with external source in the momentum equation

Figure 3.1.11: Types of solvers

Inside the system folder, fvSolution is the script file which is the setting for the solver that act as a tool to solve the equation. Since this case study is about incompressible and turbulent flow, simpleFoam is chosen to be the solver of this case study. The table 2.1.1 below shows the simulation that need to be done with different distance of x-axis and y-axis between wall and outdoor unit.

Table 2.1.1: Case studies of CFD simulation

	Wall distance (Y/mm)			
Wall distance (x/mm)	20	60	150	200
20	Case 1	Case 5	Case 9	Case 13
60	Case 2	Case 6	Case 10	Case 14
150	Case 3	Case 7	Case 11	Case 15
200	Case 4	Case 8	Case 12	Case 16

Data are recorded and can be viewed in Paraview which those data will be analyzed and graph will be plotted to determine the relationship between the wall gap and the heat distribution on the outdoor unit. OPENFOAM software is different compare to other CFD software such as ANSYS software, AcuSolve and so on because OPENFOAM operate using scripting format where user need to type the command inside command prompt to execute the function inside OPENFOAM. It don't have guided user interface to show user about all the option and function. There are some example of command that execute the function that can be seen in table 2.1.2 below. The purpose of "runApplication" is to save the log data so that the user can review the data easily.

Table 2.1.2: Type of commands

Command	Function
runApplication blockMesh	Execute the BLOCKMESH tool according to the settings from blockMeshDict and save log file into OPENFOAM
runApplication surfaceFeatureExtract	Execute surfaceFeatureExtract to extracts feature edges from the geometry surface and save log file into OPENFOAM
runApplication snappyHexMesh	Execute SNAPPYHEXMESH tool to generate mesh and save the log file into OPENFOAM
runApplication checkMesh	To check the condition of mesh after mesh generation
runApplication simpleFoam	Execute the solvers

Figure 3.1.12 to Figure 3.1.14 shows parts of the scripts of each command while figure 3.1.15 shows the command prompt.

```
Casing.stl
{
    // How to obtain raw features (extractFromFile || extractFromSurface)
    extractionMethod    extractFromSurface;

    extractFromSurfaceCoeffs
    {
        // Mark edges whose adjacent surface normals are at an angle less
        // than includedAngle as features
        // - 0  : selects no edges
        // - 180: selects all edges
        includedAngle   150;
    }

    // Write options

        // Write features to obj format for postprocessing
        writeObj                yes;
}
/*
Grill.stl
{
    // How to obtain raw features (extractFromFile || extractFromSurface)
    extractionMethod    extractFromSurface;

    extractFromSurfaceCoeffs
    {
        // Mark edges whose adjacent surface normals are at an angle less
        // than includedAngle as features
        // - 0  : selects no edges
        // - 180: selects all edges
        includedAngle   150;
```

Figure 3.1.12: Example of snappyHexMesh script file

```
// Settings for the castellatedMesh generation.
castellatedMeshControls
    {

    // Refinement parameters
    // ~~~~~~~~~~~~~~~~~~~~~

    // While refining maximum number of cells per processor. This is basically
    // the number of cells that fit on a processor. If you choose this too small
    // it will do just more refinement iterations to obtain a similar mesh.
    maxLocalCells 600000;

    // Overall cell limit (approximately). Refinement will stop immediately
    // upon reaching this number so a refinement level might not complete.
    // Note that this is the number of cells before removing the part which
    // is not 'visible' from the keepPoint. The final number of cells might
    // actually be a lot less.
    maxGlobalCells 100000000;

    // The surface refinement loop might spend lots of iterations
    // refining just a few cells. This setting will cause refinement
    // to stop if <= minimumRefine are selected for refinement. Note:
    // it will at least do one iteration (unless the number of cells
    // to refine is 0)
    minRefinementCells 10;

    // Number of buffer layers between different levels.
    // 1 means normal 2:1 refinement restriction, larger means slower
    // refinement.
    nCellsBetweenLevels 2;
```

Figure 3.1.13: Example of surfaceFeature script file

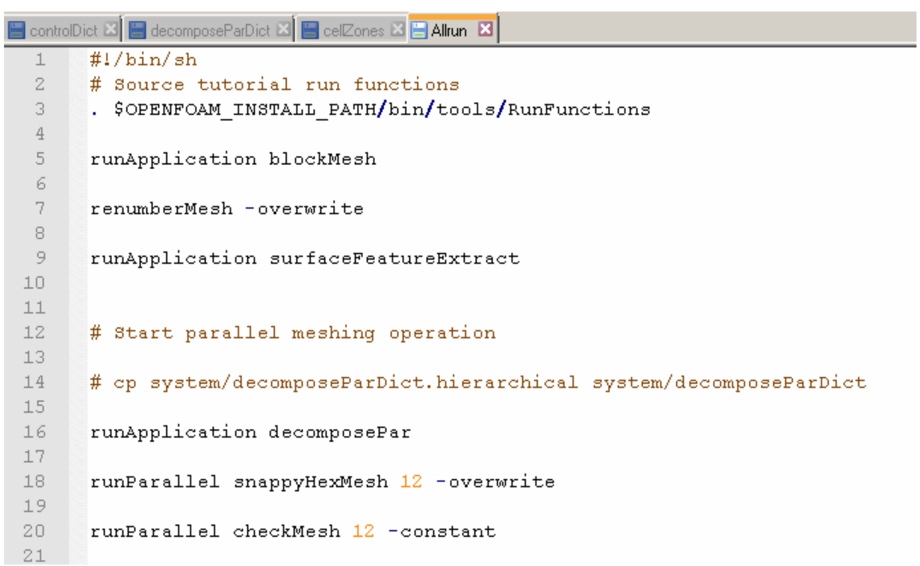
```
application     simpleFoam;

// application     porousSimpleFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         8000;

deltaT          1;

writeControl    timeStep;

writeInterval   30;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;

functions
```

Figure 3.1.14: Example of simpleFoam script file

```
/cygdrive/c/Documents and Settings/Hiar

Hian@A43S ~
$ cd /cygdrive/c/Documents\ and\ Settings/Hian/Desktop/calibrate_3.0_SimpleFoam/

Hian@A43S /cygdrive/c/Documents and Settings/Hian/Desktop/calibrate_3.0_SimpleFoam
$ blockMesh
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.3.x                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
/*  Windows port by CFD support (www.cfdsupport.com) [based on Symscape]     *\
\*---------------------------------------------------------------------------*/
Build  : 2.3.x-819030ed51bd
Exec   : D:\OpenFOAM\cygwin64\opt\OpenFOAM\OpenFOAM-2.3.x\platforms\cygwin64mingw-w64DPOpt\bin\block
Date   : Apr 10 2016
Time   : 10:57:05
Host   : "A43S"
PID    : 4300
Case   : C:/Documents and Settings/Hian/Desktop/calibrate_3.0_SimpleFoam
nProcs : 1
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Creating block mesh from
    "C:/Documents and Settings/Hian/Desktop/calibrate_3.0_SimpleFoam/constant/polyMesh/blockMeshDict
Creating curved edges
Creating topology blocks
```

Figure 3.1.15: Command Prompt

From meshing until CFD simulation in OPENFOAM software, all the process are done by typing function into the command prompt. In this project, there is one script file called All_run which is named by the user and it contain all the command written inside the file so the user just type one keyword to activate the whole process. The command must be in sequence from the first line until the final line so that the software will execute one by one from the first line to the end of the script. Figure 3.1.16 shows the script file inside All_run where the first command is blockMesh follow by renumberMesh which this function is to renumbers the cell list in order to reduce the bandwidth, reading and renumbering all fields from all the time directories. Next is surfaceExtractFeature then decomposePar to distributed the geometry into multiple pieces based on the number of processor that the computer have so that the software can simulate faster simultaneiously. After that, the snappyHexMesh function will be execute to mesh the geometry followed by checkMesh function to check the quality of the mesh. Once the mesh is good, it is ready for CFD simulation where the command simpleFoam is needed to execute the simulation. Once simulation is done, the distributed mesh needed to reconstruct to become the full set of mesh. To do that, reconstructPar function comes in handy to combine all the mesh so that the data is complete.




```
#!/bin/sh
# Source tutorial run functions
. $OPENFOAM_INSTALL_PATH/bin/tools/RunFunctions

runApplication blockMesh

renumberMesh -overwrite

runApplication surfaceFeatureExtract


# Start parallel meshing operation

# cp system/decomposeParDict.hierarchical system/decomposeParDict

runApplication decomposePar

runParallel snappyHexMesh 12 -overwrite

runParallel checkMesh 12 -constant

```

Figure 3.1.16: Example of All_run script file

Experiment

Experiment of different wall gap between 5SLY15F outdoor unit and wall is conducted to determine the rotational speed (rpm) of DC fan motor and AC fan motor. This experiment is installed in one of the test room in Daikin Research & Development (DRDM) Sdn. Bhd. The setup of the experiment are shown in figure 3.1.17 and figure 3.1.18 below. The experiment are conducted with placing the outdoor unit near the walls in X-axis and Y-axis and increasing 10mm each time after the data have been recorded using stroboscope. The data of rpm will then insert into excel and graph is generate to see the pattern of the relationship of the distance between wall and outdoor unit. Figure 3.1.19 shows the equipment (stroboscope) used in this experiment.

Figure 3.1.17: Airflow study Experiment Setup

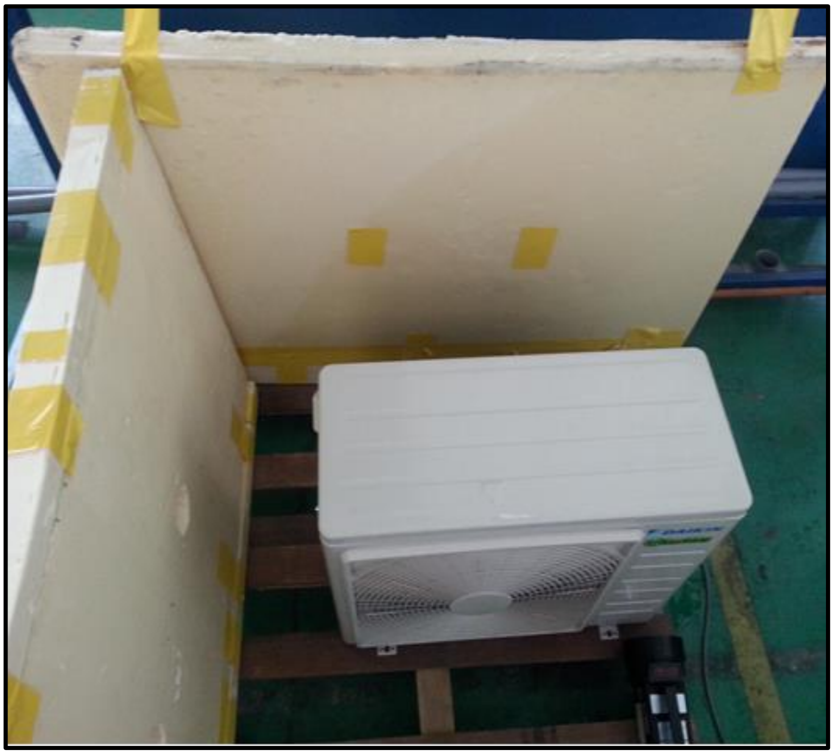


Figure 3.1.18: Airflow study experiment setup

Figure 3.1.19: Stroboscope

3.2 Flow Chart

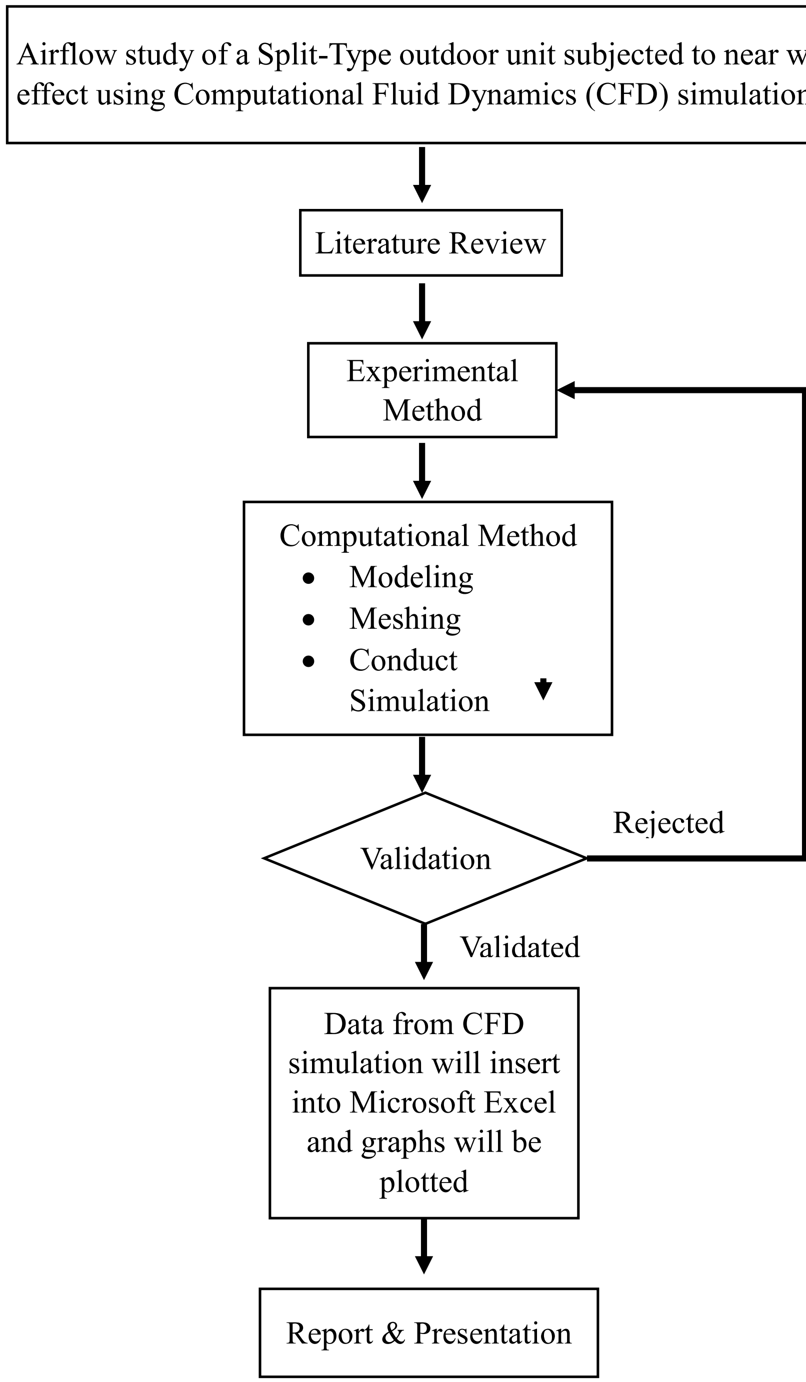


Figure 3.2.1: Flow Chart

3.3 Key Milestone

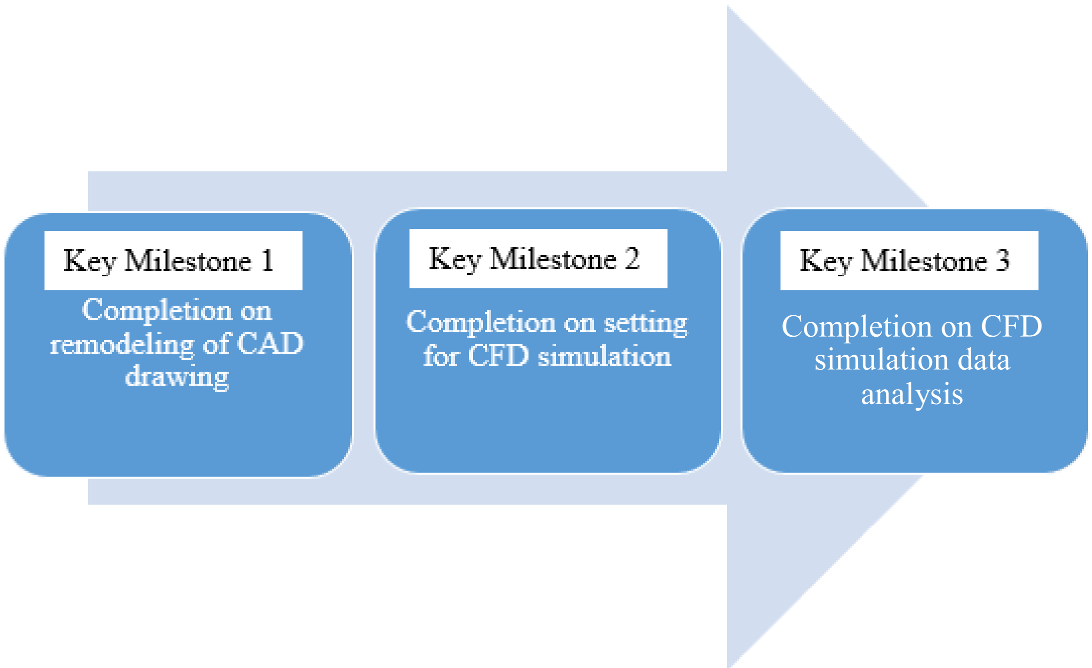


Figure 3.3.1: Key Milestone

3.4 Gantt Chart

Table 3.1: Gantt chart

No.	Activities	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28
1	Selection of Project Topic																												
2	Understanding Project Topic and Gathering information																												
3	Reconstruction and simplification progress of CAD drawing (5SLY15F)														①														
4	Settings for CFD simulation in OPENFOAM																			②									
5	CFD simulation																												
6	Analysis and presentation of results																										③		
7	Preparation of technical report for technical paper publication																												
8	Submission of final Report																												
9	Viva																												

FYP 1

FYP 2

① ② ③ Key Milestones

Chapter 4
Results and Discussion

During FYP 1, the components of outdoor unit of model 5SLY15F are collected for experiment purpose to get the data that are needed for verification with the simulation results. Figure 4.0.1 below shows the outdoor unit of model 5SLY15F.

Figure 4.0.1: Outdoor unit of 5SLY15F

For numerical analysis, the CAD drawing of outdoor unit of 5SLY15F had to be simplified and some components are needed to reconstruct to get better quality of mesh which shown in figure 4.0.2. Once the simplified model is done, mesh generation can be done in stage 2 and CFD simulation in stage 3.

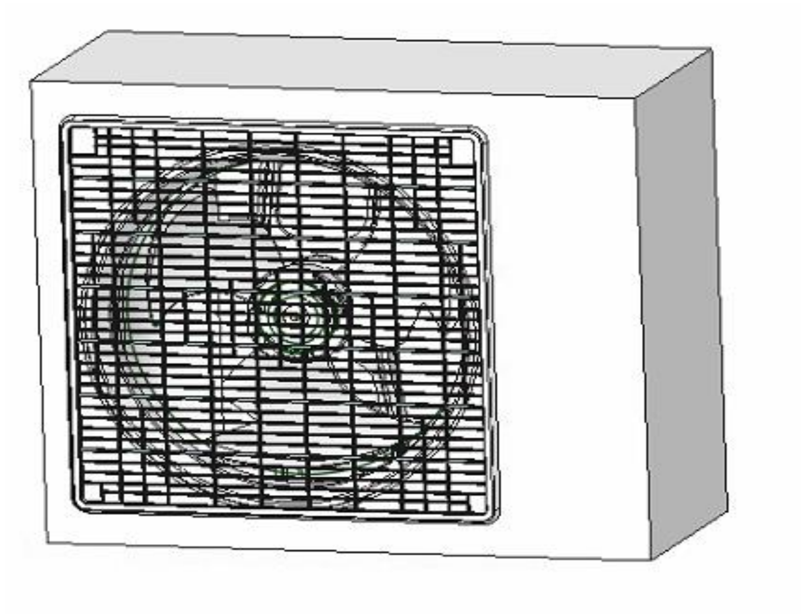


Figure 4.0.2: CAD drawing of outdoor unit of 5SLY15F

Due to some geometry problem occur in the components such as propeller and grill components, reconstruction process need to be done on those components by connecting surface by surface to form a solid components which consume a lot of time to do this part. This process can be shown in figure 4.0.3 which is already reconstructed.

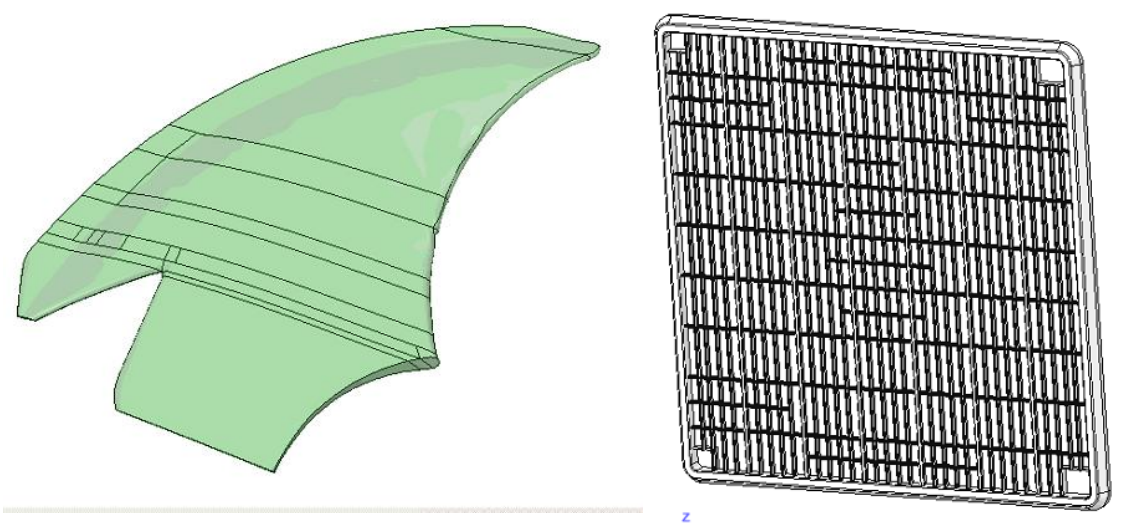
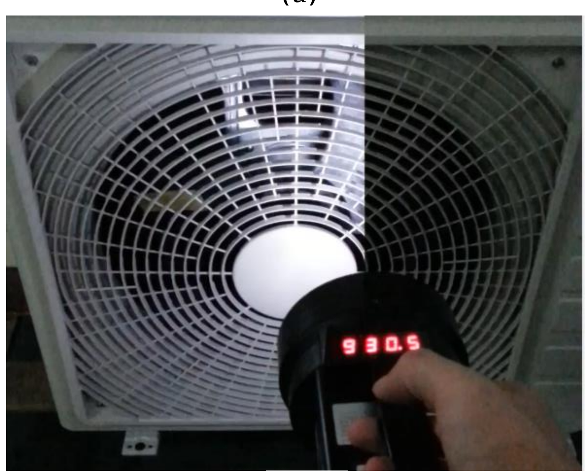


Figure 4.0.3: Reconstruction of CAD geometry

4.1 Experiment Results

There are two experiments that have been conducted to verify with the simulation and the data that were collected from the experiment are the rotational speed (rpm) of DC and AC fan motor inside the outdoor unit where the data was obtain using stroboscope on each wall gap distance between the walls and outdoor unit. A piece of yellow tape was put on the surface of fan blade as a reference point to measure the rpm of the fan, which is shown in figure 4.1.1 below. Since the stroboscope is used to stop motion for diagnostic inspection purposes. It can read the rpm of the fan blade when the rotation of the blade is reach a certain point where the fan looks static.

(a)



(b)

Figure 4.1.1: Method of rpm measurement

For the outdoor unit that installed with DC motor, it is equipped with DC power supply controller to turn on the motor only without turning on other components such as compressor. This Figure 4.1.2 shows the experiment setup which is equipped with DC power controller.

(b)

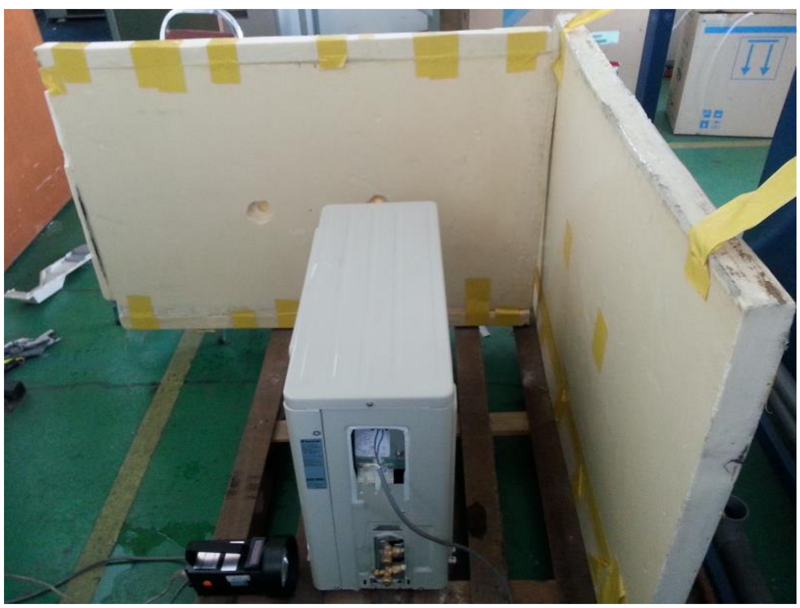


Figure 4.1.2: Outdoor unit with near wall experiment setup

The data that was gathered from the experiment are then imported into excel program and plot graphs to see the difference between DC fan motor and AC fan motor. Based on the graph plot shown in figure 4.1.3, DC fan motor considered to have constant rpm because it has smaller standard deviation compared to AC fan motor which is 2.45 and 5.03126. The data that were recorded are within the range of 918 rpm and 926 rpm which is lower than the nominal rpm that was set on the outdoor unit (930 rpm). AC fan motor has the range of 925 rpm to 946 rpm and its standard deviation is higher compared to DC fan motor. The reason DC fan motor have constant rpm is because DC fan motor use direct current electrical power to convert into mechanical power. Since DC motor have constant voltage, it will rotate on same rpm throughout the whole process based on the specification of the motor. Based on the experiment, even though the distance of wall gap is shorten, the DC motor is still running on the constant rpm but it required more power consumption to maintain the rotational speed as it consume more current. This is because the shorter the distance of wall gap, the higher the static pressure and the air flow rate going through the condenser become lower. This will affect the rotational speed of a motor. But DC motor have to rotate on the nominal rotational speed that have been set so when it detect the changes of rotational speed, it require higher current to maintain the rotational speed.

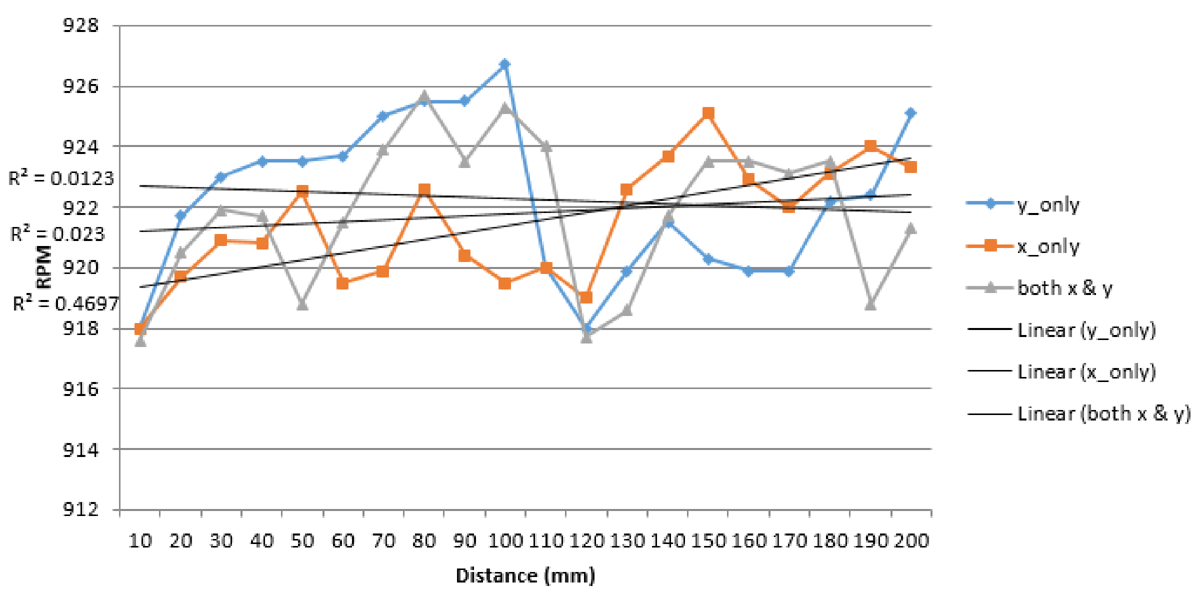


Figure 4.1.3: Graph of RPM versus distance of wall gap (DC motor)

For AC fan motor, it used alternating current electrical energy to convert into mechanical energy. It is equipped with a control panel that control the power consumption. So when the distance of wall gap changes, the rpm will change according to the distance to reach the constant power consumption. Figure 4.1.4 shows the graph of distance versus rotational speed of AC motor.

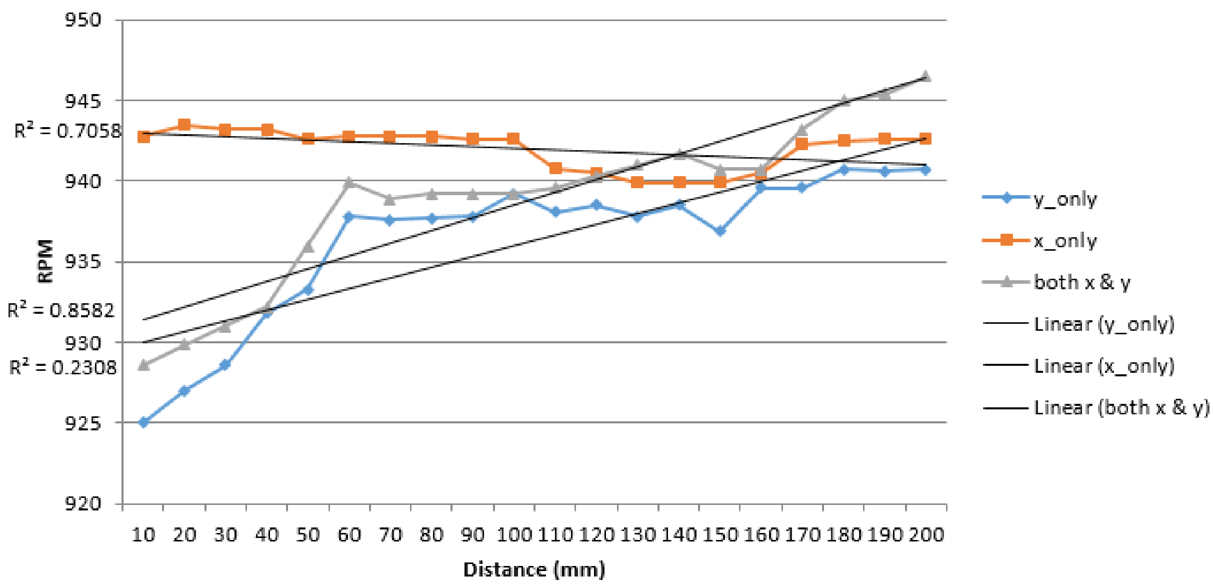


Figure 4.1.4: Graph of RPM versus distance of wall gap (AC motor)

With the data that were obtain from the experiment, it will verify the assumption of constant rpm on the fan motor which AC motor can be used because of the constant rpm through different distance of wall gap. With this verification, the rpm can be set for CFD simulation to find the air flow rate (CFM) that are the results of the simulation.

4.2 CFD Simulation

Once those rpm data have been analyze, CFD simulation can be run on those cases based on DC fan motor since it has constant rpm. With the constant rpm, heat distribution around the condenser will be the responding variable while distance of wall gap between wall and outdoor unit will become the manipulating variable. There are total of 16 cases needed to be simulate and 16 times of simulation are running at the same time. With limited resources available and some difficulty that occur, there are 10 cases that have been done while other are still on progress simulation. Table 4.1 shows the total cases for simulation with different distance.

Table 4.1: Case Studies

	Wall distance (Y/mm)			
Wall distance (x/mm)	20	60	150	200
20	Case 1	Case 5	Case 9	Case 13
60	Case 2	Case 6	Case 10	Case 14
150	Case 3	Case 7	Case 11	Case 15
200	Case 4	Case 8	Case 12	Case 16

	X (mm)	Y (mm)
Case 1	20	20
Case 2	60	20
Case 3	150	20
Case 4	200	20
Case 5	20	60
Case 6	60	60
Case 7	150	60
Case 8	200	60
Case 9	20	150
Case 10	60	150
Case 11	150	150
Case 12	200	150
Case 13	20	200
Case 14	60	200
Case 15	150	200
Case 16	200	200

In each cases, the distance are different such the distance increase from case 1 to 16. In each cases, different distance of X-axis and Y-axis was set and this distance can be changed in blockMeshDict file from 0 directory folder. To create a blockMesh, the coordinates are required to input. So with different cases, the coordinates have been set according to the distance of wall gap to create the exact scenario inside the simulation. Other files that are needed inside 0 directory, constant directory and system directory are remain the unchanged since the settings are the same throughout all the cases. Figure 4.2.1 shows the blockMesh generated on case 16.

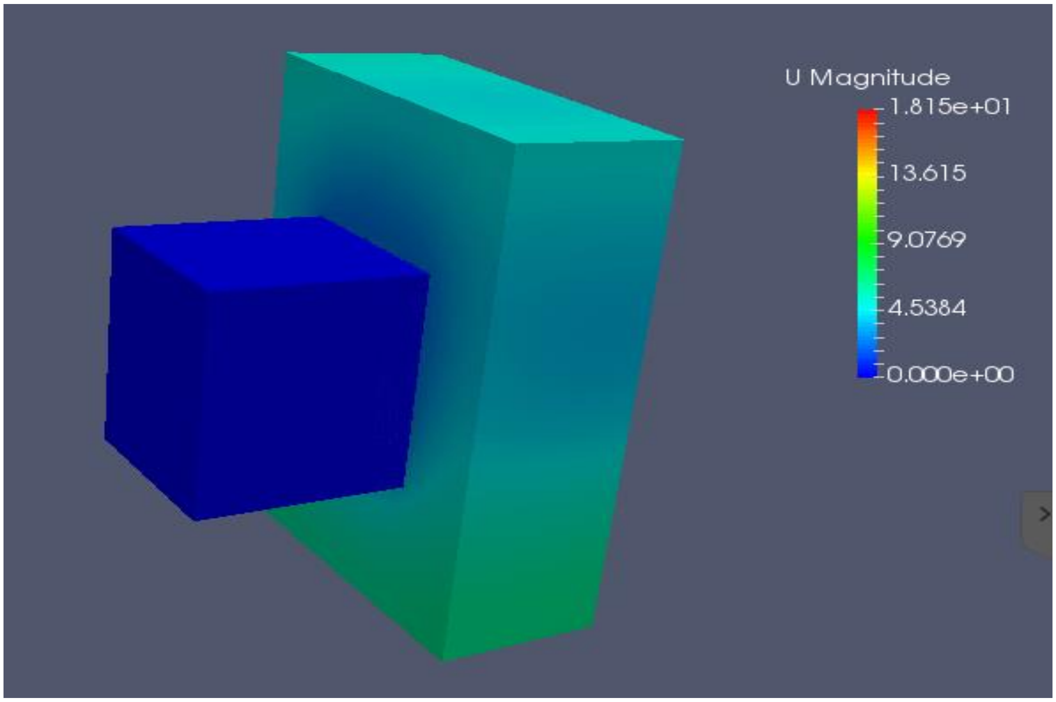


Figure 4.2.1: BlockMesh

<u>Result of air flow rate and pressure</u>

From these simulation that were done, air flow rate and pressure drop can be obtained near the condenser area since the objectives of this study is to study the flow distribution that was influence by the distance. So with the changes of distance, the flow distribution will be interrupt. The data are obtained using Paraview software which this software is a multi-platform data analysis and visualization application. User can use Paraview to build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities. Once those simulation are done, the data can be view using Paraview as shown in figure 4.2.2.

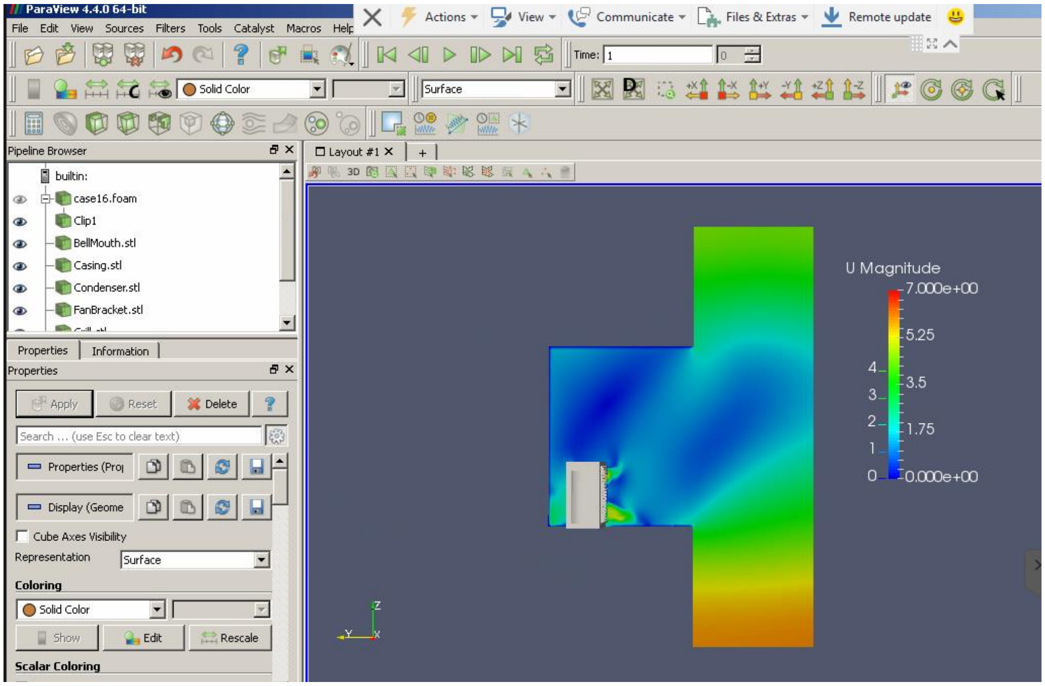


Figure 4.2.2: Paraview

The purpose of getting the air flow rate and pressure drop at the area before the air passing through the condenser is because that area determine the condition of air where it will pass through the condenser and influence the performance of air conditioning. To measure the air flow rate and pressure, a few tools needed in Paraview. First, the block will be clip to cover the area of condenser as shown in figure 4.2.3 using clip tool.

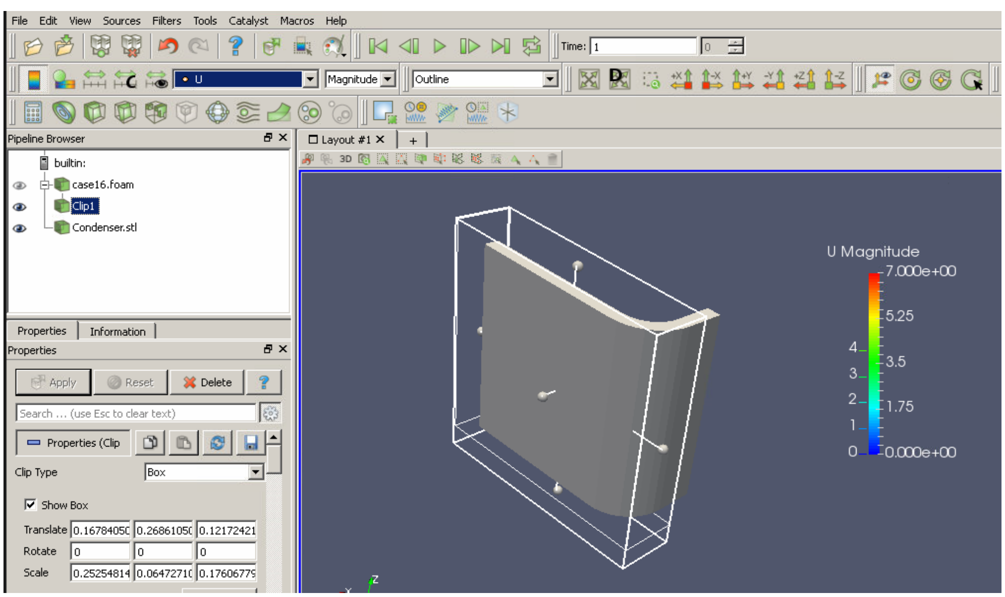


Figure 4.2.3: Clip Function

The purpose of clip is to allow slice tool to be used so that it can get the surface of the condenser. Figure 4.2.4 shows the example of slice tool.

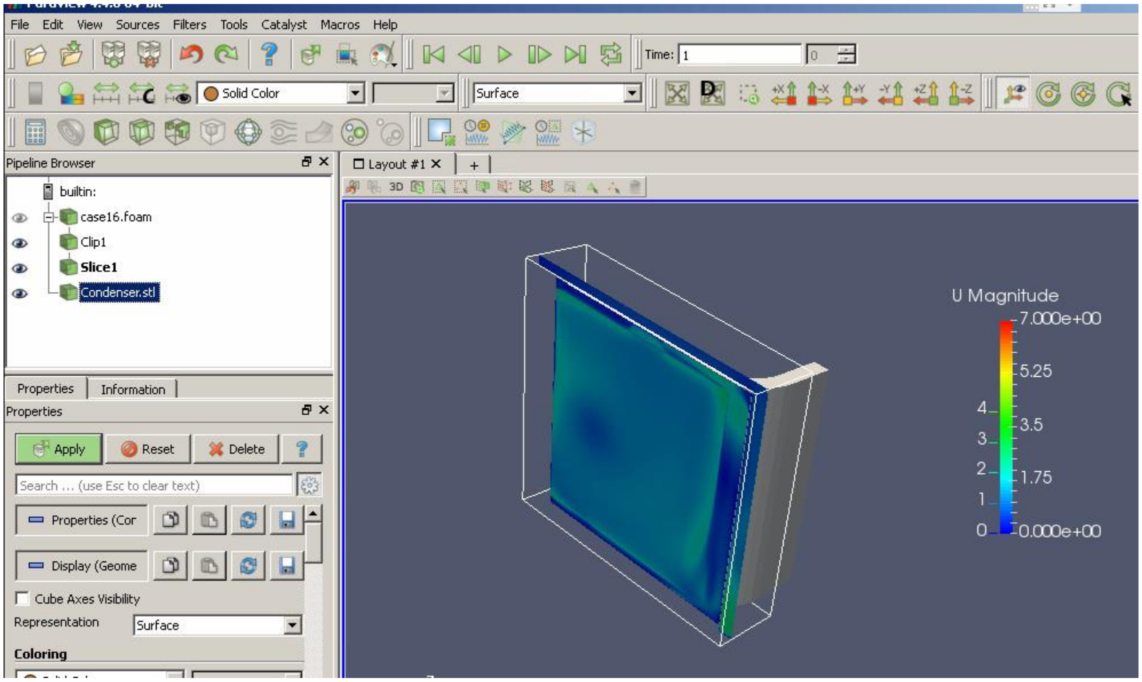


Figure 4.2.4: Slice Function

Once the surface was obtained, calculator tool is used to calculate the magnitude of velocity multiply the area of the surface to find the air flow rate that can seen in figure 4.2.5. Calculator is not needed to find pressure because paraview can view the pressure drop data since OPENFOAM already calculate the pressure drop during simulation. Once calculated air flow rate and pressure drop, descriptive statistic is used to view the data of air flow rate and pressure drop as shown in figure 4.2.6.

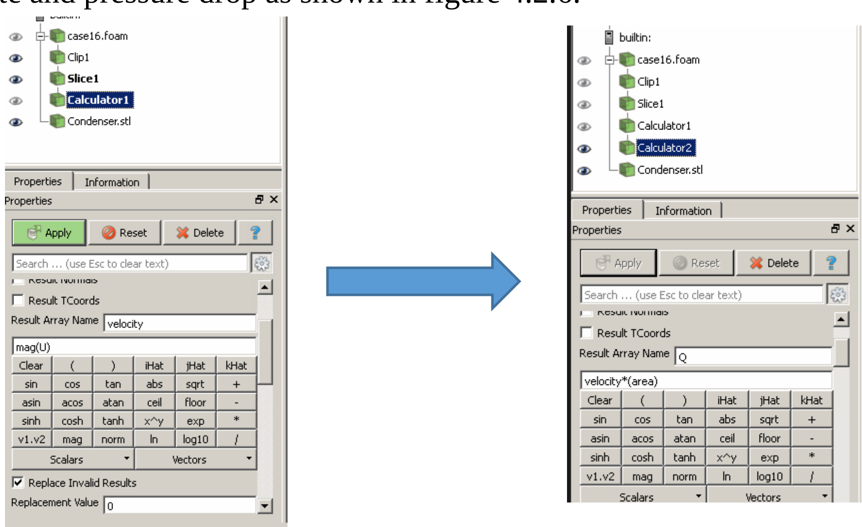


Figure 4.2.5: Calculation Function

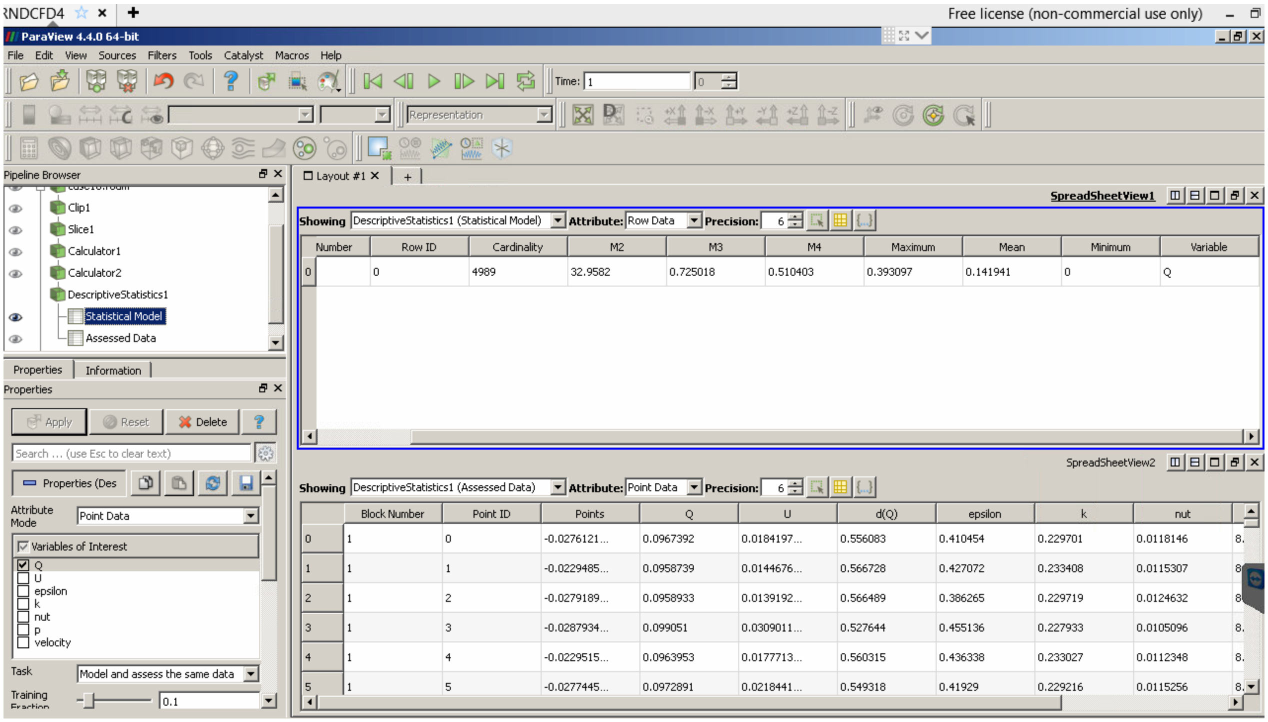


Figure 4.2.6: Descriptive Statisctic

Once all the mean value of air flow rate, Q were obtained, these data will input inside excel and a graph of air flow rate, pressure and the distance is plotted.

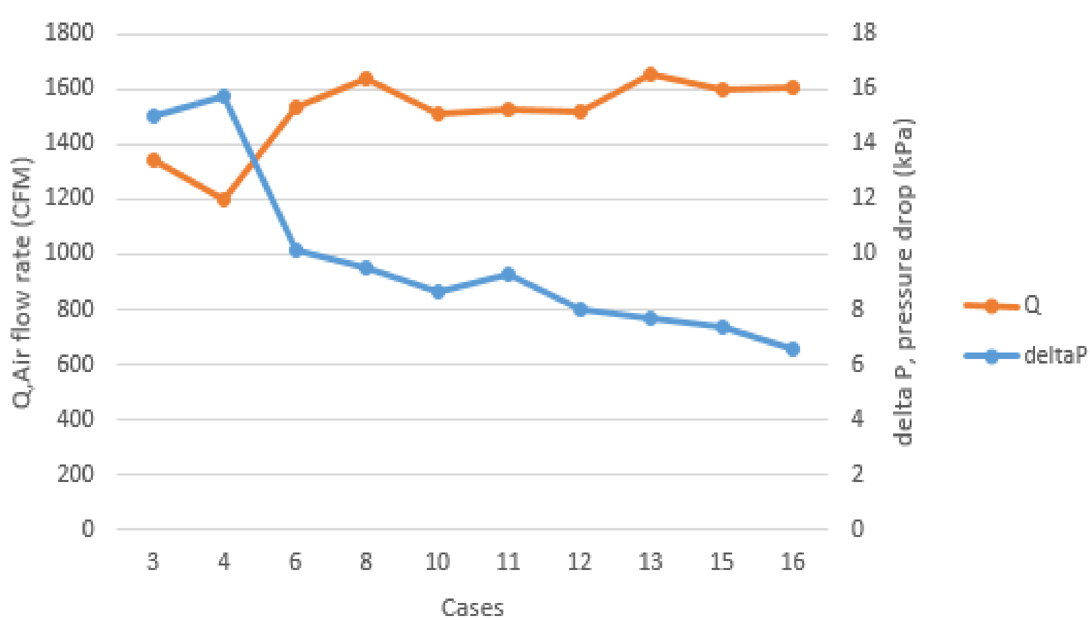


Figure 4.2.7: Data of Air flow rate and velocity in different cases

From the cases that have been simulated, a graph of air flow rate versus pressure have been plot. According to the graph, case 13 have the highest CFM (1653.287) and lower pressure (7.35kPa) compared to other cases. Case 13 have the distance of 200mm toward y-axis and 20mm toward x-axis. From the graph, case 1 which is the closest distance of wall gap between wall and outdoor unit and the CFM is lowest and pressure is high. This is because of the static pressure that occur when the wall gap is short distance. In case 1, the temperature will increase due to low heat distribution caused by low CFM. As the distance of the wall gap increase, the CFM increase and pressure decrease. From case 6, the distance of wall gap is 60mm x-axis and 60mm y-axis and the CFM is the same as case 10, 11 and 12. This mean that the minimum distance of wall gap that is suitable for installation of outdoor unit is starting from 60mm.

There is another analysis that was obtained from Paraview which is the variance of the flow. Variance is to show how far a set of number are spread out. The higher the variance, the higher the range of the set of number to be different. A variance of zero shows that all the values are identical inside a set of data. To obtain the data from Paraview, a same method was done which are to clip the condenser area and slice out the surface. Once the surface is obtained, descriptive statistic will show a set of variance data and these data will be insert into excel and graph of variance versus the distance of wall gap will be plotted. Figure 4.2.8 shows the set of variance number and graph of variance versus distance of wall gap that were assigned into different cases.

	variance	
Case	y	x
3	9.501483	0.983703
4	6.994858	1.196845
6	3.380558	6.071373
10	0.687293	1.101298
11	0.906138	0.439783
12	1.492125	0.400198
13	0.882708	2.30054
15	0.49755	0.280575
16	0.820878	0.698653

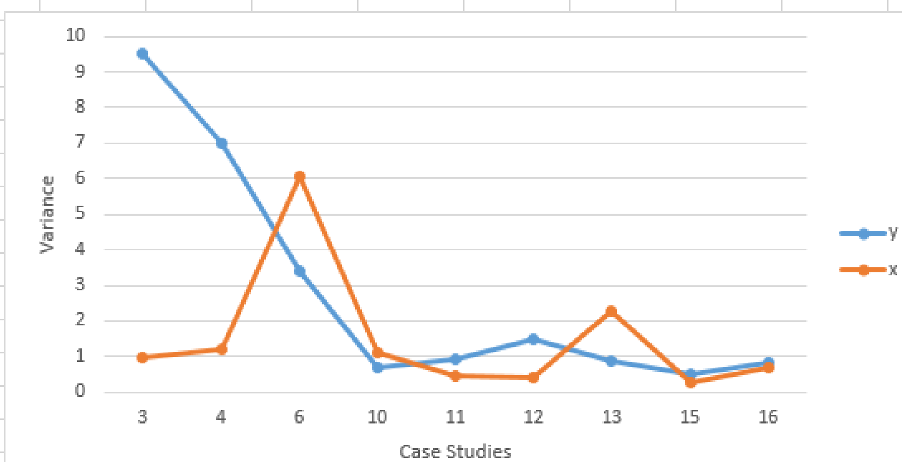


Figure 4.2.8: Data of variance in different cases with plotted graph

Based on the graph that shown above, there are two set of variance data in different cases which are in y axis and x axis. These is obtained in Paraview that can be seen in figure 4.2.9 that there are y axis surface and x axis surface that are obtained to verify the results.

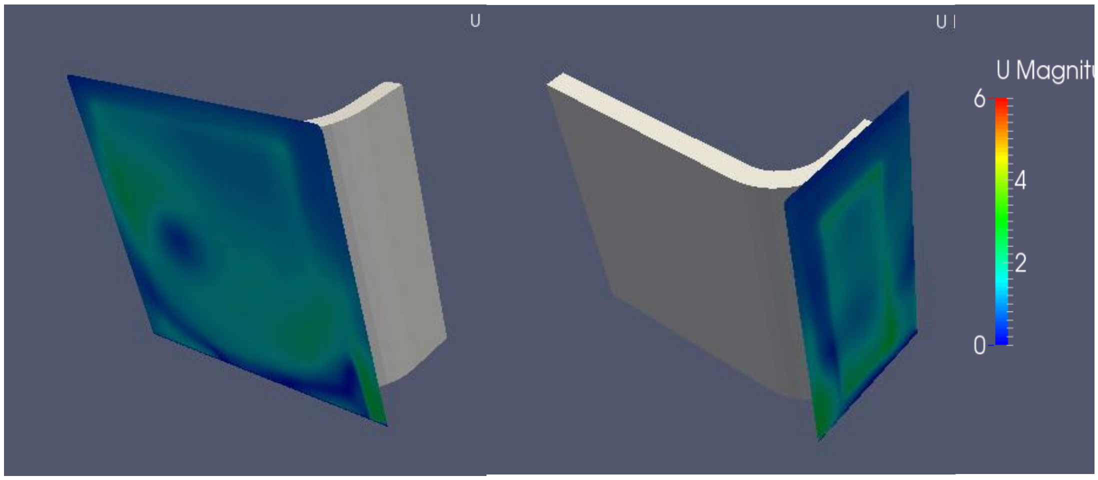
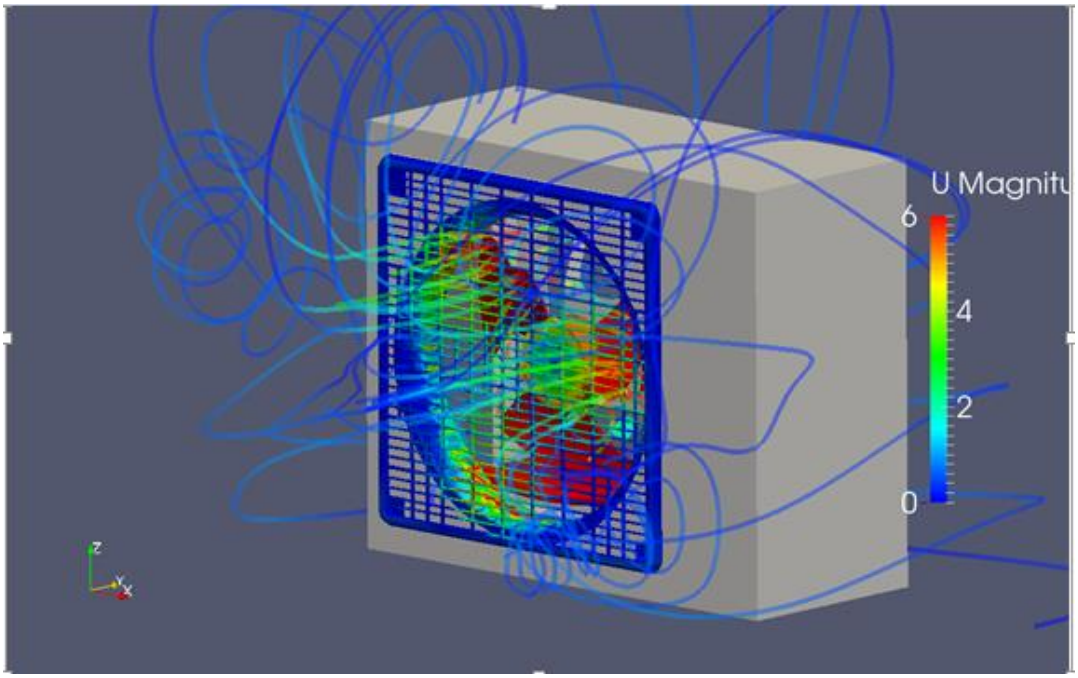


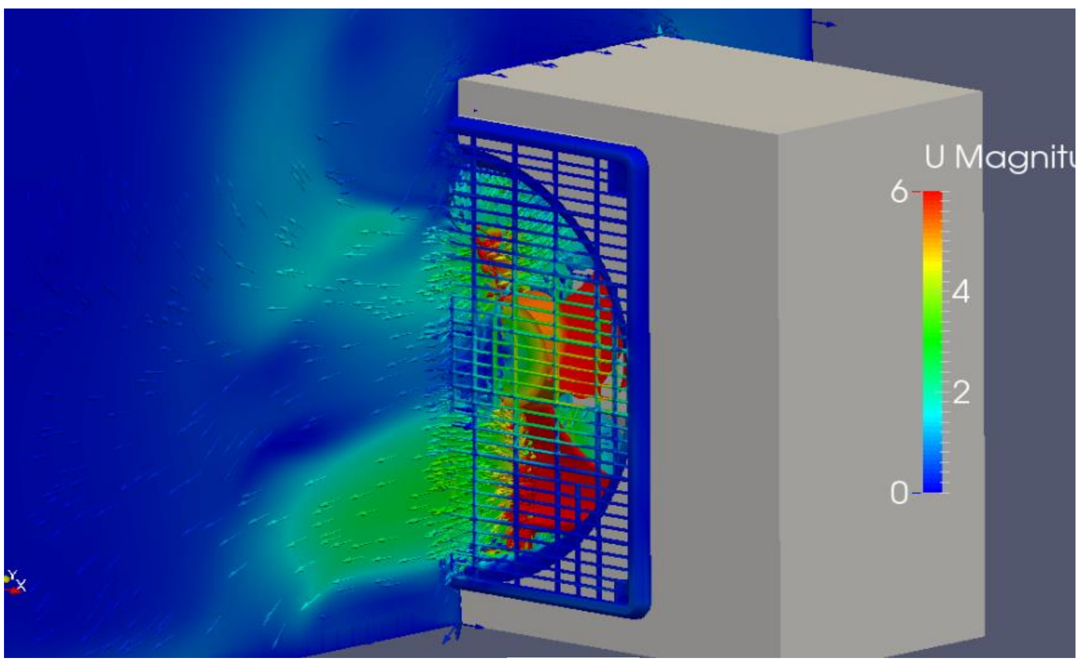
Figure 4.2.9: Y-axis and X-axis surface to obtain data

From the graph above, the variance from y –axis shows the higher the distance of wall gap, the lower the variance as the distance of wall gap increase from case 3 to case 16. But from x – axis, the variance fluctuate because in different cases, the distance in x- axis is vary with the y axis for example, case 1 to 4, the distance at x- axis changes from 20, 60, 150 and 200 while distance at y – axis remain 20. Starting from case 5, the distance at x-axis repeat from 20 again but the distance at y –axis change to 60. So based on the graph, the variance increase during 4 cases and decrease at the 5th case.

The velocity contour of the outdoor unit and walls can be seen in figure 4.2.10 and figure 4.2.11.

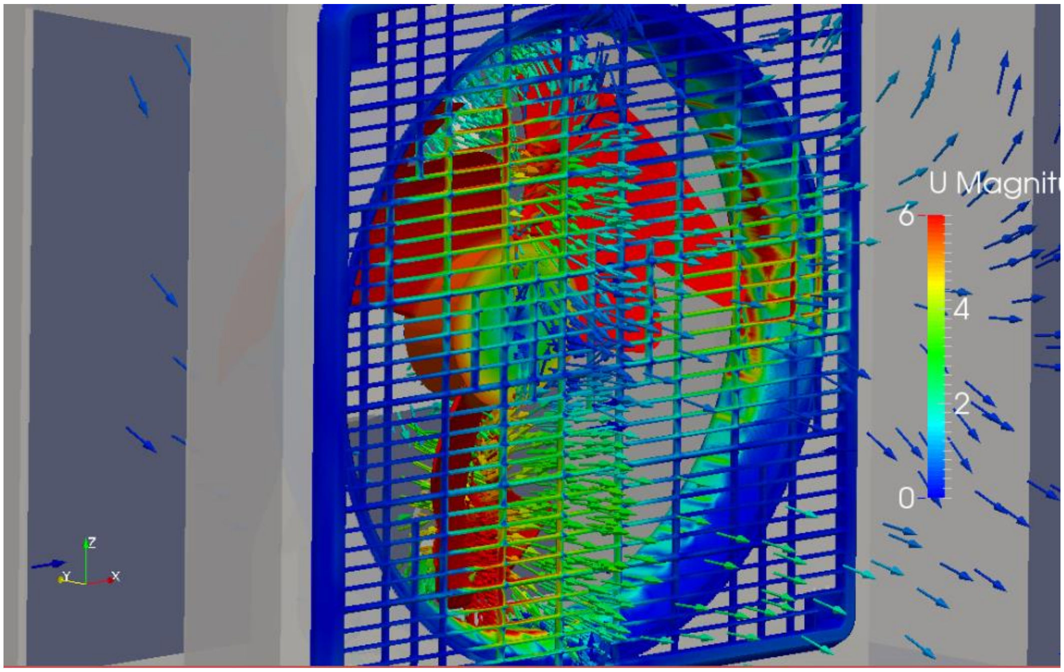


(a)

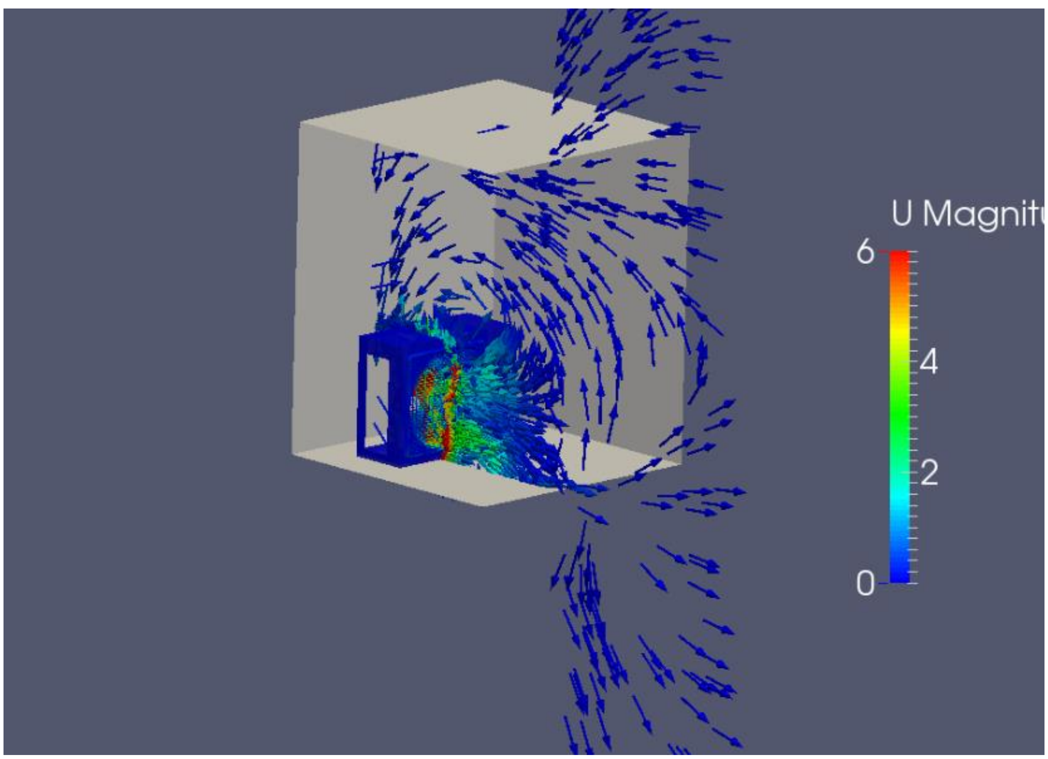


(b)

Figure 4.2.10: Velocity Contour



(a)



(b)

Figure 4.2.11: Velocity Contour

Chapter 5

Conclusion and Recommendation

The distance of wall gap between wall and outdoor unit affect the heat distribution of heat exchanger which is the condenser and hence will affect the performance of air conditioning unit. The shorter the distance of wall gap between wall and outdoor unit, the higher the static pressure generated on that area. When high static pressure occur there, air will become difficult to passing through the wall gap and hence the amount of air will become less. In this project, the objectives have been achieved which are to investigate the relationship between air flow rates (CFM) and the distance of wall gap between wall and outdoor unit. It is found that case 1 have the lowest CFM and high pressure but when the distance increases, CFM increase and pressure decrease. Based on the graph shown in figure 4.2.8, case 13 has the highest CFM with the distance of 20mm in x-axis and 200mm in y-axis. Besides that, starting from case 6, the CFM across the condenser become constant. So the minimum installation space of an outdoor unit is 60mm in x and y-axis.

Due to limited time and limited resources to conduct the study, there are only few cases that have done and further numerical work has to be conducted to concretely prove the hypothesis of this project. Never the less, statistical method should be implied on this project to strengthen the numerical results that were obtained from CFD simulation. For further studies, more detailed numerical and experimental work need to be conducted where the range of distance of wall gap need to be narrow down.

Reference

[1] *HEATING, VENTILATION, AIR CONDITIONING (HVAC)*. p.^pp. Pages.

[2] A. Michel, E. Bush, J. Nipkow, C. U. Brunner, and H. Bo, "Energy efficient room air conditioners–best available technology (BAT)."

[3] A. Avara and E. Daneshgar, "Optimum placement of condensing units of split-type air-conditioners by numerical simulation," *Energy and Buildings,* vol. 40, pp. 1268-1272, // 2008.

[4] Y. A. Çengel, *Thermodynamics: an engineering approach* vol. 7th SI 7th SI. Singapore;London;: McGraw-Hill.

[5] *INSTALLATION MANUAL - Daikin AC*. Available: http://www.daikinac.com/content/assets/DOC/InstallationManuals/RXS09_12LVJU%20Installation%20Manual.pdf

[6] T. T. Chow, Z. Lin, and Q. W. Wang, "Effect of building re-entrant shape on performance of air-cooled condensing units," *Energy and Buildings,* vol. 32, pp. 143-152, 7// 2000.

[7] V. Bahel, H. Millet, M. Hickey, H. Pham, G. P. Herroon, and G. L. Greschl, "Heating and cooling system with variable capacity compressor," ed: Google Patents, 1997.

[8] K. Suefuji and S. Nakayama, "Practical method for analysis and estimation of reciprocating hermetic compressor performance," 1980.

[9] X. Zhao, J. Sun, and Z. Zhang, "Prediction and measurement of axial flow fan aerodynamic and aeroacoustic performance in a split-type air-conditioner outdoor unit," *International Journal of Refrigeration,* vol. 36, pp. 1098-1108, 5// 2013.

[10] R. Saidur, H. H. Masjuki, M. Y. Jamaluddin, and S. Ahmed, "Energy and associated greenhouse gas emissions from household appliances in Malaysia," *Energy Policy,* vol. 35, pp. 1648-1657, 3// 2007.

[11] S. Sadrzadehrafiei, K. S. S. Mat, and C. Lim, "Energy consumption and energy saving in Malaysian office buildings."

[12] R. Saidur, "Energy consumption, energy savings, and emission analysis in Malaysian office buildings," *Energy Policy,* vol. 37, pp. 4104-4113, 10// 2009.

[13] T. Kubota, S. Jeong, and D. H. C. Toe, "Energy Consumption and Air-Conditioning Usage in Residential Buildings of Malaysia (Renewable Energy)," *国際協力研究誌,* vol. 17, pp. 61-69, 2011.

[14] T. T. Chow, Z. Lin, and X. Y. Yang, "Placement of condensing units of split-type air-conditioners at low-rise residences," *Applied Thermal Engineering,* vol. 22, pp. 1431-1444, 9// 2002.

[15] T. T. Chow, Z. Lin, and J. P. Liu, "Effect of condensing unit layout at building re-entrant on split-type air-conditioner performance," *Energy and Buildings,* vol. 34, pp. 237-244, 3// 2002.

[16] K. Ryu, K.-S. Lee, and B.-S. Kim, "Optimum placement of top discharge outdoor unit installed near a wall," *Energy and Buildings,* vol. 59, pp. 228-235, 4// 2013.