Journal of Engineering Science and Technology
8th EURECA 2017 Special Issue August (2018) 79 - 91

AIRFLOW STUDY OF A SPILT-TYPE OUTDOOR UNIT SUBJECTED TO NEAR WALL EFFECT USING COMPUTATIONAL FLUID DYNAMIC (CFD) SIMULATION

SYED AHMAD FAHMI BIN SYED IBRAHIM[1], DOUGLAS TONG KUM TIEN[1,*], MOHD ANUAR BIN ABD AZIZ[2], CHIN WAI MENG[2]

[1]School of Engineering, Taylor's University, Taylor's Lakeside Campus, No. 1 Jalan Taylor's, 47500, Subang Jaya, Selangor DE, Malaysia
[2]Daikin Research & Development Malaysia Sdn Bhd,, Persiaran Bukit Rahman Putra 3, 47000 Sg Buloh, Selangor DE, Malaysia
*Corresponding Author: douglaskumtien.tong@taylors.edu.my

Abstract

The efficiency of the split-type air conditioner, which consists of indoor and outdoor units, is determined by the refrigeration cycle efficiency of its condensation, compression and evaporation processes. The key factors that affect the performance of the outdoor unit are the air flow profile and the refrigerant pressure at the compressor discharge. If the air flow rate decreases, the discharge pressure of the compressor increases. This will cause the power consumption to increase and overall efficiency to drop. One of the factors affecting the air flow profile is the installation distance between the outdoor unit and the wall. The distance of this wall gap is crucial because it affects the static pressure in the gap. This static pressure interferes with the air flow towards the outdoor unit. Therefore, the smaller the gap, the higher the static pressure and hence less air will flow through the outdoor unit. The purpose of this research is to study the air flow of a split-type outdoor air conditioner unit when it is placed near the wall and investigate its optimum placement from the wall. For the first stage of this investigation, the 3D CAD modelling of the DAIKIN outdoor unit of 5SL15F SERIES was completed using ANSYS SPACECLAIM software. The meshing was done using SnappyHexMesh code. Next, the simulation was conducted using OPENFOAM software to determine the air flow profile. The air flow profile was then inputted into the CoilDesigner software. Then, the CoilDesigner software was used to create the heat exchanger condition for the condenser and evaporator prior to transferring this condition into the VapCyc software. The VapCyc software is a refrigerant cycle programme and it was used to predict the discharge pressure at the compressor based on the obtained air flow profile. The input for the VapCyc software comprises of the specifications of the four air conditioner components, namely the condenser, evaporator, expansion device and compressor. The results show that optimum distance is at 200 mm for both *x*-axis and *y*-axis directions with 1000 mm for the *z*-axis. It has the highest airflow rate and lowest discharge pressure at the compressor.

Keywords: CFD simulation, Split-type air-conditioner, Near wall installation, OpenFOAM.

1. Introduction

Usage of a split-type air conditioner units is common in Malaysia as the country is located along the Equatorial line which experiences tropical climate for the entire year with high temperature and humidity [1]. This makes the air conditioner important as it helps maintain comfortable indoor temperature and humidity levels. Most of the residential area uses split-type air conditioner units because they are smaller than other types of air conditioner units [2].

A split-type air conditioner unit consists of two different parts which are the indoor and outdoor units. The indoor unit contains an evaporator while the outdoor unit contains a compressor and a condenser. The indoor unit absorbs heat inside the room while the outdoor unit releases the heat into the air through a heat exchange process inside the unit. This research will focus mainly on the outdoor unit as tasked by the industrial partner. In the process of designing the outdoor unit, there are certain conditions that must be applied, one of which is that the heat exchanger inside the outdoor unit must not be obstructed by other parts so that the air may flow through it. The flow of the air is created when the surrounding air is drawn in by a fan through the condenser and then discharged into the surroundings. In order to quantify the air passing through the heat exchanger, air volumetric flow rate is used.

There are many factors affecting the performance of the outdoor unit and one of them is the installation distance between the wall and the outdoor unit [3]. The wall gap distance is closely related to the air flow and it is necessary for the heat exchange process. Hence, the wall gap affecting the performance of the outdoor unit and the performance of the air conditioner system can be evaluated using the Coefficient of Performance (COP) formula [2, 4].

Essentially, the air temperature and flow rate affect the performance. Avara et al. stated that for every 1°C increment in temperature, there will be a 3% decrease of the COP value [5]. Furthermore, when the temperature reaches beyond 45°C, the entire air conditioning system will fail due to extreme conditions at the condenser and the evaporator [6]. This may be caused by the wall gap distance of the outdoor unit. An inappropriate wall gap distance could contribute to increasing the temperature and decreasing the air flow rate of the heat exchanger.

Chow et al. [7] studied the effect of the of the placement and re-entry of the outdoor unit. The author stated that the placement of the outdoor unit for all kinds of re-entrant played an important role in the performance. Also, that in increasing the number of the outdoor units placed together at tall buildings, the heat rejected will accumulate and cause a mass of heat around the system and hence affect the performance of the outdoor unit.

Avara et al. conducted a simulation using the k-epsilon model to search on the optimum placement of the outdoor unit. The relationship between the temperature and the placement are shown in Figs. 1 and 2. The author stated that at a certain distance, there is no change in flow rate. However, there will be hot air recirculation occurring that will result in temperature increase. When the distance of the installation is too near to the wall, the air flow will be low. Accordingly, the hot air recirculation affected the air flow rate due to the static pressure.

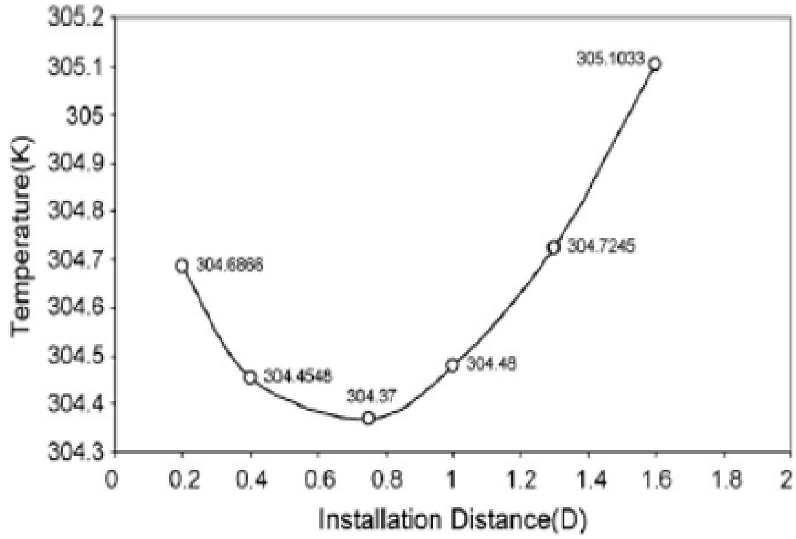


Fig. 1. Graph of temperature against the installation distance [5].

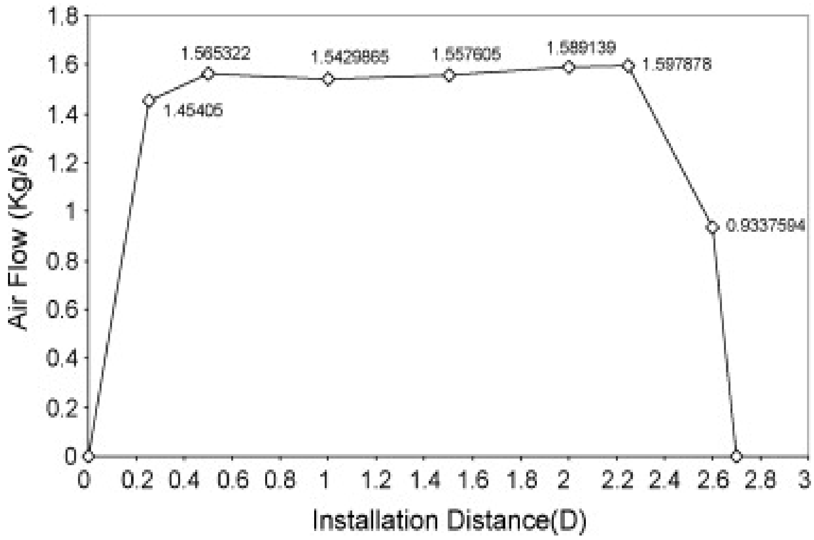


Fig. 2. Graph of air flow against the installation distance [5].

To analyse the flow and velocity of a fluid, Computational Fluid Dynamic (CFD) simulation is one of the best ways to identify the air flow around the outdoor unit. Avara and Daneshgar [5] conducted research using CFD simulation with the k-epsilon (k- ε) model being chosen to identify the turbulence flow created near the wall condition and high rpm speed of the fan blade. Furthermore, the k-ε model is able to give better quality and stable flow simulations. This was also supported by Chen [8] in the research about the comparison of the k-ε model for an indoor flow. However, radiative heat transfer was neglected due to the complex calculation in the simulation [3].

This research's objectives are to investigate the relationship between the air flow rate, measured in cubic feet per minute (CFM), with the installation distance of the outdoor unit from the wall and to predict the operating pressure of the compressor. CFD simulation is carried out to investigate the air flow rate when the outdoor unit is place at a certain wall gap distance. The distance should be measured along the *x*-, *y*-, and *z*-axes directions. These parameters were defined by referencing the research by Avara and Daneshgar [5]. However, research [5] did not consider changes along the *z*-axis. Therefore, the effect of the *z*-axis distance is examined in this paper. Based on the air flow profile produced in the simulation, VapCyc software will be used to predict the temperature and discharge pressure at the compressor. Lastly, this paper aims to determine the optimum placement of the outdoor unit when it is subjected to a near wall condition.

2. Research Methodology

The methodology employed for this study is discussed below. Each stage is explained with the help of relevant figures. The work starts with the creation of the model in ANSYS version 15.0, followed by CFD simulation with OPENFOAM version 3.4, and then analysed by using CoilDesigner and VapCyc software. There will be further information about CoilDesigner and VapCyc software in subsequent sections of this paper.

2.1. Stage 1

The model creation and the identification of study cases, which represents the first stage of this study, are elaborated in the subsections to follow.

2.1.1. CAD drawing of the outdoor unit

The original drawings of the outdoor unit 5SLY15F SERIES provided by DAIKIN are detailed with some complicated geometry. Hence, a simplified model is done using ANSYS SPACECLAIM where the important components are separated, leaving the rest being removed (de-featured). This process will create a simpler mesh and save a lot of time for simulation. The important parts left are the fan blade, the casing of the outdoor unit, the grill, the bell mouth and the heat exchanger (condenser). The drawings for each part is shown in Fig. 3.

Based on Fig. 3, the drawings are a simplified version where all the complex geometry has been removed. These simplified drawings produce a lesser number of mesh elements and eliminate error during meshing. An assembly drawing is shown in Fig. 4, which is the representation model used in the simulation.

Subsequently, the simulation has to correspond with the real operation where the fan blade rotates at a certain speed (rpm). To ensure that the simulation model is correct, a Multiple Reference Frame (MRF) zone has been created. During the simulation, the OPENFOAM software was used to create a boundary around the fan blade which allowed it to rotate inside the MRF zone itself. Since the MRF zone was assigned separately, it can perform simulations under different speeds (rpm). The MRF zone created is shown in Fig. 5.

The purpose of adding the MRF zone into the fan blade was to ensure that the software will recognize which parts were moving (rotating) and stationary.

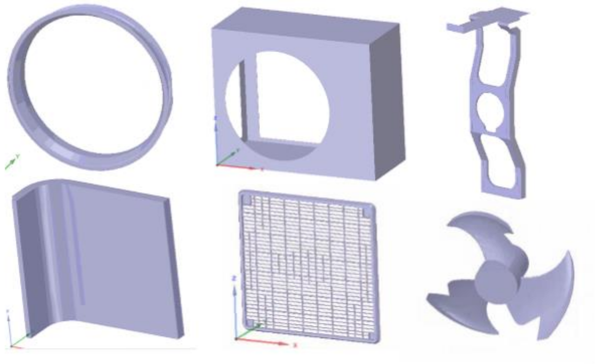


Fig. 3. CAD drawing of the parts for the outdoor unit.

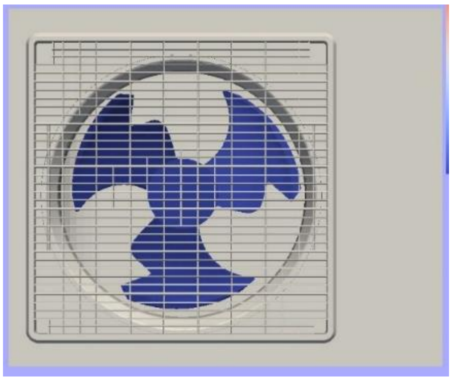


Fig. 4. Assembly drawing of the outdoor unit.

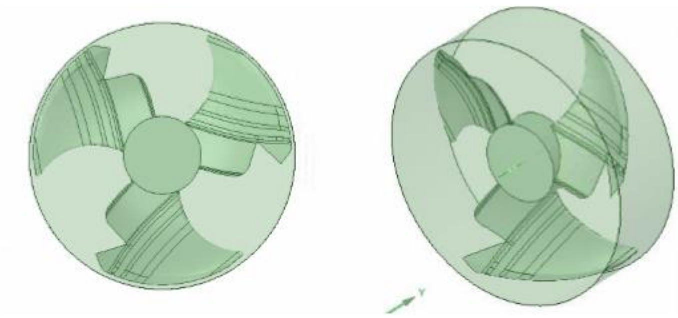


Fig. 5. Multiple reference frame zone.

2.1.2. Case studies

Before proceeding with the simulation, several parameters must be set to allow comparison and analysis of the results for the optimum placement. The fan speed for every case will be constant while the variables are the distance of the outdoor unit from the wall along the *x*, *y*, and *z*-axis, as shown in Fig. 6.

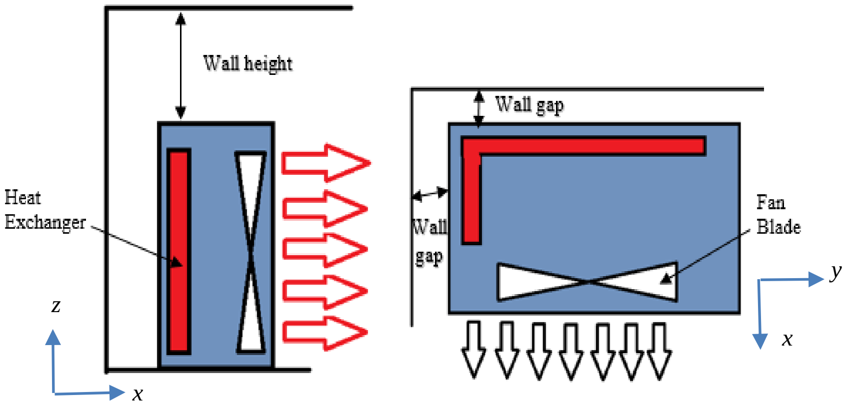


Fig. 6. Manipulated variables.

Table 1 represents the cases for the simulation where three different wall distances have been manipulated. There was a total of 6 simulations conducted throughout the research where the *x* and *y* distances were varied between 60 mm and 200 mm. For any distance lower than 60 mm, the simulation produces an error due to the small gap. According to Avara and Daneshgar [5] for a distance more than 200 mm, the air temperature will remain nearly unchanged. Therefore, the range chosen was between 60 mm to 200 mm. As for the z distance, this was set at 50 and 1000 mm.

Table 1. Parameters of case studies.

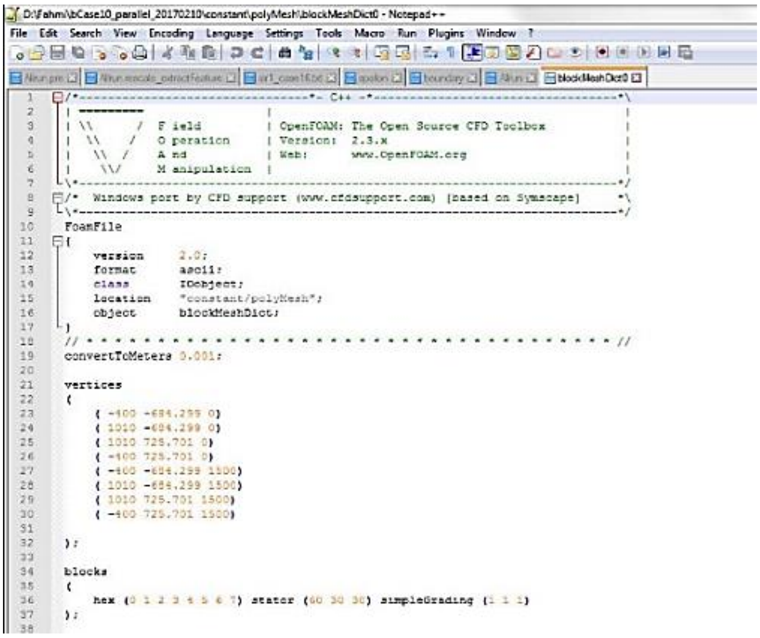
Case Studies	Wall Distance		
	x-axis(mm)	*y*-axis(mm)	*z*-axis(mm)
Case 1	60	60	500
Case 2	60	60	1000
Case 3	150	150	500
Case 4	150	150	1000
Case 5	200	200	500
Case 6	200	200	1000

2.2. Stage 2

CFD simulation with OPENFOAM version 3.4, which represents the second stage of the study is elaborated in the subsections to follow.

2.2.1. Meshing of the model

After the first part was completed, the model was then meshed inside the OPENFOAM software using the SnappyHexMesh method, which created a fine mesh. Since the software uses the C++ language, all the information and editing were done in coding format as shown in Fig. 7. Figure 8 shows the sample of a refined mesh created on the fan blade using the SnappyHexMesh.



D:\Fahmi\bCase10_parallel_20170210\constant\polyMesh\blockMeshDict0 - Notepad++

File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  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]     *\
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       IOobject;
    location    "constant/polyMesh";
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
convertToMeters 0.001;

vertices
(
    ( -400 -684.299 0)
    ( 1010 -684.299 0)
    ( 1010 725.701 0)
    ( -400 725.701 0)
    ( -400 -684.299 1500)
    ( 1010 -684.299 1500)
    ( 1010 725.701 1500)
    ( -400 725.701 1500)

);

blocks
(
    hex (0 1 2 3 4 5 6 7) stator (60 30 30) simpleGrading (1 1 1)
);
```

Fig. 7. OPENFOAM software coding.

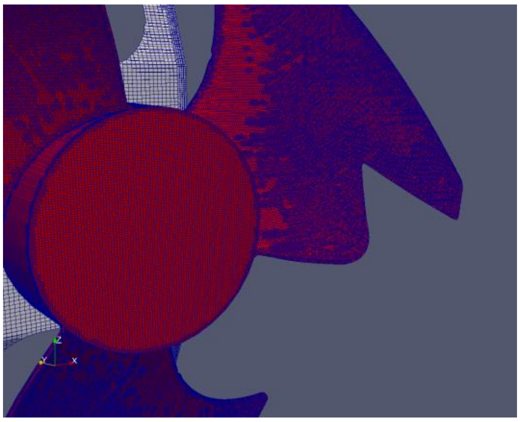


Fig. 8. Refined mesh at the fan blade.

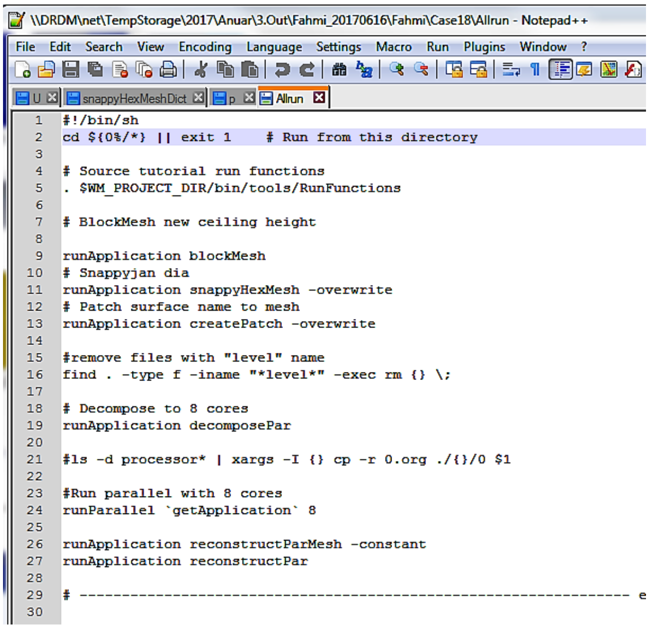
2.2.2. CFD Simulation

CFD simulation was then completed by using the OPENFOAM software which consists of multiple solvers. This simulation assumes incompressible flow which narrows down the list of solvers. “simpleFoam” was chosen as the solver due to its steady-state characteristic suitable for the turbulent flow region around the fan blade. However, OPENFOAM is an open source software which does not come with a user guide. A list of the few commands used in performing the simulation is shown in Table 2.

All the executed commands were recorded and saved in a log file which can be viewed during the process. Figure 9 shows a sample of the command and Fig. 10 shows the data recorded in the log file.

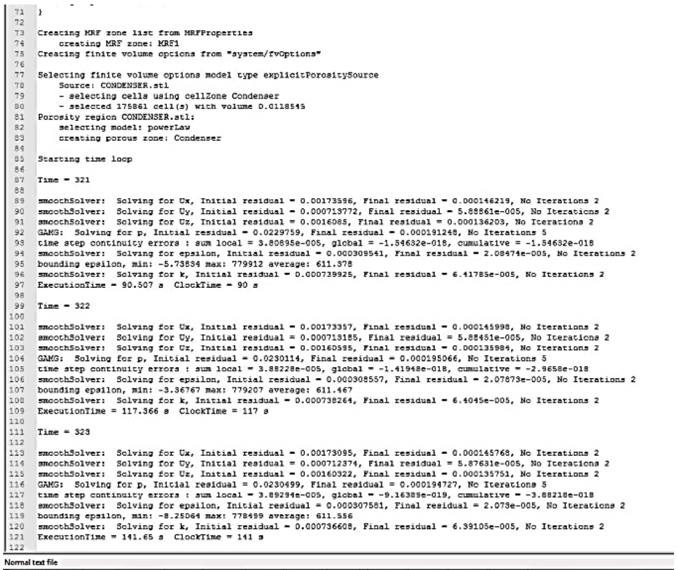
Table 2. Commands used in OPENFOAM.

Command	Execute
blockMesh	Applying the basic mesh into the 3D-model.
snappyHexMesh	Refining the mesh.
simpleFoam	Run the simulation.
Allrun	Run all the commands according to the sequent that is set.




```
#!/bin/sh
cd ${0%/*} || exit 1    # Run from this directory

# Source tutorial run functions
. $WM_PROJECT_DIR/bin/tools/RunFunctions

# BlockMesh new ceiling height

runApplication blockMesh
# Snappyjan dia
runApplication snappyHexMesh -overwrite
# Patch surface name to mesh
runApplication createPatch -overwrite

#remove files with "level" name
find . -type f -iname "*level*" -exec rm {} \;

# Decompose to 8 cores
runApplication decomposePar

#ls -d processor* | xargs -I {} cp -r 0.org ./{}/0 $1

#Run parallel with 8 cores
runParallel `getApplication` 8

runApplication reconstructParMesh -constant
runApplication reconstructPar

# ---------------------------------------------------------------- e
```

Fig. 9. Allrun command script file.



```
}

Creating MRF zone list from MRFProperties
    creating MRF zone: MRF1
Creating finite volume options from "system/fvOptions"

Selecting finite volume options model type explicitPorositySource
    Source: CONDENSER.stl
    - selecting cells using cellZone Condenser
    - selected 175861 cell(s) with volume 0.0118545
Porosity region CONDENSER.stl:
    selecting model: powerLaw
    creating porous zone: Condenser

Starting time loop

Time = 321

smoothSolver:  Solving for Ux, Initial residual = 0.00173596, Final residual = 0.000146219, No Iterations 2
smoothSolver:  Solving for Uy, Initial residual = 0.000713772, Final residual = 5.88861e-005, No Iterations 2
smoothSolver:  Solving for Uz, Initial residual = 0.0016085, Final residual = 0.000136203, No Iterations 2
GAMG:  Solving for p, Initial residual = 0.0229759, Final residual = 0.000191248, No Iterations 5
time step continuity errors : sum local = 3.80895e-005, global = -1.54632e-018, cumulative = -1.54632e-018
smoothSolver:  Solving for epsilon, Initial residual = 0.000309541, Final residual = 2.08474e-005, No Iterations 2
bounding epsilon, min: -5.73834 max: 779912 average: 611.378
smoothSolver:  Solving for k, Initial residual = 0.000739925, Final residual = 6.41785e-005, No Iterations 2
ExecutionTime = 90.507 s  ClockTime = 90 s

Time = 322

smoothSolver:  Solving for Ux, Initial residual = 0.00173357, Final residual = 0.000145998, No Iterations 2
smoothSolver:  Solving for Uy, Initial residual = 0.000713185, Final residual = 5.88451e-005, No Iterations 2
smoothSolver:  Solving for Uz, Initial residual = 0.00160595, Final residual = 0.000135984, No Iterations 2
GAMG:  Solving for p, Initial residual = 0.0230114, Final residual = 0.000195066, No Iterations 5
time step continuity errors : sum local = 3.88228e-005, global = -1.41948e-018, cumulative = -2.9658e-018
smoothSolver:  Solving for epsilon, Initial residual = 0.000308557, Final residual = 2.07873e-005, No Iterations 2
bounding epsilon, min: -3.36767 max: 779207 average: 611.467
smoothSolver:  Solving for k, Initial residual = 0.000738264, Final residual = 6.4045e-005, No Iterations 2
ExecutionTime = 117.366 s  ClockTime = 117 s

Time = 323

smoothSolver:  Solving for Ux, Initial residual = 0.00173095, Final residual = 0.000145768, No Iterations 2
smoothSolver:  Solving for Uy, Initial residual = 0.000712374, Final residual = 5.87631e-005, No Iterations 2
smoothSolver:  Solving for Uz, Initial residual = 0.00160322, Final residual = 0.000135751, No Iterations 2
GAMG:  Solving for p, Initial residual = 0.0230499, Final residual = 0.000194727, No Iterations 5
time step continuity errors : sum local = 3.89294e-005, global = -9.16389e-019, cumulative = -3.88218e-018
smoothSolver:  Solving for epsilon, Initial residual = 0.000307581, Final residual = 2.073e-005, No Iterations 2
bounding epsilon, min: -8.25064 max: 778499 average: 611.556
smoothSolver:  Solving for k, Initial residual = 0.000736608, Final residual = 6.39105e-005, No Iterations 2
ExecutionTime = 141.65 s  ClockTime = 141 s
```


Fig. 10. Sample of a running simulation in a log file.

The purpose of running the simulation is to evaluate the air volume flow rate through the heat exchanger. The air volume flow rate and air velocity profile through the heat exchanger coil were extracted from the simulation and the results analysed.

2.3. Stage 3

Analysis using CoilDesigner and VapCyc software, which represents the third stage of the study, are elaborated in the subsections to follow.

2.3.1. CoilDesigner software

Another software called CoilDesigner was used to analyze the heat exchanger performance. The actual heat exchanger, or also known coil, was drawn in this particular software. Since an air conditioner system consists of two coils, which are the indoor coil (evaporator) and the outdoor coil (condenser), both of these have different design specifications. Figure 11 shows the model of the condenser coil which has been discretized into smaller cells.

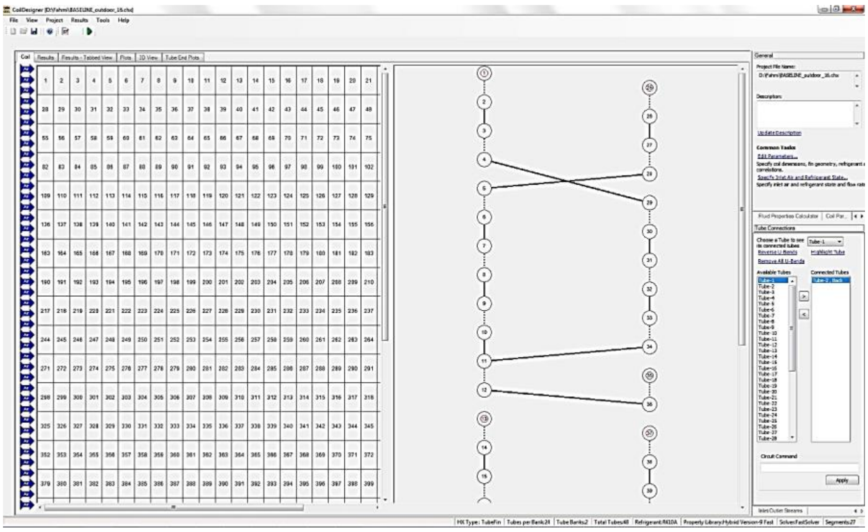


Fig. 11. Model of the condenser coil using CoilDesigner.

The air flow velocity profile obtained from the CFD simulation was then input into the corresponding discrete cell elements in CoilDesigner. For a known entering air conditions, the outcome of this software is to obtain the air leaving temperature of the condenser and evaporator units which became the inputs for the VapCyc software.

2.3.2. VapCyc software

Using the information obtained from the CoilDesigner software, the heat exchanger performance was then input into the VapCyc software which simulate the refrigeration cycle of the air-conditioner system. Data from the condenser, evaporator, and compressor were applied accordingly as shown in Fig. 12.

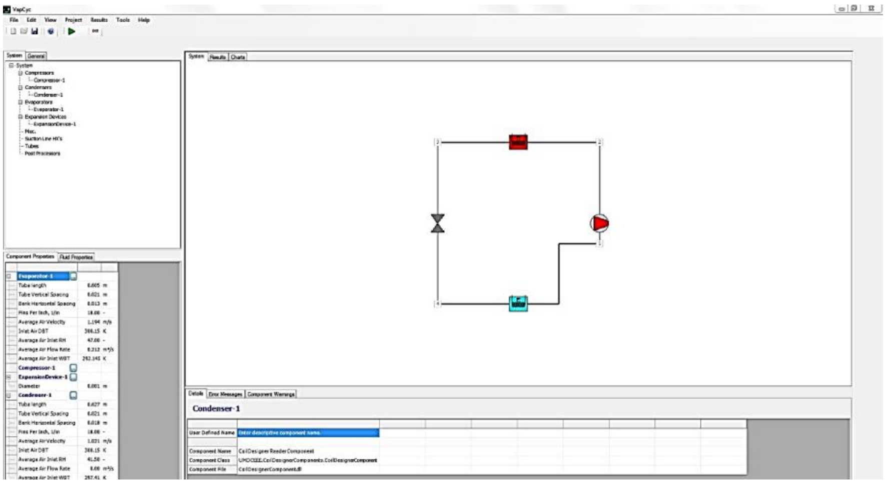


Fig. 12. Refrigeration Cycle in VapCyc software.

The purpose of using VapCyc software was to calculate the predicted value of the compressor discharge pressure according to the air flow rate when subjected to near wall conditions. The results were recorded and analyzed to determine the optimum placement of the outdoor unit.

3. Results and Discussion

The results of the airflow profile, air flowrate and discharge pressure are discussed in the following subsections.

3.1. Airflow profile

The air flow profile is shown in Fig. 13. It is revealed that a few important factors influence the air flow rate. Firstly, changes of static pressure at some locations affect the flow velocity profile. For example, the air velocity decreased as a result of the static pressure build-up that occurred due to blockages of the confined space.

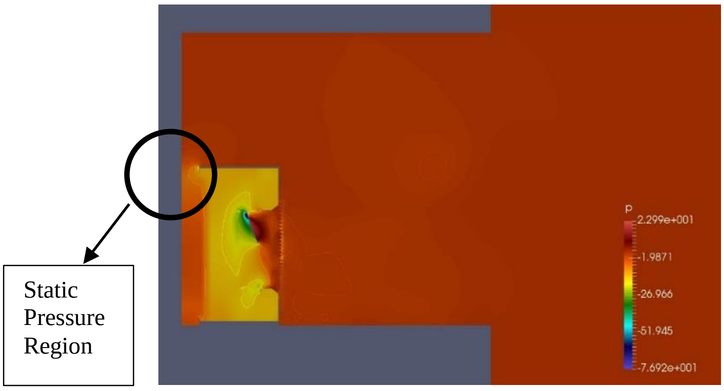


Fig. 13. Static pressure behind the outdoor unit.

The Bernoulli's Principle states that high static pressure will result in a low fluid velocity. Therefore, when there is a smaller gap between the outdoor unit and the wall, the static pressure will increase which then causes the flowrate, in cubic feet per minute (CFM), to decrease at the same time. Secondly, the air flow profile shows a backflow or a heat recirculation due to the suction from the rotating fan as shown in Fig. 14.

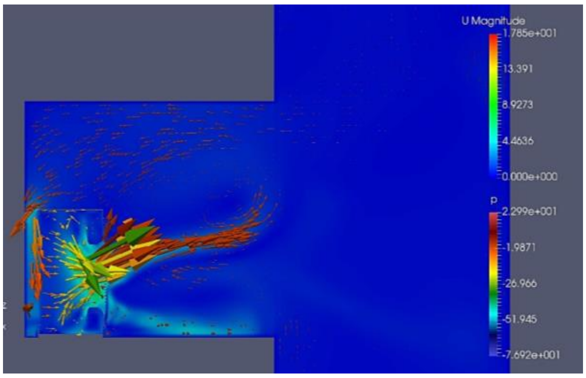


Fig. 14. Hot air recirculation flow profile.

The hot air recirculation is the backflow of the hot air that is blown out. This is due to the suction from the rotating fan causing less ambient air to be drawn from other directions. The hot air increases the entering air temperature of the condenser. Therefore, the discharge pressure of the compressor is affected as well.

3.2. Flowrate (CFM) and discharge pressure

Figure 15 shows the graph of CFM vs. discharge pressure (MPa) of the compressor for each of the cases studied. The graph shows that CFM is inversely proportional with the discharge pressure of the compressor. When the CFM value is low, the discharge pressure of the compressor becomes high.

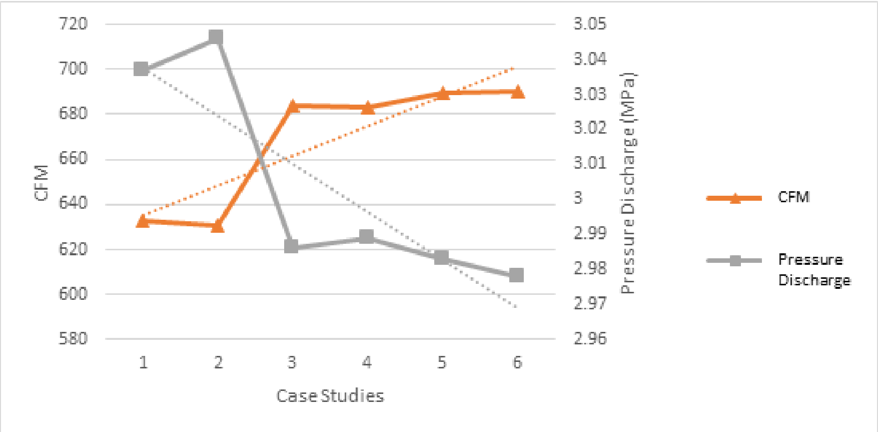


Fig. 15. CFM and pressure discharge (MPa) of the compressor.

This situation occurred due to the restricted air flow. Plus, the pressure is inter-related with the area. Since case 6 has the largest wall gap, the CFM value is high and there is a larger amount of heat transfer taking place. There are also some other factors that might affect the discharge pressure such as the degree of refrigerant subcool and superheat. However, for this study, these have been fixed as constants throughout the simulation.

Referring to the Table 3, it is clear that the minimum wall gap of 60 mm produces a very low value of CFM while on the other hand, a wall gap of 200 mm produces a high value of CFM. The CFM value remains nearly unchanged between 150 mm to 200 mm. However, the manipulated *z*-axis does not produce significant effect for each case.

Table 3. CFM results.

Case Studies	Power Consumption (W)	CFM
1	1078.91	630.31
2	1076.77	632.98
3	1063.92	683.40
4	1063.17	683.97
5	1062.42	689.48
6	1062.31	690.05

In summary, for every increment of the wall gap, there is a significant change in the CFM value and discharge pressure value. This shows that CFM is correlated to the discharge pressure of the compressor. When the discharge pressure is high, the power consumption of the compressor will increase too. Although the difference in power consumption is small, it will still affect the operating cost. In other words, the cost of the electricity will increase due to the extra workload of the compressor. In some cases, it will cause the entire system to fail due to the extreme condition. Therefore, it is important to place the outdoor unit at a proper distance due to the effect of the discharge pressure of the compressor.

4. Conclusion

The methodology of analysing the performance of compressor using CFD (ANSYS-SpaceClaim version 16, SnappyHexMesh, OpenFoam version 3.0) and system simulation (CoilDesigner, VapCycle) has been demonstrated. It is shown that smaller wall gap at constant fan speed produces high flow resistance due to higher static pressure. Hence, the air flow rate (CFM) becomes less. The optimum placement of the outdoor unit from this study is 200 mm from the wall along the *x*-axis and *y*-axis. For *z*-axis, which is the top wall height, minimum effect is observed up to a minimum distance of 500 mm. Based on the CFM value, the discharge pressure of the compressor has been predicted. It is demonstrated that at lower values of CFM produced, higher discharge pressure is observed, which in turn will further push the compressor to exceed its safe operating envelope.

Acknowledgement

This research was a collaborative project with DAIKIN Research & Development Malaysia Sdn Bhd. The authors would like to acknowledge their appreciation to DAIKIN Malaysia Sdn. Bhd.

References

1. Jamaludin, N.; Mohammed, N.I.; Khamidi, M.F.; and Abdul Wahab, S.N. (2015). Thermal comfort of residential building in Malaysia at different micro-climates. *Procedia - Social and Behavioral Sciences*, 170, 613-623.
2. Çengel, Y.A.; and Boles, M.A. (2011). *Thermodynamics : An engineering approach*. McGraw-Hill, New York, NY.
3. Ryu, K.; Lee, K.-S.; and Kim, B.-S. (2013). Optimum placement of top discharge outdoor unit installed near a wall. *Energy and Buildings,* 59, 228-235.
4. Jin, W.; Zheng, Y.; Zhang, Y.; and Jiang, Y. (2015). Experimental study of factors affecting the performance of a semi-enclosed outdoor air-conditioning unit. *Procedia Engineering*, 121, 1713-1720.
5. Avara, A.; and Daneshgar, E. (2008). Optimum placement of condensing units of split-type air-conditioners by numerical simulation. *Energy and Buildings,* 40(7), 268-1272.
6. Chow, T.T.; Lin, Z.; and Yang, X.Y. (2002). Placement of condensing units of split-type air-conditioners at low-rise residences. *Applied Thermal Engineering*, 22(13), 1431-1444.
7. Chow, T.T.; Lin, Z.; and Liu, J.P. (2002). Effect of condensing unit layout at building re-entrant on split-type air-conditioner performance. *Energy and Building*s, 34(3), 237-244.
8. Chen, Q. (1995). Comparison of different k -ε models for indoor air flow computations. *Numerical Heat Transfer, Part B: Fundamentals,* 28(3) 353-369.