Progress in Engineering Application and Technology Vol. 3 No. 1 (2022) 187–195
© Universiti Tun Hussein Onn Malaysia Publisher's Office

PEAT

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/peat
e-ISSN : 2773-5303

Simulation of Forced Ventilation in Closed Building as COVID-19 Precaution

Cheong Yung Khai[1], **Fatimah Mohamed Yusop**[1]*

[1]Department of Civil Engineering Technology, Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor

*Corresponding Author Designation

DOI: https://doi.org/10.30880/peat.2022.03.01.021
Received 16 January 2022; Accepted 11 April 2022; Available online 25 June 2022

Abstract: Fresh air is referred as the supply of air for human which containing oxygen particles in the air. Unfortunately, the current outbreak of COVID-19 has becoming fresh air supply is needed more in closed area which would enable to neutralize the virus of COVID-19. The supply of air in the closed area basically from the HVAC system such as air-conditioning. The air movement is circulated and reused in the closed area. Ventilation system as forced ventilation can be used for more supply of fresh air in a closed area. It is usually used to control the air quality by adulterate unnecessary particles in temperature, and airflow velocity. In closed area, COVID-19 infection is easier to transmit the virus while fresh air supply insufficient. Hence, a Computational Fluid Dynamics (CFD) simulation is used to simulate the airflow velocity and temperature in a closed area by using force ventilation to the building. Besides, a model of closed area that used for simulation was also designed by using the computer-aided design software, SolidWork. The model is simulated without simulation and simulation with forced ventilation. From the result, it can be said that the airflow between both of simulation has their own air distribution and velocity. In addition, the temperature distribution for without ventilation and forced ventilation is in the range of thermal comfort condition. This resulted the simulations are simulated efficiently. At the end of the study, it can be concluded that the simulation with forced ventilation has more airflow velocity generated compared to the simulation without ventilation.

Keywords: Fresh air, COVID-19, Forced Ventilation, Computational Fluid Dynamics (CFD)

1. Introduction

Coronavirus disease known as COVID-19 has affected our daily life. It has affected many developments such as construction, economy, and so on. Coronavirus disease (COVID-19) is a lung infectious disease that a second generation of severe acute respiratory syndrome (SARS). COVID-19 virus would make people mild to diminish respiratory illness and recuperate without treatment. Thus, some of the medical problems who fundamental with diabetes, cancer and so on would develop serious

*Corresponding author: rahmatm@uthm.edu.my

illness if infected COVID-19 [1]. The transmission of the COVID-19 has related to the airflow in a surrounding environment. In the movement of air, it has caused the existance of airflow. In a closed space, the air mostly is provided by HVAC system such as air-conditioner. The airflow of a fluid basically travels from a higher pressure to lower pressure in a place. This is because of atmospheric air pressure which acts from a few factors of temperature and composition. By increasing the ventilation of closed area is a key method of dispersing viral particles. Ventilation can reduce the risk of transmission when one of persons that infected the virus infect others. The flow of air would persuade through artificial mankind innovate product such as fans and air-conditioning in closed area.

In closed area, COVID-19 infection in single-occupancy rooms is easier to transmit the virus [2]. Virus transmission in this outbreak cannot be explained by particles transmission alone. Larger respiratory particles remain in the air for only a short time and travel only short distances. The particles of virus transmission were prompted by air-conditioned ventilation. The key factor for infection was the direction of the airflow [3].

Computational fluid dynamics (CFD) is a modelling technique that can be predict airflows patterns around a specific arears has chosen. CFD is a tool that creates a virtual airflow and building thermal model to evaluate and optimize of a design building. The calculations basically can be simulating the airflow and the interaction of fluids between liquid and gases by partitions conditions. CFD normally applied in a large area because the accuracy of readings in meshing structure would determine while the simulation analyzed. Therefore, this research study will used CFD simulation is to simulate the air flow in a closed area by forced ventilation to the building.

2. Methodology

The methodology used for research is related to the simulation of computational fluid dynamics software which propose a strategy of force ventilation as a method to prevent COVID-19. The aim of this research is to simulate the airflows and ventilation inside of laboratory rooms. With the data simulated, a suggested strategy of force ventilation as a method to prevent COVID-19. The process of the research would show the importance of methodology for researchers so that they would construct the research in an efficient approach without any mistake taken during the investigation. The research is determining a strategy of force ventilation as a method to prevent COVID-19 virus during the pandemic period. The methodology contains a flowchart which acts as a guideline of the research project shown as Figure 1. The research of methodology is an approach that must carried out to achieve the goal of research. During achieving the goal of research, literature review must be done before the chapter of methodology due to context conceptual of research. In this research, the method of research is significant that carried out with few of method to achieve the aim and the objective.

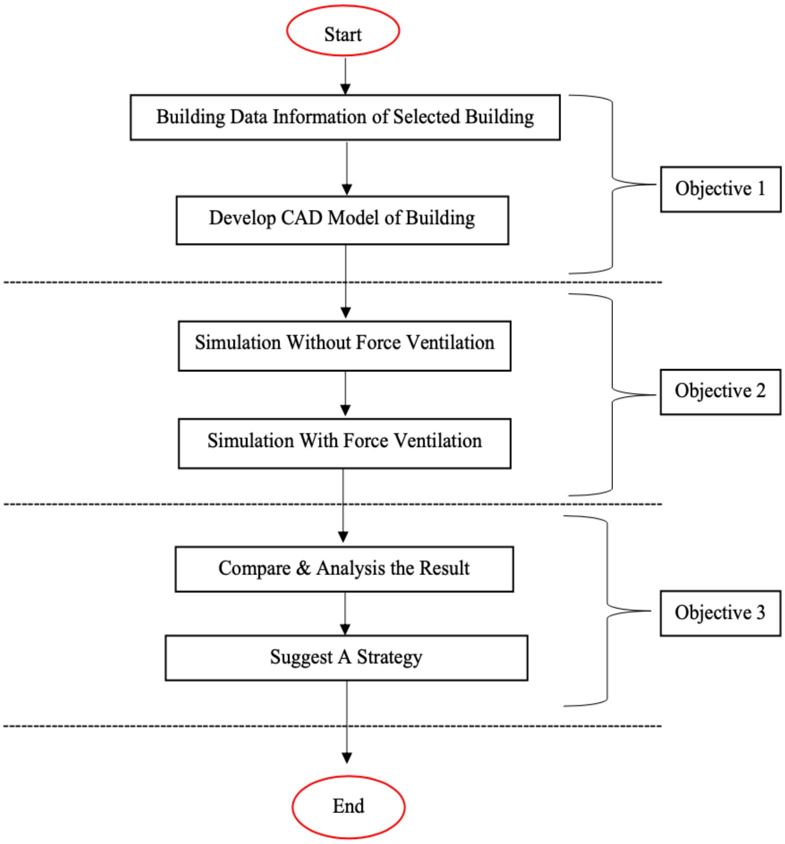

Figure 1: Flow Chart of Methodology

In this research, there are few methods of research which are CAD model, simulation, and comparison on result. CAD software is mainly used for mechanical design which would include the steps of creating a technical drawing with the usage of computer [4].

In the simulation, it consists of two method which are simulate without force ventilation and simulate with force ventilation. Stretch can identify the object or model are correctly in shape while calculation can analyze the airflow in the object or model. This strategy can be used because the method are limitations. Researcher would think that the approach would balance the deficiency in a process of work. Next, assume that, there are some of the resource and data would benefit in explanation of analyze data. The data simulated would be in two forms which are stretched graph and 3D model graphic. The data is based on the airflow in the laboratory room at building of Blok G of Univesiti Tun Hussein Onn Malaysia, Pagoh. The explanation of the data would represent an airflow in a laboratory room and suggest a strategy which would help to prevent COVID-19 virus during the pandemic. Hence, the simulation results must be done to achieve the goal of this research.

2.1 Develop CAD model

Computer-aided design is a software which helps to create and modify a design from a building. CAD software is mainly used for mechanical design which would include the steps of creating a technical drawing with the usage of computer. In the research project, CAD software is used to create a model which figures in two dimensional (2D) or solid work in three-dimensional (3D). The design of the model is to intend the building components and measurements to constitute what facilities has conducted in a closed area as laboratory rooms. After creating the model, the building structure of laboratory rooms is completed. The objective of develop the physical building structure of laboratory room is achieved while applied the model.

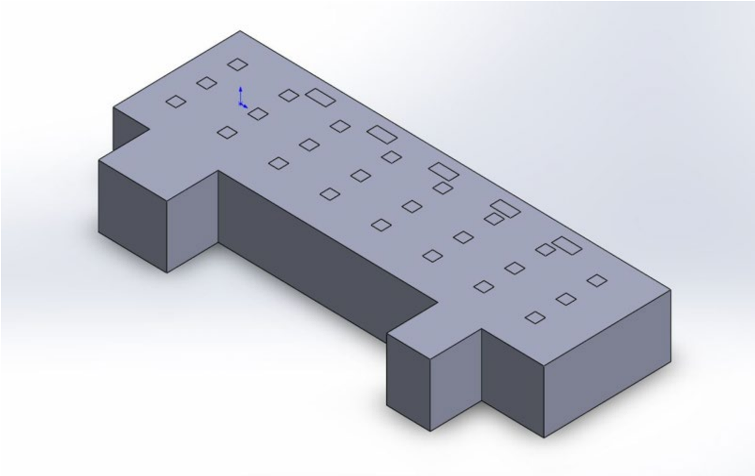

Figure 2: CAD model by using Solidwork

2.2 Simulation Process

The next stages are to manipulate the simulation by using ANSYS-FLUENT. The process of the simulation is geometry, grid generation, solver and data visualization.

2.2.1 Grid generation

Grid generation is a procedure that would gone through on meshing the CAD model into the element. This research project was used geometry to display the meshing on elements. The minimum sizes of element of the model are 2.0 mm as shown in Figure 3 and Table 1.

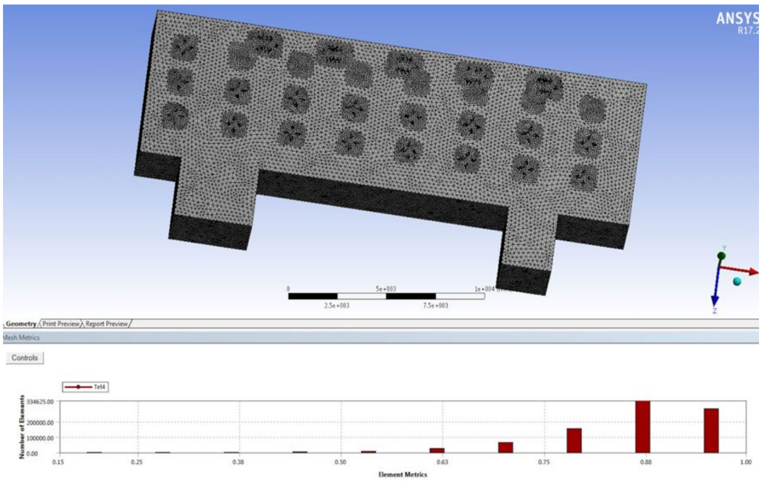

Figure 3: Mesh of CAD model

Table 1: Properties of mesh

Elements Node	157204
Smoothing	High
Minimum Size	2.0 mm
Maximum Face Size	200.0 mm

2.3 Model Simulation

The model is obtained by referring the simulation of experimental data by [5]. They had recommended the setup model for simulation of airflow in closed area. For example, the parameter of setup to ensure the results are valid to be compared by using different studies.

Table 2: Model Selection Method

Items	
Domain Viscous model	k-epsilon (2 equation)
k-∈ Model	RNG
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind

3. Results and Discussion

It will discuss on the findings achieved from the simulation in avoid transmission of COVID-19 in laboratory rooms by using computational fluid dynamics (CFD). The results simulated would be discuss more about the condition of simulation which is corresponded to the airflow content in the laboratory rooms. The airflow content is presented by two ventilation which are natural ventilation and force ventilation in the laboratory room.

3.1 Simulation in Closed Area

An experimental result has simulated which is without ventilation. The average of air velocity is calculated by cutting the plan of the model which would present the accurate reading for the simulation. The temperature of the model is 296 K. The average air velocity for model without ventilation is 0.8 m/s as shown in Figure 4. Referring to the data, the results of model does not meet the standard air flows which further consideration are needed to improve the air velocity to prevent transmission of COVID-19. The results shown that model without ventilation is reasonably with better airflow velocity in laboratory room.

Figure 5 shows the CFD stream plot of the air velocity for laboratory room. By referring on the figures, it can show that the air distribution pattern of model for centralized the air velocity.

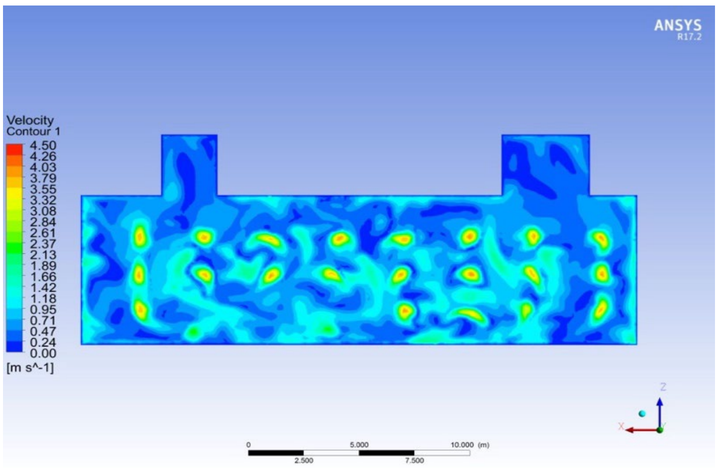

Figure 4: Air velocity distribution in closed area

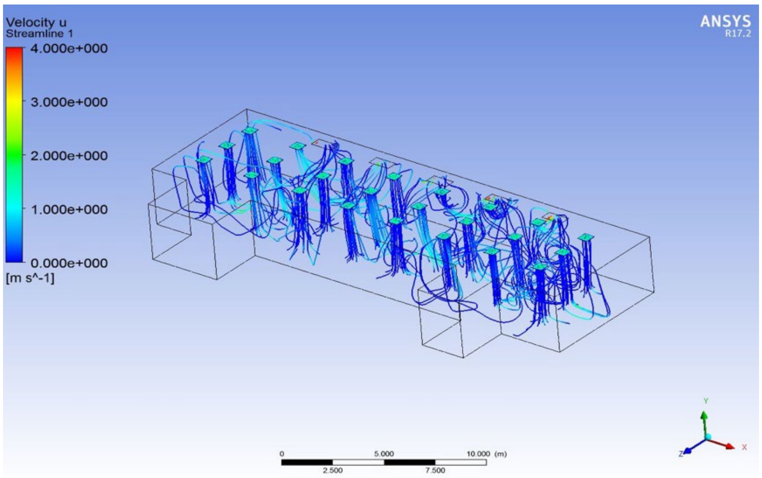

Figure 5: Air velocity streamline in closed area

3.2 Simulation with Forced Ventilation

In the procedures of simulate forced ventilation, the factors of doors and windows has considered which would provide more fresh air into the laboratory room when the forces of air pressure of air-conditioning are not efficient to circulate air through laboratory room. Forced ventilation is needed to improve the airflows of laboratory rooms. The average air velocity for model with forced ventilation is 1.0 m/s as shown in Figure 6. The temperature of model is 292 K because of external airflow from out which transmit into the model as shown in Figure 7. This resulted that the air velocity is flow in a good condition which could generate more fresh air in the laboratory room. Hence, the transmission of COVID-19 can be prevented by applying forced ventilation.

Figure 8 shows the CFD stream plot of the inlet air velocity for laboratory room. By referring on the figures, it can show that the air distribution pattern of model which allows to determine the efficient ways on prevention of transmission COVID-19 virus by the airflow.

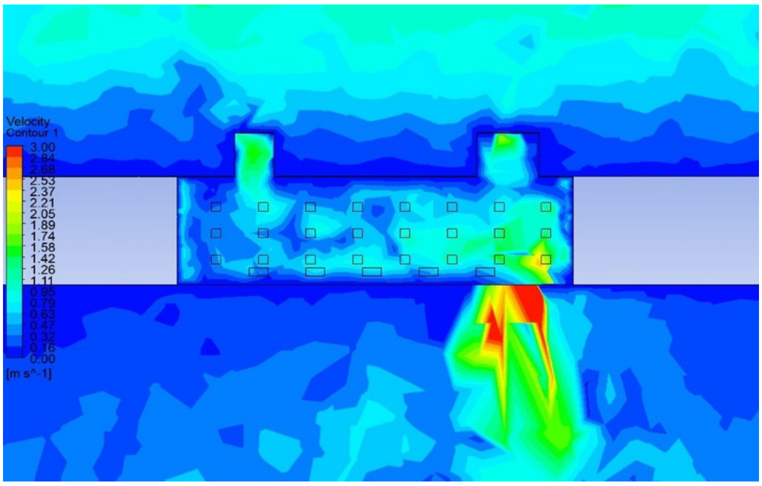

Figure 6: Air velocity distribution with forced ventilation

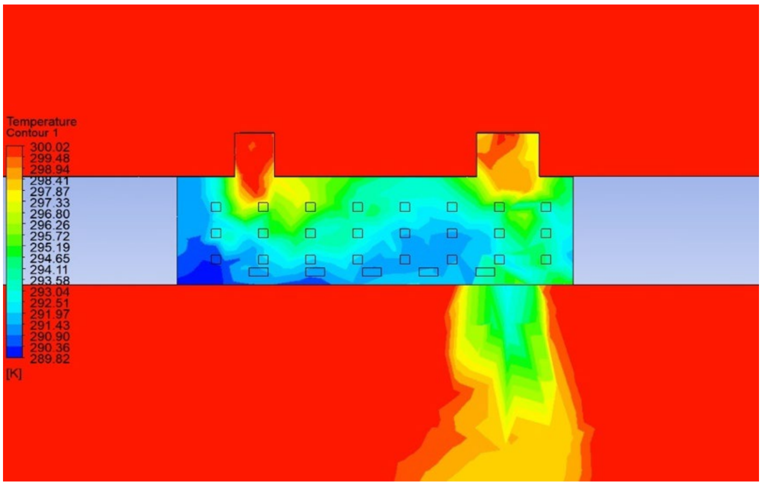

Figure 7: Temperature distribution with forced ventilation

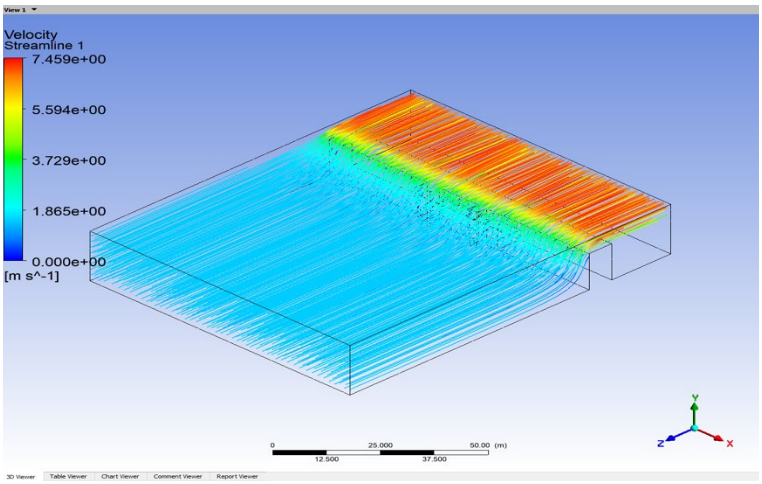

Figure 8: Air velocity streamline with forced ventilation

3.3 Air Changes Per Hours

The air changes per hour have calculated by using formula of air changes per hour which referred [6]. An air changes per hour means the air enters and exits from laboratory room from the HVAC system in an hour. The number of air changes has represented to balance the airflow which brings advantages to prevent the spread of COVID-19 virus. The data of the results shown in Table 2.

Table 2: The data of air flow rates for forced ventilation of laboratory room

Components	Flow rates
Inlets	112.0568 m^3/s
Outlets	186.7614 m^3/s

The of air changes per hour for forced ventilation of laboratory room is 6.85 ACH. Air changes per hour is an important measurement for estimates the efficiency of air is exchanged in a closed area.

In the research project, there are two doors in the laboratory rooms which represented as door 1 and door 2. The doors have presented as inlets which is the airflows enter into the model. The average of air velocity is calculated by cutting the plan of the model which would present the accurate reading for the simulation. By opening the door 1 and door 2, the average air velocity is 0.3 m/s as shown in Figure 8.

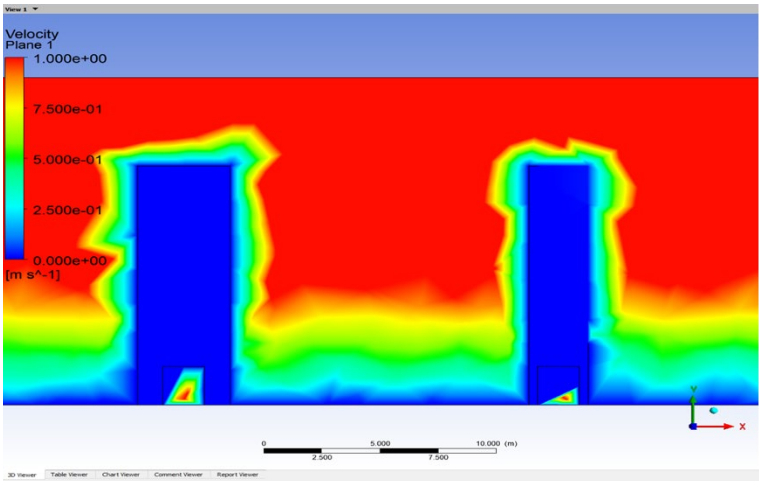

Figure 9: Inlets air velocity distribution with forced ventilation

Moreover, laboratory room consists of a bar of windows which the windows are connected to one another. The windows have assumed as one window with the overall length and height of the bar of windows. The windows presented as outlets which is the exits airflows out the model. The average air velocity for windows is 1.0 m/s as shown in Figure 9. Referring to the data, the results of windows has extracted more airflows compared to doors. Referring to the data, the results of windows has extracted more airflows compared to doors.

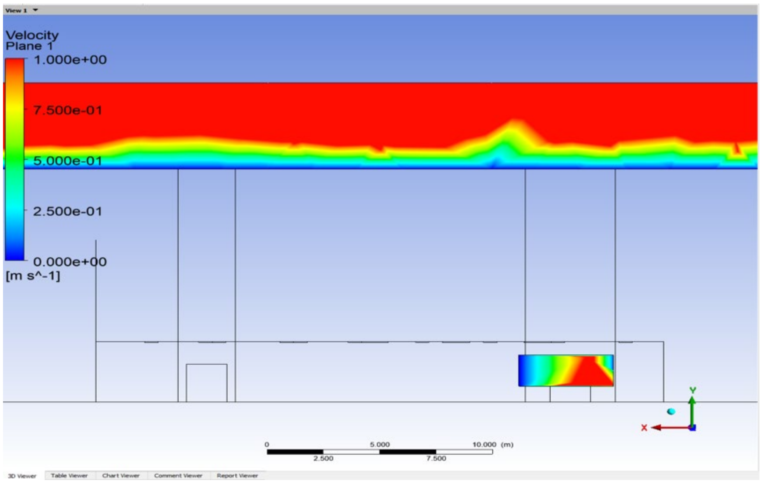

Figure 10: Outlets air velocity distribution with forced ventilation

3.4 Summary and Suggestion

The summary of temperature and velocity of the simulation data is analyzed. The temperature in closed area is in the range of 296 K to 299 K which is achieved the standard physical parameters of a laboratory rooms. But forced ventilation is considered to increase the efficiently to generate more fresh air. The temperature is 292 K which is lower temperature than standard physical parameter. Based on the simulation in closed area, it has shown that the air velocity in the model is slower than simulation with forced ventilation results. By using forced ventilation, airflows of the model are increased which would prevent the transmission of COVID-19 in the laboratory rooms. The method of forced ventilation is better than natural ventilation while generating fresh air inside laboratory room.

4. Conclusion

This study was conducted to simulate the airflows without and with force ventilation in a laboratory room. The method was used to prevent the transmission of current COVID-19 virus in a laboratory room. The main objective of the study is to determine the airflow of velocity in a laboratory room by using natural ventilation and forced ventilation. This application could show the accurate air velocity that needed for a laboratory room to prevent the transmission of COVID-19. The application of CFD for simulation is a good option compared to experimental. This is because it could save time and cost which would generate the same results as experimental. Moreover, a strategy as an approach to prevent COVID-19 by using forced ventilation has contributed. In the data analysis, a forced ventilation method is the most efficient ways to increase air velocity which would generate more fresh air into the laboratory room. This resulted as the more user friendly as to its defined functions.

4.1 Recommendation

In order to secure the more detail designed model to be more applicable generate the results of simulation in the future. There are some of the recommendations for further research and improvements needed including the following:

1. The simulation is considered to have more smaller sizing range of the model in meshing which to increase the results to be converges. Furthermore, a specific solution method will have allowed to generate more exact results in iterations.
2. In meshing, the properties of named selection on components should be recorded to avoid mistakes and error before simulate the air flow results.
3. The resulted plane air velocity displayed should be considered more specific which to ensure the calculation of ACH more accurate in advanced.

Acknowledgement

The authors would like to thank the Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] WHO Headquaters, Preventing and mitigating COVID-19 at work. *Schools, businesses and institution*, CC BY-NC-SA 3.0 IGO, 2021.

[2] Richard M. Lynch, “Practical Steps to Improve Air Flow in Long-Term Care Resident Rooms to Reduce COVID-19 Infection Risk”, *Environmental Safety Management*, ELSEVIER, 2020

[3] Jianyun Lu, Jieni Gu, Kuibiao Li, “COVID-19 outbreak Associated with Air conditioning in Restaurant”, *Disease Center and Prevention*, 2020

[4] David A. Madsen, David P. Madsen, “Engineering Drawing and Design”, *Cengage Learning*, ISBN: 1111309574, 2012.

[5] C.F. Tan, B. T. Tee, S. Lin (2015), “Indoor Airflow Simulation inside Lecture Room: A CFD Approach”, Centre for Advanced Research on Energy, Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Retrieved from 2021, https://www.researchgate.net/publication/282305261

[6] Rob Doc Falke (2016), “Use the Air Changes Calculation to Determine Room CFM”, Retrieved from 2021, https://www.contractingbusiness.com/service/article/20868246/use-the-air-changes-calculation-to-determine-room-cfm