A Numerical Study on Indoor Air Quality (IAQ) Designs for Hospital Waiting Room in The Context of COVID-19 For Hot and Warm Humidity Climate

M N Rahman Y1,*, Z M Razlan2, M Nazrin Y2,5, N A A Razali1, M I Izham3, I Ibrahim5, H Desa4, N N Zulkepli4, M A Azizan4

1Faculty of Electronic Engineering Technology (FTKEN), Universiti Malaysia Perlis (UniMAP), Pauh Putra Campus, 02600 Arau, Perlis, Malaysia
2Faculty of Mechanical Engineering Technology (FTKM), Universiti Malaysia Perlis (UniMAP), Pauh Putra Campus, 02600 Arau, Perlis, Malaysia
3Faculty of Applied and Human Sciences (FSGM), Universiti Malaysia Perlis (UniMAP), Jalan Alor Setar-Kangar, 01000 Kangar, Perlis, Malaysia
4Centre of Excellence for Unmanned Aerial Systems (COEUAS), Universiti Malaysia Perlis (UniMAP), Jalan Alor Setar-Kangar, 01000 Kangar, Perlis, Malaysia
5Cawangan Kejuruteraan Mekanikal, JKR Negeri Perlis, KM3 Jalan Raja Syed Perlis, 01000 Kangar, Perlis, Malaysia

*nrurrahman@unimap.edu.my

Abstract. COVID-19 is a virus originated from Corona Virus which can severe acute respiratory syndrome (SARS) symptoms such as chest pain, dry cough, fever, and difficulty breathing. The AC and ventilation system is not only important for the thermal comfort occupants but to ensure the room is safe and free from infectious virus. Thermal comfort is important measurement in indoor space which influenced by temperature, Relative Humidity (RH), airflow velocity and others. This research was executed and focused on lecture room in Bilik Persatuan 10, Universiti Malaysia Perlis (UniMAP) instead of real hospital waiting room. It comes with the room dimensions 11.87m (Length) x 5.17m (Width) x 2.93m (Height) for the numerical study. In addition, Computational Fluid Dynamics (CFD) analysis is used to investigate the air flow pattern and temperature distribution inside the room. By using software Ansys FLUENT 19, field experimental and simulation work can be compared which have 14.55% difference in temperature distribution. It is expected by increasing the air velocity of the AC inlet diffuser influence the pattern of airflow in the room, but average temperature remains same for all these conditions.

1. Introduction

COVID-19 can be spread through coughing, sneezing, and touching the infected surface based on the reported cases reported. In addition, a few possible transmissions of the virus as the virus diameter are from 50 to 200nm [1], such as droplet transmission, close contact and possibly, aerosol transmission. In comparison, airborne transmission refers to microbes with droplet nuclei where the virus particle size is less than 5µm where it can transmit to others over distances greater than 1m as it can remain in the air for an extended time [2]. The airflow distribution and pureness from the air conditioning system is vital as it will affect whether occupants perceive the airflow velocity, RH, and temperature in the aseptic and
comfortable zone. During this pandemic, it is important to measure the thermal comfort in a room and to reduce infection cases if it happened.

2. Literature Review
Warm and hot humidity climate is where the region experiences more than 20cm in (50cm) rainfall, for example is Malaysia. At higher temperatures and higher relative humidity, the virus viability was quickly lost. Besides, wind (airflow velocity) also influences the trajectory of the virus droplets to land on the surfaces. This is based on of the case study where it is founded that air velocity from the Heating, Ventilation, and Air Conditioning HVAC system can cause dispersion of the virus and greater distribution of the contaminated air to circulate farther from the spreader [3]. However, it is also proven that an increase in RH between the recommended range and temperature influenced the reduction of new cases and new deaths where it can be different between the countries [4].

2.1 Design of hospital waiting room
The suggestion to minimize the contamination rates from the airborne pollutants and increase IAQ is by eliminating or reducing the contaminant concentration in the room by applying a few guidelines for designing HVAC systems. The critical design recommendation is to increase the airflow pattern inside the room, improve a more efficient ventilation system and provide additional purification in the air conditioner (AC) return duct or inside the room directly that is rated to collect or remove small particulates. Guidelines from MS1525:2007 recommended for a room to be set at temperature 23-26 with the range of relative humidity at 55%-70% and air velocity at 0.15m/s-0.50m/s [5].

2.2 Gathering Information of room
This study used a lecture room in UniMAP in Figure 1 named Bilik Persatuan 10 where the dimension of room, temperature, RH, and air flow velocity in this room taken by using appropriate tools as shown in Table 1.

| Measurements | Length (m) | Height (m) | Wide (m) |
|--------------|------------|------------|----------|
| Room         | 11.87      | 2.63       | 5.17     |
| Window       | 1.77       | 1.16       | -        |
| Door         | 1.41       | 2.04       | -        |

Figure 1. Sketching of hospital waiting room

The method for measurement of temperature distribution can be done through an Anemometer, an instrument to measure the air velocity of the AC outlet diffuser. At the same time, RH meter is used to measure the temperature and relative humidity of the room. This study also used FLIR Ex-Series infrared cameras to capture any surface thermal image.
2.3 Analysis and simulation
A CFD in the HVAC system helps to determine the parameters such as airflow and surface temperature and airflow speed at any point in the design space. The result can be observed in terms of temperature distribution and local thermal comfort visualizations, revealing that the displacement ventilation energy can benefit lower energy consumption and IAQ [6].

2.3.1 Numerical Analysis. Numerical analysis can be defined as the area of mathematics and computer science that creates, analyses, and implements algorithms for solving the problems of continuous mathematics numerically [7]. Based on the case studies, k-turbulence, Navier-Stokes’s equation, and continuity equation mostly being used in describing the flow in a room which is being computed by using CFD.

3. Methodology
The function of CFD is applied to analyse the thermal properties and modelling airflow by applying mathematics, physics and visualising the gas and liquid flows based on Navier-Stokes Equation and other related equations [8]. Mainly, the software used is Ansys 19.2 and the simulation done by using FLUENT process. Grid Independence Study (GIS) is executed to identify the exact mesh size that is selected in the numerical simulation [9].

3.1 Modelling of room
Figure 2 shows the type of diffuser used in the lecture room is a 4-way cassette air conditioner with dimensions 256mm (Height) x 840mm (Length) x 840mm (Width). However, since the location of the air conditioner diffuser located axisymmetric for the room, assuming the AC's airflow is the same for both sides of the room based on the figure below.

![Figure 2. Geometry of the lecture room with exhaust outlet vent](image)

3.2 Mesh Generation of room
The meshing process is essential for the accuracy, convergence, and speed of the simulation used for finite volume method analysis. The function of the inflation layer meshing in ANSYS is designed to create thin elements that can capture the normal gradient with minimal elements, capturing the velocity and temperature gradient near no-slip walls [10]. The number of nodes is 2,058,119 with 2,859,136 elements.

Besides, the assumptions made for the simulation is such as the AC units in the lecture room is fully functioning and running well, the room is fully sealed and enclosed without any holes or gaps except doors and windows, the outside temperature on the surface of the room is constant, and the internal heat sources emitted from the lights, such as lamps, will be neglected due to minimal effect on the temperature.
3.3 Thermal image inside the room

The FLIR Ex-series infrared camera is used to find the object that generated heat as shown in Table 2. In addition, this thermal imaging camera is helpful to identify the object that radiates heat as illustrated in Figure 3 and 4. The hotter a given object is, the more radiation it will spill into the environment.

| Object                     | Maximum Surface Temperature (°C) |
|----------------------------|----------------------------------|
| Wall Exposed to Outside    | 26.4                             |
| Fluorescence Lamp          | 29.6                             |
| Emergency Light            | 24.9                             |
| Whiteboard                 | 22.2                             |

![Fluorescence lamp thermal image](image1)

![Wall exposed to outside thermal image](image2)

3.4 Experimental data temperature inside the room

The data was taken by measuring each point from 1 meter to 1 meter with the height of the digital thermometer at 1 meter from the floor as shown in Table 3.

| Point | Temperature (°C) | Relative Humidity (%) |
|-------|------------------|-----------------------|
| 1     | 19.6             | 53.2                  |
| 2     | 19.6             | 52.4                  |
| 3     | 19.6             | 52.7                  |
| 4     | 19.6             | 53.1                  |
| 5     | 19.6             | 52.4                  |
| 6     | 19.1             | 52.0                  |
| 7     | 19.1             | 52.0                  |
| 8     | 19.1             | 52.5                  |
| 9     | 19.4             | 54.9                  |
| 10    | 19.0             | 58.2                  |
| 11    | 19.0             | 56.7                  |
| 12    | 15.6             | 62.7                  |
| 13    | 16.0             | 65.5                  |
| 14    | 16.3             | 65.5                  |
| 15    | 17.2             | 65.1                  |
| 16    | 17.7             | 62.5                  |
| 17    | 18.1             | 59.3                  |
| 18    | 16.2             | 57.2                  |
| 19    | 16.3             | 59.7                  |
| 20    | 17.3             | 60.2                  |
| 21    | 17.6             | 58.3                  |
Average room temperature = Total temperature each point/ Number of points = 18.14°C
Average room RH = Total RH each point/ Number of points = 62%

4. Result and Discussion
The simulation result will be compared based on the lecture room designs before and after adding the exhaust outlet (vent). The function of AC’s return air vent is to remove the heat from the inside environment by maintaining optimum temperature and filtering out debris from the room. Therefore, the proposed new design of the lecture is by adding the four exhaust vents with each dimension of 20cm (Length) x 20cm (Width). For instance, the thermal comfort condition and IAQ can be affected by setting up different angles of the inlet vent. Besides, it has been demonstrated that using the CFD method by varying the height of the return air vent can reduce the energy consumption while maintaining the thermal comfort and IAQ within the specified range [10].

4.1 Streamlines inside the lecture room with four return outlets
By inserting four return outlets with a gap of 2.3m, the difference in air velocity and data temperature can be observed. Figure 5 shows that the streamlines are distributed evenly for both sides but more congested on the left side of the room. Due to the design of the air-conditioners with 4-way AC outlet diffusers, it influences the streamline of the airflow in the room.

4.2 Temperature Distribution Inside the Room
Based on CFD approach by inserting the source temperature with 15.5°C, the temperature obtained for the lecture room is 15.5°C (288.65 K) and it is distributed evenly throughout the room as shown in Figure 6.
4.3 Airflow velocity distribution inside the room with four return outlets

The airflow that comes out from the inlet of the air conditioner can be seen in Figure 7 which velocity is about 0.125 m/s. In contrast, the maximum air velocity is 0.5 m/s located near the air conditioners’ inlet. By observing the contour of the plane on each axis, the average air velocity is 0.025 m/s-0.035 m/s, where each plane is in the centre of the room.

![Figure 7. Contour of velocity at XZ axis at 0.5 m/s](image)

4.4 Comparison of experimental data and simulation data on without outlet exhaust model

The air velocity dispersions from the two air-conditioning units only fall at two points of seats as shown in Figure 8. Compare with the model lecture room, which has the return outlets, the dispersion of air velocity creates a better flow of air velocity with ‘air-curtain’ effects. It is demonstrated that there is a 14.55% error in the temperature difference of simulation and experimental data. The percentage difference is due to the inflation, the geometry of the lecture room, meshing, fluid properties, boundary conditions, solver setting, solution methods and the result obtained from the CFD post can all influence the simulation modelling [11].

Percentage difference of temperature distribution inside the lecture room:

\[
\% \text{ Error} = \frac{\text{Experimental data} - \text{Simulation data}}{\text{Simulation data}} \times 100
\]

\[
\% \text{ Error} = \frac{18.14^\circ C - 15.5^\circ C}{18.14} \times 100 = 14.55\%
\]

![Figure 8. Contour of velocity at XZ axis for room without exhaust outlet vent](image)

4.5 Comparison of velocity for model room with different air velocity outlet diffuser

For further comparison, the air velocity outlet diffuser is increased with the increment of 0.25 m/s to find the suitable temperature for the room to achieve maximum thermal comfort as illustrated in Table 4. As the air velocity increased, the airflow pattern becomes more shaper in the form of the ‘air-curtain’ effect. However, increasing too high air velocity at the inlet is not recommended as it can cause turbulence and disrupt the particles in the room. Even though the air velocity is increased, this system does not affect the air temperature and relative humidity.
| No. | Supplied air velocity (m/s) | Average room air velocity (m/s) |
|-----|----------------------------|-------------------------------|
| 1   | 0.50                       | 0.18                          |
| 2   | 0.75                       | 0.28                          |
| 3   | 1.00                       | 0.37                          |

4.6 Grid independence study of the simulation result

Figure 9 shows four different meshing nodes are examined by using the different numbers of the element size. When the number of nodes is increased, the average temperature at Plane 1, which is at the Z-axis, decreases, achieving the desired temperature in the room. Furthermore, since the number of nodes is increasing, the accuracy of the calculation also increases which can achieve the thermal comfort condition requirements.

![Figure 9. Average temperature at Plane 1 vs Number of nodes](image)

5. Conclusion

During this pandemic COVID-19, there is a need for changes of social distancing and a new IAQ implementation system of HVAC to reduce the risk of virus infection between the occupants in the hospital waiting room. To achieve cold air thermal comfort and pureness of the room, the parameters that need to be controlled are air temperature, RH, air velocity, pressure, and airflow circulation pattern. A lecture room in UniMAP is used for experimental analysis of air distribution throughout the room. The dimensions of the room are 11.57m (Length), 5.17m (Width) and 2.63m (Height). There are two AC units installed in the room, which are 4-ways cassette type. The percentage difference of temperature between the experimental and simulation result is 14.55%. Next, the second model room with four exhaust outlets is applied in the Ansys for the new proposed design to achieve better thermal comfort and improve air cleanliness. As the air velocity increase, the airflow pattern becomes more curve. In addition, high air velocity may cause turbulence in the room and increasing the risk of infections. Therefore, it is recommended to keep the air velocity between 0.3m/s to 0.5m/s with temperature 16°C to 18°C to achieve thermal comfort conditions in the room.

References

[1] Marcelo I Guzman 2020 An overview of the effect of bioaerosol size in coronavirus disease. *Int J Health Plann Mgmt* 1-10.

[2] Yu Feng, Thierry Marchal, Ted Sperry and Hang Yi 2020 Influence of wind and relative humidity on the social distancing effectiveness to prevent COVID-19 airborne transmission: A numerical study. *Journal of Aerosol Science*, 147.
[3] Layeni A, Nwaokocha C, Olamide O, Giwa, S, Tongo S, & Onabanjo O et al. 2020 Computational Analysis of a Lecture Room Ventilation System. *Zero-Energy Buildings - New Approaches And Technologies*.

[4] Mecenas P, Bastos R, Vallinoto A, & Normando D. 2020 Effects of temperature and humidity on the spread of COVID-19: A systematic review. *PLOS ONE* 15 (9).

[5] CODE OF PRACTICE ON ENERGY EFFICIENCY AND USE OF RENEWABLE ENERGY FOR NON-RESIDENTIAL BUILDINGS (FIRST REVISION), 2007, *MS 1525 2007*.

[6] Role of CFD Simulation in HVAC System Design, HPAC Engineering 2018. [Online] https://www.hpac.com/fire-smoke/article/20929589/role-of-cfd-simulation-in-hvac-system-design.

[7] C Popovici and V Hudişteanu 2016 Numerical Simulation of HVAC System Functionality in a Sociocultural Building, *Procedia Technology*, 22, 535-542.

[8] An introduction to CFD: what, why and how. Femto Engineering. 2021 [Online] https://www.femto.eu/stories/what-is-cfd/#.

[9] D Zhao, N Han, E Goh, J Cater and A Reinecke 2019 3D-printed miniature Savonious wind harvester, *Wind Turbines and Aerodynamics, Energy Harvesters*, 42.

[10] Tips & Tricks: Inflation Layer Meshing in ANSYS Computational Fluid Dynamics (CFD) Blog – LEAP Australia & New Zealand, Computationalfluiddynamics.com.au, 2012. [Online] https://www.computationalfluiddynamics.com.au/tips-tricks-inflation-layer-meshing-in-ansys/.

[11] S Lin, B Tee and C Tan 2015 Indoor Airflow Simulation inside Lecture Room: A CFD Approach, *IOP Conference Series: Materials Science and Engineering*, 88, 012008.