CFD Simulation of Addition Mechanical Ventilation in Health Clinic Room with Various Number of Exhaust Fan

I Putu Widiarta[1]
[1]*(Mechanical Engineering Department, Udayana University, Bali, Indonesia)*

ABSTRACT: *The Covid-19 pandemic is currently still being faced in the world, so it is necessary to control the spread, especially controlling the spread in health clinics. Controlling the spread of viruses through the air can be done by installing mechanical ventilation that is because light airborne particles are expelled by an exhaust fan. the addition of a mechanical ventilation system makes air exchange occur and air exchange can form an airflow pattern. The flow pattern formed is very important to analyze because light airborne particles tend to follow the airflow pattern. To know the flow pattern can be done by CFD simulation. In this study, the CFD simulations were carried out with a transient time by varying the number of exhaust fans and the angle of the air conditioner flapper. The simulation result shows without the addition of a mechanical ventilation system the vortex is occur caused by the circulation of the air conditioner blower and the angle of the air conditioner flapper provides a different flow direction. Meanwhile, the addition of a mechanical ventilation system can produce air exchange and vortex flow becomes superposition flow. With the air exchange, the risk of transmitting the virus through the air can be reduced. In addition, air exchange can cause the room temperature to increase. Besides that, variations in the number of exhaust fan cause variations in the pressure value where the greater number of exhaust fans installed, the pressure becomes decrease while the velocity value becomes increase.*

KEYWORDS*- Mechanical ventilation system, CFD simulation, airflow pattern, pressure and temperature.*

I. INTRODUCTION

The health clinic is the place of the health facilities service. The building of health clinics has a large potential virus spread through the air or airborne infection, especially in the conditions of the covid-19 pandemic that the world is still facing today [1]. Controlling the spread of the airborne infection can be done by optimizing the ventilation system [2]. In general, ventilation systems are used to improve indoor air quality as physical and chemical [3]. Ventilation rate is caused by the pressure difference, the value of pressure difference is influence to value of ventilation rate [4]. The ventilation system can be the classification into two types that is natural ventilation and mechanical ventilation [5]. Natural ventilation is the ventilation where the movement of air is natural manner while mechanical ventilation is needed tools such as a fan to create of ventilation rate. In general, mechanical ventilation uses an exhaust fan to create a ventilation rate [6]. The Health Ministry of Indonesia has been creating the standard value of ventilation rate where the ventilation rate must be reaching the minimum value of air change hour (ACH) is 6 per hour [7].

The addition of a mechanical ventilation system aims to circulate and exchange indoor air where the dirty air is expelled through the exhaust fan channels and the fresh air enters through the inflow channels [8]. The air entering and leaving the room can form of air flow pattern [9]. The airflow pattern is important to analyze because the movement of light airborne particles tends to follow the airflow pattern [10].

The role of the HVAC (Heating Ventilating and Air Conditioning) system during the pandemic covid-19 is very important because the ventilation system can be reduced airborne particles in room so that the risk of airborne infection can be reduced [11]. In addition, a filtration system is very important, where the filter is keeping the entering air quality [12].

The addition of airflow exchange using mechanical ventilation in the room where the indoor air is conditioned by an air conditioner can be increasing the cooling load because the air with low temperature is expelled by an exhaust fan and the air with high temperature is entered through inflow channels and then the indoor air temperature becomes highly [13]. Increasing indoor air temperature can influence the consumption of electric energy because the air conditioner can not reach the air temperature setting so the air conditioner is still operated [14]. The value of indoor air temperature is effect to the room thermal comfort. Indonesian National Standard (SNI) has been creating the standard value of thermal comfort with the range temperature is 20.5°C – 27°.8C [15].

Knowing the airflow pattern with the addition of mechanical ventilation can be done by using the computational fluid dynamics (CFD) method [16]. The CFD method is relevant to obtaining the physic phenomena of airflow such as flow pattern, velocity, pressure, vector, and temperature [17]. CFD has the variant of type simulation that is the steady simulation and transient simulation. The steady simulation is a fluid flow simulation where the flow time is negligible while the transient simulation is a fluid flow simulation that is considered the flow time [18]. As we know, the air is very easier to deform after receiving stress so time can affect the formation of flow patterns [19]. Thus, it is necessary to carry out a transient CFD simulation of the addition of mechanical ventilation in the health clinic room to obtain and analyze the physical phenomena.

II. METHOD

2.1 Research Schematic

In this study, the clinic that was used as the object is the Bayu Suta Medical Service clinic. The schema of the clinic is shown in Figure 1.

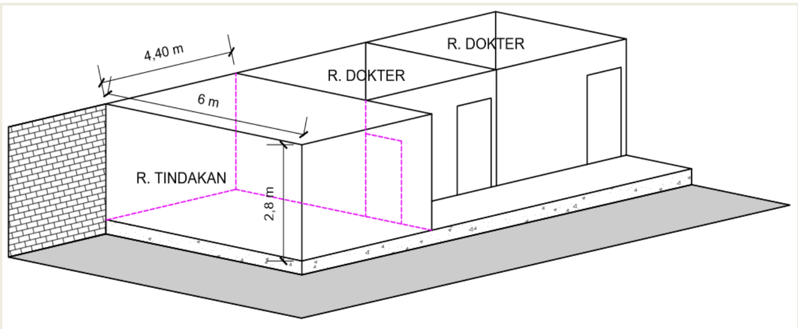


Figure 1. Scheme of health clinic.

The treatment room (R.TINDAKAN) was chosen as the object for the CFD simulation because the treatment room was not equipped with a ventilation system and the schematic addition of mechanical ventilation has been illustrated in Figure 2.

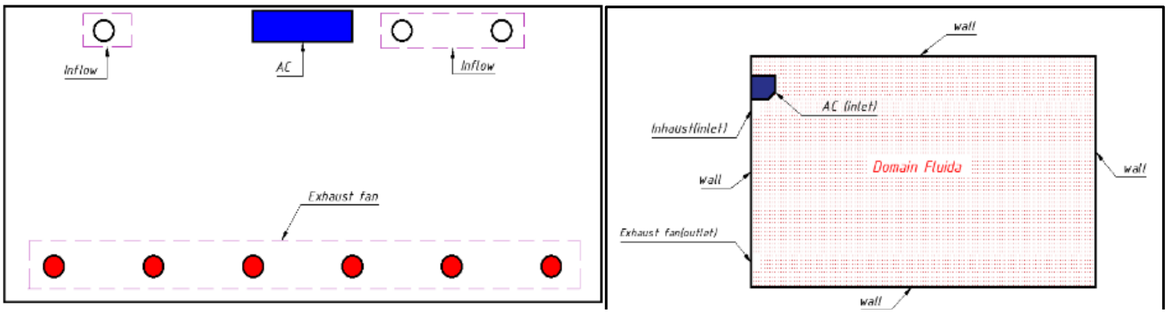


Figure 2. Schematic of mechanical ventilation addition.

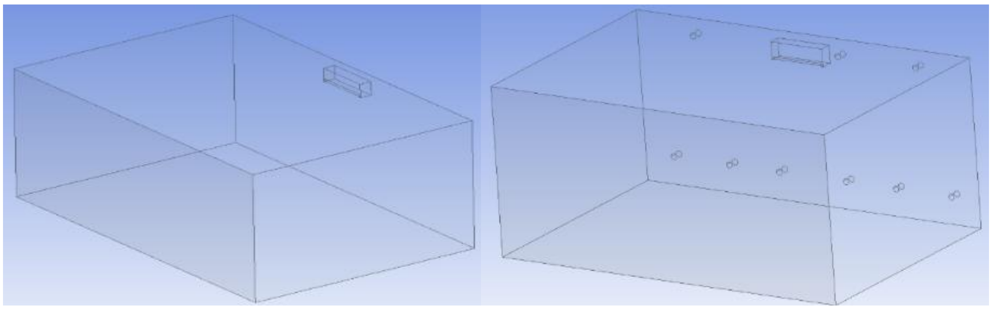
Fig. 2 is shown the schematic of the mechanical ventilation addition. In this case, the type of air conditioner is used split type and the flapper AC (fan guide) is varied, which is 40°, 50°, and 60°. In addition, the number of exhaust fans also varied, namely 2, 4, and 6. In this case, the first simulation is a clinic room without the addition of ventilation and the next is a clinic room with the addition of mechanical ventilation.

2.2 CFD simulation.

The CFD or computational fluid dynamic simulation using Ansys Fluent software. In general, the CFD simulation consists of 4 processes, namely geometry, mesh, physic, and result.

2.2.1 Geometry.

Simulation used negative part or only fluid domain without solid. CFD technique with negative part aims to save the number of mesh so that the iterations process becomes faster and less memory needed. Fluid domain of computation as shown in Figure 3.



(a) (b)

Figure 3. Fluid domain. (a) fluid domain without ventilation, (b) fluid domain with ventilation.

The dimension of fluid domain is 4.48m x 2.80m x 6m, thus the volume of fluid domain is 75.26m^3, dimension of fluid domain follows the dimension of health clinic that is choose as an object on this research.

2.2.2 Meshing.

Meshing is the process of enumerating fluid domains into a grid. The number of grids of the mesh greatly affects the simulation result i.e., the convergence value and accuracy of the result. Smaller grid size of the mesh, the simulation result becomes smoother and accuracy becomes high, and also the convergence standard value can be achieved. Type Mesh used in this research is poly hex-core type mesh where this mesh is a combination of polyhedral and hex-core type mesh. The advantages of poly hex-core mesh are the relatively smaller number of mesh and high accuracy [20]. Poly hex-core mesh is shown in Figure 4.

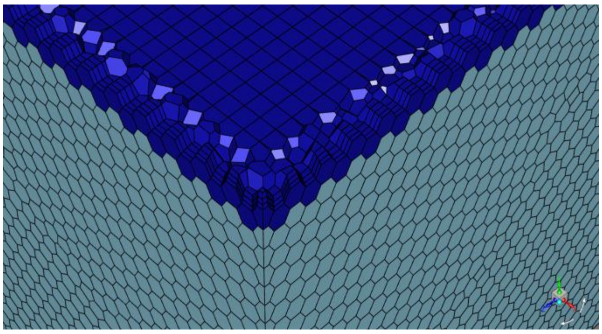


Figure 4. Poly hex-core mesh.

Mesh is set with a minimum cell's length is 0,005859m and maximum cells length is 0,04687m. The summary of the meshing result is shown in Table 1.

Table 1. Summary of the meshing result.

No	Parameter	Value
1	Cells	867.700
2	Faces	3.287.310
3	Nodes	1.636.738
4	Volume statistic:	
	Maximum volume (m^3)	2.28e-04
	Minimum volume (m^3)	4.21e-09
5	Face area statistic:	
	Maximum face area (m^2)	3.26e-03

Minimum face area (m^2)	9.23e-08

2.2.3 Solution.

Simulation of fluid flow is solved by governing equations that are continuity equation, momentum equation, and energy equation [21]. While the turbulence model used is the RANS (Reynolds Average Navier Stokes) turbulence model Namely K-epsilon (K-ε). The RANS turbulence model is used to solve average turbulence flow []. The k-epsilon turbulence model has two-equation namely turbulence kinetic energy (k) and turbulence dissipation rate (ε). Turbulence kinetic energy has been modeled as an equation (1) [22].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial t}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij} - \rho\varepsilon \qquad (1)$$

While turbulence dissipation rate has been modeled by equation (2) [22]

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial t}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right] + C1_\varepsilon\frac{\varepsilon}{k}2\mu_t E_{ij} - C2_\varepsilon\frac{\varepsilon^2}{k} \qquad (2)$$

Where x is the coordinate x-directional axis, ρ is the density of the fluid, E_{ij} is the deformation of fluid, and μ_t is eddy viscosity. Eddy viscosity has been modeled by equation (3) [23].

$$\mu_t = \rho C_\mu \frac{K^2}{\varepsilon} \qquad (3)$$

Equation (3) involves a constant that can be set it is σ_k, σ_ε, $C1_\varepsilon$, $C2_\varepsilon$. The value of this constant can be lowered by a number of fittings for the turbulence flow wide range, namely: C_μ=0,09; σ_k= 1,0; σ_ε= 1,3; $C1_\varepsilon$= 1,44; $C2_\varepsilon$=1,92 [24].

The solution method or solver used in this simulation is the SIMPLE scheme (Semi Implicit Method for Pressure Linked Equation), which is the numerical procedure for solving the Navier-Stokes equation that uses the relationship between velocity and pressure to obtain mass conservation and pressure field value [25].

2.2.4 Transient CFD Simulation Setup.

The transient CFD simulation used several parameter settings among them as shown in Table 2.

Table 2. Parameter settings.

Parameters	**Settings**
Solver type	Pressure based
Pressure formulation	Pressure gauge
Velocity formulation	Absolute
Time	Transient
Gravity	Y-axis (-9.81m/s^2)
Energy equation	On
Turbulence model	k-epsilon (standard with wall function)
Fluid density and viscosity	Fluent data base
Boundary conditions:	
1. inlet	Velocity inlet
2. outlet	Pressure outlet
3. wall	No slip condition
Solver scheme	SIMPLE
Solution initialization	Standard
Run calculation:	

1. Iteration number	6000
2. Time scale factor	0.02
3. Flow time	60 seconds

Initial value that is inputted at the boundary conditions (pressure outlet, velocity inlet, and temperature) using experimental data that has been measured.

III. RESULT AND DISCUSSION

3.1 Simulation without addition of mechanical ventilation.

Simulation result without the addition of a ventilations system is carried out by varying the flapper angle of the air conditioner. Simulation results that is velocity contour are shown in Figure 5.

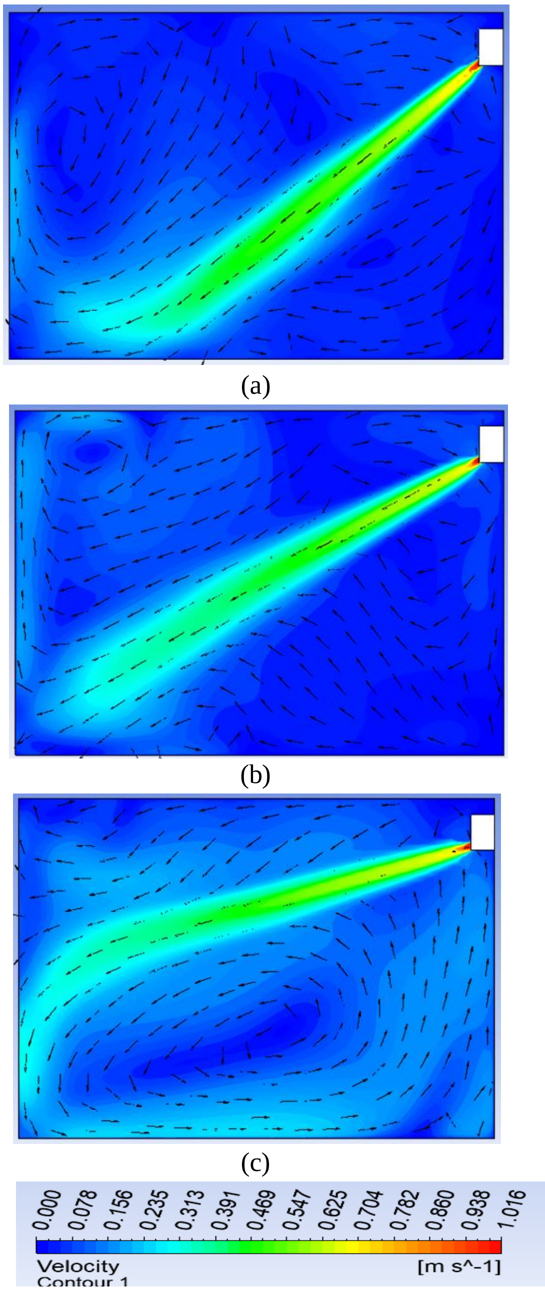


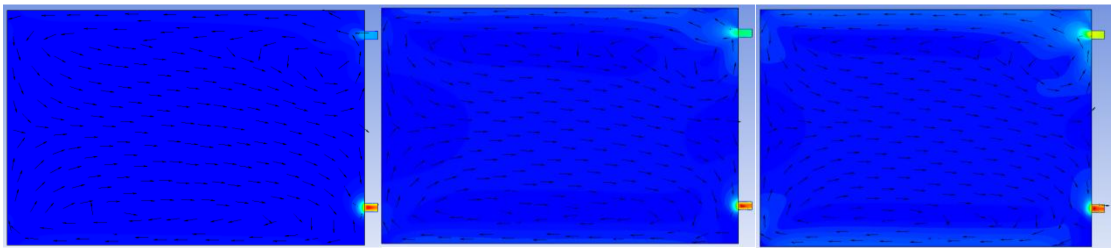
Figure 5. Vector and velocity contour without addition of ventilation system. (a) flapper angle 40, flapper angle 60, (c) flapper angle 80.

Figure 5 shows vector and velocity contour, the vector has been set using flow directional and the color of the vector is set constant variable. In general, without the addition of a ventilation system, the flow is continuously circulated in the room without any air changes. This condition is very dangerous because light particles airborne circulated continuously in the room so it is very dangerous for patients or medical personnel thus the risk of airborne infection becomes high. Therefore, it is necessary to add a ventilation system to create air exchange in the room and the risk of transmitting the virus through the air can be minimized. Meanwhile, the AC flapper angle influences on the input flow, causing a flow vortex to occurs in the room and the vortex occur at different positions due to variations in the flapper angle.

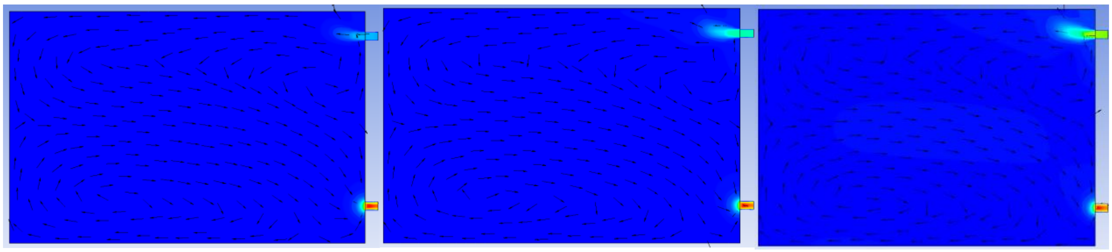
From the simulation result, the room temperature obtained is 22°C, where this value already meets the thermal comfort standards. The temperature simulation results shown the same result as the AC temperature this is due to the absence of additional cooling load.

3.2 Transient simulation with addition of ventilation system.

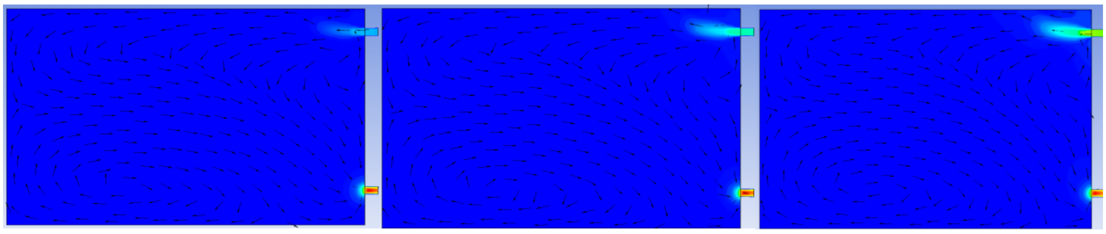
Transient simulations with the addition of a ventilation system are carried out by varying the number of exhaust fans. The simulation was carried out with a flow time of 60 seconds. Simulation results are shown in Figure 6.



(3in 2ex at time= 1s) (3in 4ex at time= 1s) (3in 6ex at time= 1s)



(3in 2ex at time= 30s) (3in 4ex at time= 30s) (3in 6ex at time= 30s)



(3in 2ex at time= 60s) (3in 4ex at time= 60s) (3in 6ex at time= 60s)

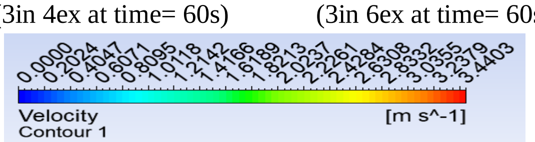


Figure 6. Vector and velocity contour at flow time 1s to 60s.

Figure 6 shows the result of transient simulations with variations in the number of exhaust fans the velocity vector is used for directional flow and the color of the vector is set constant variable. Each variation of the exhaust fans shows the exchange of air through the inflow and exhaust fan mechanism. With the exchange of air, the airborne particles in the room can be removed through the exhaust fan and fresh air enters through the inflow channels so that the risk of airborne virus transmission can be reduced.

From simulation results, the varying number of exhaust fan produce a difference velocity of the airflow that enters through the inflow channels this is caused by the different in pressure generated by the exhaust fan thus the more of number exhaust fan is added then the air indoor pressure becomes decrease due to the large mass flow of air is expelled from the room. The pressure difference is used to make air enters naturally through the inflow channels so that the greater value of pressure differences the velocity of airflow through inflow channels can be increased. The simulation results show that the variation in the number of 6 exhaust fans produces the highest airflow velocity value compared to variations of 2 and 4 exhaust fans.

On the odder hand, the large value of the mass flow of goes out through exhaust can produce a large value of air change hour or ACH. A high ACH value can accelerate the reduction of airborne particles in the room because more air mass is expelled so that light airborne particles are also expelled along with the air as well as the movement of airborne particles following the flow pattern that occurs [10, 26].

The simulation results also show that time affects the airflow pattern. Each exhaust fan variation in the 1st second shows that air is starting to be sucked by the exhaust fans. Then at 30 seconds, it looks like the air has started entering through inflow channels where the variations of 6 exhaust fans produce the highest air velocity entering the room and the exhaust fan still sucks the room air and the airflow pattern seen from velocity vector has shown a change in a pattern where the air begins to focus on moving towards the exhaust fans. At 60 seconds the air around the exhaust fan has focused on moving towards the exhaust fan channels. The difference in flow pattern from variations in the number of exhaust fans is only on the value of the flow velocity. From the simulation result, the average value of pressure in each variation is shown in Figure 7.

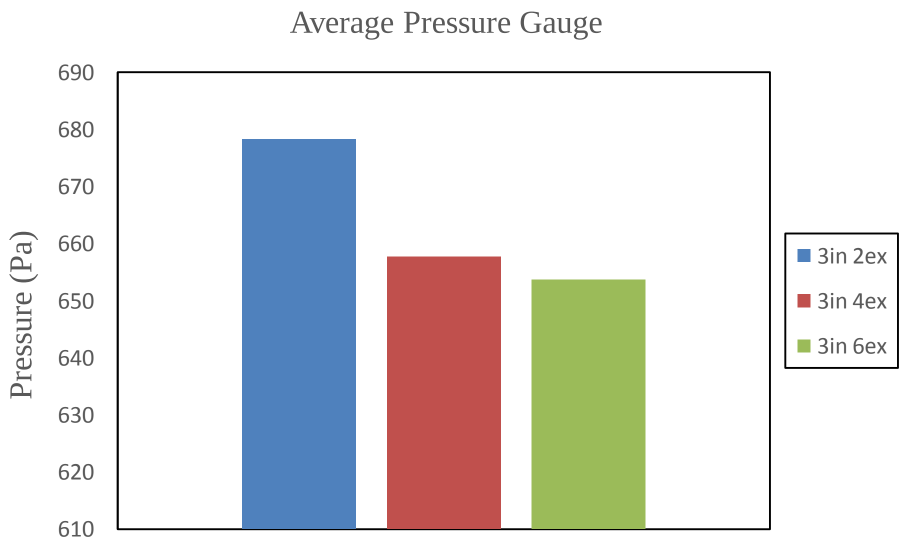
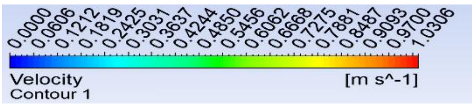


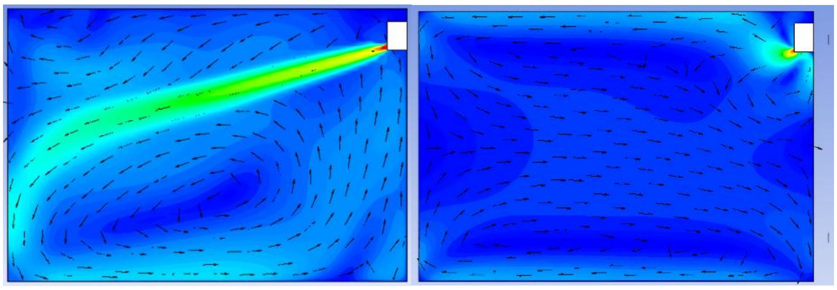
Figure 7. Average pressure gauge.

Figure 7 shows the value of the average pressure difference with variations in the number of exhaust fans. The graph in Figure 7 shows that the more exhaust fan installed, the average pressure in the room becomes lower so that variations of 6 exhaust fans can produce a high flow rate and high ACH (Air Change Hour) value but the decrease in the pressure value is not too significant thus, the standard pressure for occupant to breathing activity still can be achieved.

3.3 Flow pattern with and without the addition of ventilation system.

With the addition of a ventilation system, a different flow pattern occurs, where the flow pattern is viewed at the center position of the building as shown in Figure 8.





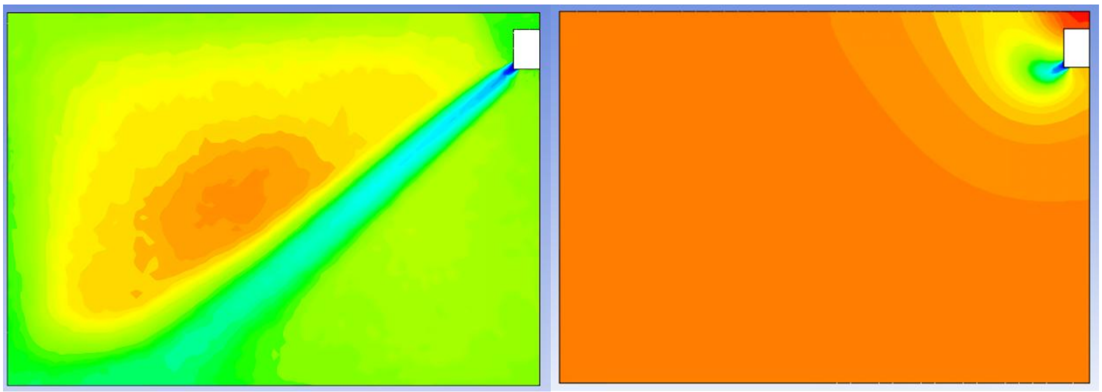
(a) (b)

Figure 8. Vector and velocity contour at center position. (a) without ventilation, (b) with addition mechanical ventilation.

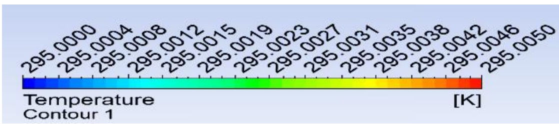
Figure 8 shows that there is a difference in flow patterns. With the addition of a ventilation system, the vortex resulting from the circulation of the air conditioner (AC) blower turns into superposition flow. This change in flow pattern is caused by the effect of the exhaust fan which continues to suck air in the room and the flow that enters through the inflow. In addition, the air conditioner flapper angle (guide fan) is no longer significant in influencing the formations of the flow pattern.

3.4 Temperature difference with and without addition of ventilation system.

The simulation result shows that there is a difference in temperature contour under conditions without and with the addition of a ventilation system as shown in Figure 8.



(a) (b)



Figrure 8. Temperature contour. (a) without addition mechanical ventilation, (b) with addition mechanical ventilation.

Figure 8 shows the temperature difference without and with the addition of a ventilation system there is an increased temperature value due to the addition of a ventilation system (inflow dan exhaust fan), this is caused by the addition of heat into the room so that the temperature becomes high. The mechanism of added heat is the presence of outdoor air with high temperature entering through the inflow and conditioned air being expelled through an exhaust fan so that the room temperature becomes high and the consumption of electric power becomes increases because the Air Conditioner (AC) can not reach the set temperature.

The temperature generated by the addition of the ventilation system still meets the standard of thermal comfort that has been determined by Indonesia Standard National [15].

IV. CONCLUSION

From the result of this study, the things that can be concluded are: flow pattern without the addition of a ventilation system and variations in the angle of the air conditioner flapper occur vortex and vortex occur at the different positions due to different direction of flow from the air conditioner. With the addition of a ventilation system occur air exchange, while the variations of 6 exhaust fans produce the highest flow velocity value compared to variations of 2 and 4 exhaust fans. On the odder hand, with the addition of a ventilation system, the vortex flow can be reduced becomes superposition flow and the flapper angle of the air conditioner is no longer significantly contributes to the formation of a flow pattern. The addition of mechanical ventilation also can increase the value of the room temperature, but the risk of transmitting the virus through the air can be reduced.

REFERENCES

[1] Laimore, S, Stone, K, E, Huang, R, McLeight, C 2021, 'Infectious screening in a dedicated primary care clinic for children in foster care', Child Abuse & Neglect, vol. 117, DOI: https://doi.org/10.1016/j.chiabu.2021.105074.

[2] Deng, X, Gong, G, He, X, Shi, X, Mo, L 2021, 'Control of exhaled SARS CoV-2-laden aerosol in the interpersonal breathing microenvironment in a ventilated room with limited space air stability', Journal of Environmental Science, pp. 1-29, DOI: https://doi.org/10.1016/j.jes.2021.01.025.

[3] Kim, M. H, Baek, K. O, Park, G. G, Jang, J. Y, Lee, J. H 2020, 'A study on concentration, identification, and reduction of airborne microorganism in the miltary working dong clinic', Safety and Heat at Work, Vol. 11, pp. 517-525.

[4] Emmerich, S. J, Heinzerling, D, Choi, J, Persily, A. K 2013, 'Multizone modeling of strategies to reduce the spread of airborne infectious agent in healthcare facilities', Building and Environment, vol. 60, pp. 105-115, DOI:http://dx.doi.org/10.1016/j.buildenv.2012.11.013.

[5] Hajdkukiewics, M, Geron, M, & Keane, M 2013, 'Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room', Building and Environment, Vol. 70, PP. 73-89.

[6] Chen, C. Y, Chen, P. H, Chen, J. K, Su, T. C 2021, 'Recommendations for ventilation of indoor space to reduce Covid-19 transmission', Journal of Formosan Medical Association, vol. 120, pp. 2055-2060.

[7] Ministry of Health of the Republic of Indonesia 2016, 'Regulation of minister of health of the Republic of Indonesia Number 24 concerning technical requirements for building and hospital'.

[8] Ho, S. H, Rosario, L, & Rahman, M. M 2009, 'Three-dimensional analysis for hospital operating room thermal comfort and contaminant removal', Applied Thermal Engineering, Vol. 29, pp. 2080-2092.

[9] Kumar, N, Kubota, T, Tominaga, Y, Zhirzadi, M, Bardhan, R 2021, 'CFD simulation of wind-inducted ventilation in appartement building with vertical voids', Building and Environment, Vol. 194.

[10] Cheong, C. H, Park, B, Ryu, S. R 2021, 'Effect of under-floor air distribution system to prevent the spread of airborne pathogen in classrooms', Case Studies in Thermal Engineering, vol. 28.

[11] Zhou, Y & Ji, S 2021, 'Experimental and numerical study on the transport of droplet aerosol generated by occupants in fever clinic', Building and Environment, vol. 187, DOI: https://doi.org/10.1016/j.buildenv.2020.107402.

[12] Sodiq, A, Khan, M. A, Naas, M, Amhamed, A 2021, 'Addressing COVID-19 contagion through the HVAC systems by reviewing indoor airborne nature of infectious microbes: Will an innovative air recirculation concept provide a practical solution', Environmental Research, vol. 199.

[13] Khatri, R, Khare, V. R, Kumar, H 2020, 'Spatial distribution of temperature and airflow analysis in radiant cooling system using CFD technique', Energy Report, Vol. 6, pp. 268-275.

[14] Ascione, F, Masi, R. F. D, Mastellone, M, Vanoli, G. P 2020, ‘The design of safe class rooms of educational building facing contagions and transmission of disease: A novel approach combining audits, calibrated energy model, building performance, and computational fluid dynamic simulation’. Energy and Buildings, vol. 230, DOI: https://doi.org/10.1016/j.enbuild.2020.110533.

[15] Indonesian Standard National Number 03-6572-2011, ‘Regarding procedures for planning ventilation and air conditioning systems in building’.

[16] Shirzadi, M, Mirzaei, P. A, Tominaga, Y 2020, ‘RANS model calibration using stochastic optimization for accuracy improvement of urban air flow CFD modeling’, Journal of Building Engineering, vol. 32, DOI:https://doi.org/10.1016/j.jobe.2020.101756.

[17] Zheng, X, Montazeri, H, Blocken, B 2020, ‘CFD simulations of wind flow and mean surface pressure for building with balconies: Comparison of RANS and LES’. Building and Environment, vol.173, DOI: https://doi.org/10.1016/j.buildenv.2020.106747.

[18] Nguyen, V. T, Danlos, A, Ravelet, F, Deligant, M, Solis, M, Khelladi, S, Bakir, F. 2021,’Numerical analysis of a novel twin-impeller centrifugal compressor’, Computation, vol. 9, No. 143, pp. 1-17.

[19] Munson, B, Young, D. F, Okiishi, T. H, Huebsch, W. W 2009, ‘Fundamentals of fluid mechanics’ John Willey and Sons, Inc, six editions.

[20] Karkouilas, D. G, Tzoganis, E. D, Panagiotopoulos, A. G, Acheimastos, S. G. D, Margaris, D. P 2022, ‘Computational Fluid Dynamics study of wing in airflow and air solid-flow using three difference meshing technique and comparison with experimental result in wind tunnel’, Computation, Vol. 10, No.34, pp. 1-24.

[21] Versteeg H. K, Malalasekera, W 2009, ‘An introductions to computational fluid dynamics’, second edition, Pearson Education.

[22] Johns, W % Launder B 1972, ‘The prediction of laminarization with a two-equation model of turbulence’, Journal of Heat and Mass Transfer, Vol. 15, pp. 301-314.

[23] Liu, Y, & Hinrichsen 2014, ‘Study on CFD-PBM turbulence closure based on k-epsilon and reynolds stress model for heterogeneous bubble column flow’, Computer and Fluid, Vol. 105, pp. 91-100.

[24] Qi, Y & Ishihara, T 2018, ‘Numerical study of turbulent flow around of a row of trees and an isolated building by using modified k-epsilon model and LES model’, Journal of Wind Engineering & Industrial Aerodynamics, vol.177, pp. 293-305.

[25] Anderson, J. D. Jr 1995, ‘Computational fluid dynamic: The basic with application’. McGraw-Hill, international edition.

[26] Ren, F. Y, Huang, Q, Marzouk, T, Richard, R, Pembroke, K, Martone, P, Venner, T, Malmstrom, H, Eliav, E 2021, ‘Effect of mechanical ventilation and portable air cleaner on aerosol removal from dental treatment rooms’, Journal of Dentistry, vol. 105, DOI: https://doi.org/10.1016/j.jdent.2020.103576.