International Journal of Application on Sciences, Technology and Engineering (IJASTE)
Volume 1, Issue 2, 2023. ISSN:2987-2499

Analysis of Thermal Comfort in L 609 Classroom by Using SST *k-ω* Turbulence Model

Ryo Gozali[1,a)], Steven Darmawan[2,b)], Harto Tanujaya[3,c)]

Author Affiliations
[1,2,3)]*Universitas Tarumanagara*

Author Emails
[a)]*ryogozali.515180025@stu.untar.ac.id*
[b)]*stevend@ft.untar.ac.id*
[c)]*hartotan@ft.untar.ac.id*

Submitted: January-February 2023, Revised: March 30 2023, Accepted: May 25, 2023

Abstract

In today's era, more and more people spend most of their time indoors, with artificial climatic conditions, where thermal comfort is a basic factor such as a glazing system (ventilation), both for dimensions and material characteristics, this basic factor is very important because it affects the parameters involved in thermal comfort. One of the requirements for thermal comfort is good air circulation, good air circulation can also prevent the spread of the COVID-19 virus that is currently engulfing the world.Therefore, research is needed to analyze the air flow engineering in the room, especially the classroom to get results that will later be seen whether it has met the demands of room comfort, especially thermal comfort. The purpose of this study is to provide recommendations for the configuration of air circulation in the L609 classroom and perform a CFD analysis on air circulation in the classroom which can create thermal comfort. Based on all the simulation results carried out with 2 outlet configurations and 5 different speed fan configurations, the obtained results are in accordance with the ASHRAE 55 standard in 2020 only in the first configuration with a speed of 0.79 m/s. in the second configuration all the simulation results obtained are not in accordance with the thermal comfort limit according to ASHRAE 55 in 2020 because the speed obtained is more than 0.8 m/s and the temperature obtained is more than 297 K. Considering the COVID-19 pandemic, there must be circulation air in order to break down the aerosol concentration of the COVID-19 virus itself. Therefore, the author recommends using a classroom configuration with an open door so that air circulation can flow properly.

Keywords: thermal comfort, CFD, air circulation.

1. INTRODUCTION

In today's era, more and more people spend most of their time indoors, with artificial climatic conditions, where thermal comfort is a basic factor such as a glazing system (ventilation), both for dimensions and material characteristics, this basic factor is very important because it affects the parameters involved in thermal comfort. Indoor airflow is also very complex for a number of reasons, Also, indoor airflow is associated with three-dimensional and temperature instability. With so many complexities in the flow of air in the room, it is impossible to obtain an analytical solution to the differential equation. Analysis (CFD) is needed to get the solution of these equations.[1]

In addition to affecting thermal comfort, air circulation can also affect the speed of the spread of the virus that is hitting the world today, namely COVID-19. The process of spreading Covid-19 does not stop in public spaces, but also stalks us when we are in a room that has poor ventilation. In preventing the spread of COVID-19 indoors, the use and ventilation conditions in a room must

https://doi.org/10.24912/ijaste.v1.i2.473-483

get good attention. This is because, the more the room is closed or has poor air circulation, the process of spreading COVID-19 will be very easy to occur. Conversely, if there is a room that has good ventilation and air exchange, it will further minimize the process of spreading COVID-19 in the room.[2]

As the next step of this thermal comfort analysis research, researchers will continue the research that has been done by other researchers regarding thermal comfort with the assumption of loads from users and objects in the room. then the simulation is run in the form of a temperature profile, pressure profile, air velocity profile along with a streamline that shows air movement. to get results that will later be seen whether it has met the demands of space comfort, especially thermal comfort or not. The purpose of this research is to present a 3-dimensional model of the L609 classroom and perform a CFD analysis of the air circulation in the classroom to achieve thermal comfort based on ASHRAE 55 thermal comfort standard.[3]

2. RESEARCH METHOD

This research consists of two main method including direct measurement of the flow at L609 classroom and numerical method which uses the direct measurement results to subjected geometry. Geometry model of the L 609 provided by Autodesk Fusion 360 and numerical method using CFD conducted with Fluent on Ansys Fluent 2021 Academic Version as shown in Figure 1.

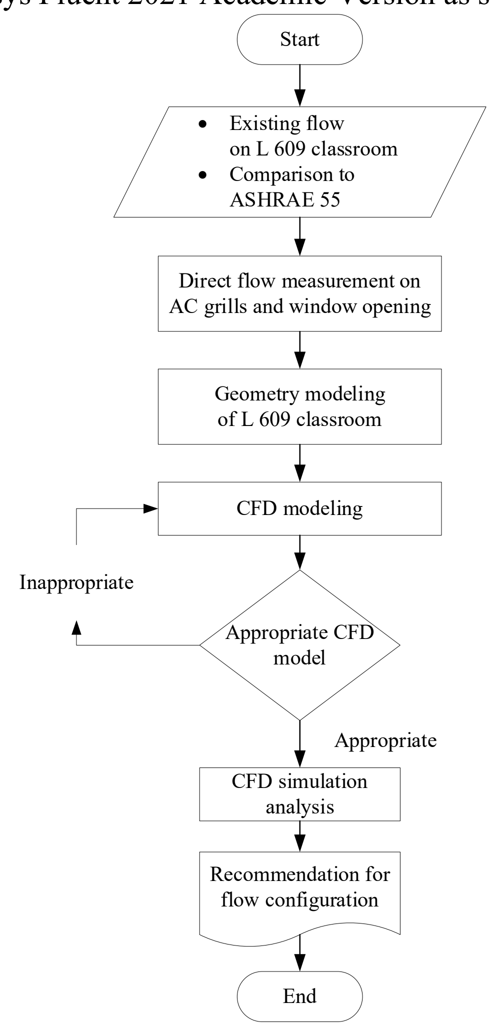


Figure 1. Research flowchart.

https://doi.org/10.24912/ijaste.v1.i2.473-483

In carrying out the analysis to incompressible and inviscid fluid flow, Bernoulli equation based on the Law of Conservation of Mechanical Energy and Pressure is used as well as Reynolds number as reference for fluid velocity [4], [5].

$$P_1 + \frac{1}{2}\rho\vartheta_1^2 + \rho g h_1 = P_2 + \frac{1}{2}\rho\vartheta_2^2 + \rho g h_2 \qquad (1)$$

$$Re = \frac{\rho.\vartheta.d}{\mu} \qquad (2)$$

The k-ω SST turbulence model [Menter 1993] is a two-equation eddy-viscosity model that has become very popular. The shear stress transport (SST) formulation combines the best of both worlds. In Menter's enhanced models, the k-omega . model combined with the k-epsilon model so that it has good capabilities in the area around the wall and a low Reynolds number as an advantage of k-omega, and the flow with a low adverse pressure gradient and its insensitivity in the free stream area which is the advantage of k-epsilon. The following is the equation of SST k-ω. [6], [7]

$$\frac{\partial}{\partial_t}(\rho k) + \frac{\partial}{\partial_{x_i}}(\rho k u_i) = \frac{\partial}{\partial_{x_j}}\left[\tau_\omega \frac{\partial k}{\partial_{x_j}}\right] + G_k - y_k + S_k \ldots\ldots(3)$$

$$\frac{\partial}{\partial_t}(\rho\underline{\omega}) + \frac{\partial}{\partial_{x_j}}(\rho\underline{\omega}u_j) = \frac{\partial}{\partial_{x_j}}\left[\tau_\omega \frac{\partial \varepsilon}{\partial_{x_j}}\right] + G_\omega - Y_\omega + D_\omega + S_\omega \ldots\ldots(4)$$

The purpose of this simulation is to determine the flow pattern in the classroom. To be able to start this simulation, there are 3 stages that will be passed including: pre-processing, processing, post-processing. Direct measurement of fluid velocity inlet as its leaving the AC grill and air velocity at window opening is conducted using digital flow meter.

2.1. PRE-PROCESSING

Geometric model provided 3-dimensionally of L609 Classroom as shown in Figure 2 and dimension as on Table 1 accordingly.

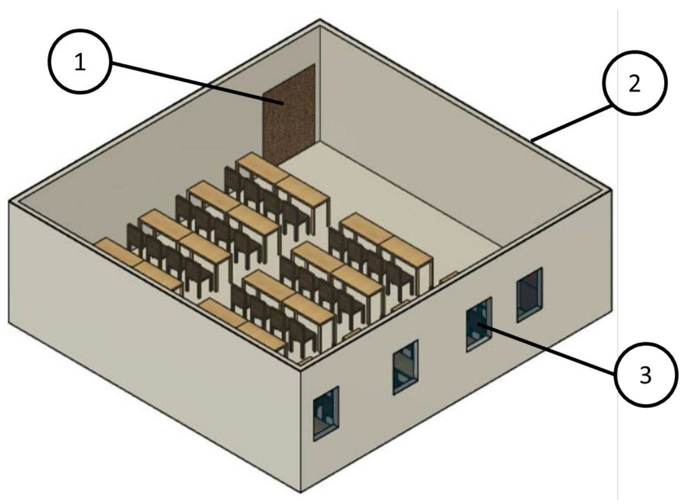


Figure 2. Geometry Model of L 609 Classroom

https://doi.org/10.24912/ijaste.v1.i2.473-483

Table 1. Property of L 609 Classroom

Number	Type	Dimension (cm)
1	Door	16 cm x 20.9 cm
2	Wall Thickness	15 cm
3	Window opening	7.6 cm x 10.8 cm

Next is to determine the position of the inlet and outlet in the classroom. Description of the simulation configuration can be seen in table 2.

Table 2. CFD simulation Configuration

Inlet	Outlet	Dimension	Temperature
3 pcs AC grilles and windows	door	AC grilles: 87 cm × 7 cm Door: 210 cm × 160 cm Door gap width: 1 cm	AC grilles (Inlet): 293 K Door Gap (Outlet): 300 K Wall: 305 K

For each simulation configuration that is made, the inlet speed in the AC will be taken directly in the L609 classroom using a Flowmeter. The following is a table of fan speed of Air Conditioner data on the L609 room AC:

Table 3. Description of AC blower speed

Blower speed	*Velocity* (m/s)
1	3,2
2	3,6
3	4
4	4,5
5	4,7

In order to obtain actual air flow data in the classroom to be used in each CFD simulation configuration, the inlet speed in the window will be taken directly in the L609 classroom using an anemometer. The following is a table of inlet speed data for each window.

Table 4. Description of wind velocity at window opening

Window	Air Velocity (m/s)	Temperature (°C)
Window 1	0.051	26.1
Window 2	0.028	27
Window 3	0.083	26.6
Window 4	0.026	26.7

After the classroom geometry has been created, the geometry from the Fusion 360 software is imported into the ANSYS 2022 R2 software for volume extraction and boundary conditions (determining Inlet, Outlet and Wall). After the geometry stage is complete, continue with the meshing process.

https://doi.org/10.24912/ijaste.v1.i2.473-483

2.1.1 CFD MODEL

In this meshing process, the geometry is divided into small parts. The size of the mesh contained in an object will affect the accuracy of the CFD analysis to be carried out. The smaller the mesh size on an object, the more accurate the results will be.

This study uses a tetrahedral to cover each corner on the room as well as corners on the windows area. Figure 3 shows the generated mesh including assigned boundary type for inlet and outlet.

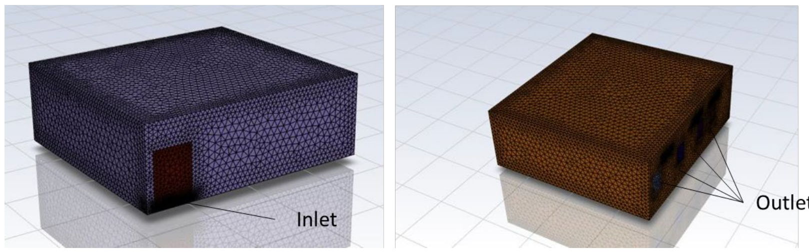
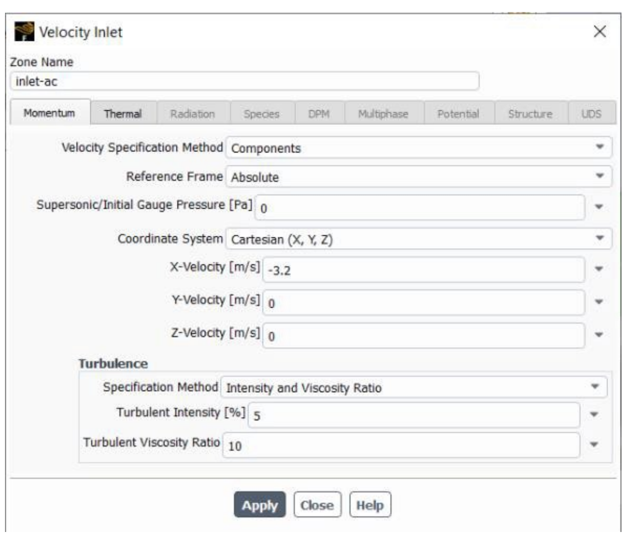


Figure 3. Gometry mesh

2.2. PROCESSING

At this stage, there are several things that must be done to complete this classroom simulation. This stage is the most important stage because it includes all existing parameters, from general, model, material, cell zone conditions, boundary conditions, solutions to run calculations.

In this classroom simulation, the default solution method is based on pressure. At this stage of the model, energy is made on because of the temperature used in the process. Then for viscous using standard SST unit *k-ω* with standard wall function. The rest of the parameters will be set by default. At this stage the material is divided into 2 parts, namely fluid and solid. For the fluid material used is air and for the solid material used is gypsum. The boundary conditions in this classroom simulation are divided into 4 parts, namely inlet, outlet, internal, and also wall. The inlet is divided into 5 parts, namely 4 windows and also the air conditioner uses the type of inlet velocity inlet which is influenced by temperature. For the outlet using the type of pressure outlet which is influenced by temperature. Assigned Boundary conditions provided in Figure 4 and Figure 5.



https://doi.org/10.24912/ijaste.v1.i2.473-483

Figure 4. Boundary Condition Inlet AC

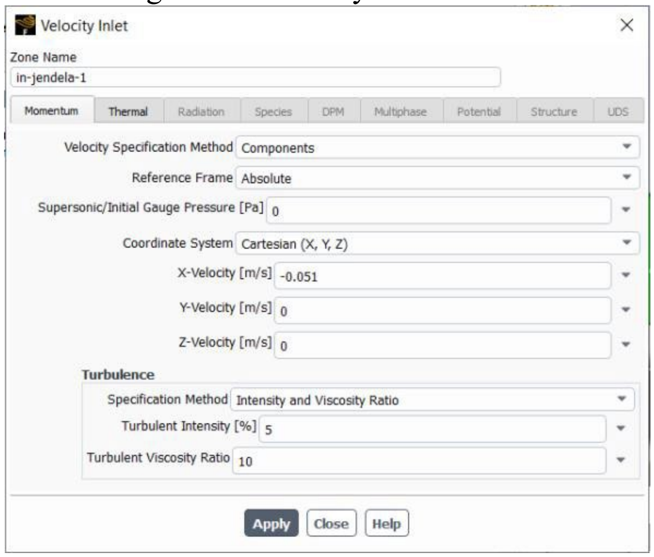


Figure 5. *Boundary Condition Inlet* Window

3. RESULT AND DISCUSSION

CFD simulation to L609 Classroom in Universitas Tarumanagara conducted. Post-processing in the form of CFD simulation results provided in the form of temperature profiles, pressure profiles, air velocity profiles along with streamlines that show air distribution. The simulation results that have been carried out are divided into two configurations, namely open-door simulation and closed-door simulation.

To clearly seen the simulation results, the plane feature is used to represent several speed points for each profile and configuration in the classroom. The first plane is located parallel to the outlet door which is located on the XY axis at a position of 0.11 meters from the negative Z axis. The second plane made is in the middle of the classroom which is also located on the XY axis at 0.48 meters from the negative Z axis. The third plane made is in the back row of student seats which is also located on the XY axis with a position of 0.86 meters from the negative Z axis. The fourth and fifth planes are different from the previous plane, if the previous plane is on the XY axis, the fourth and fifth planes are on the YZ axis. The fourth plane is close to the outlet with a position of 0.2 meters from the positive X-axis while the fifth plane is in the middle of the class with a position of 0.6 meters from the positive X-axis. That is the plane used to support the results in this L 609 classroom CFD simulation, the plane was made based on previous research. The CFD simulation results are divided into 3 parts in the form of temperature profile, pressure profile, profile air speed. These imaginary planes are provided in Figure 6.

https://doi.org/10.24912/ijaste.v1.i2.473-483

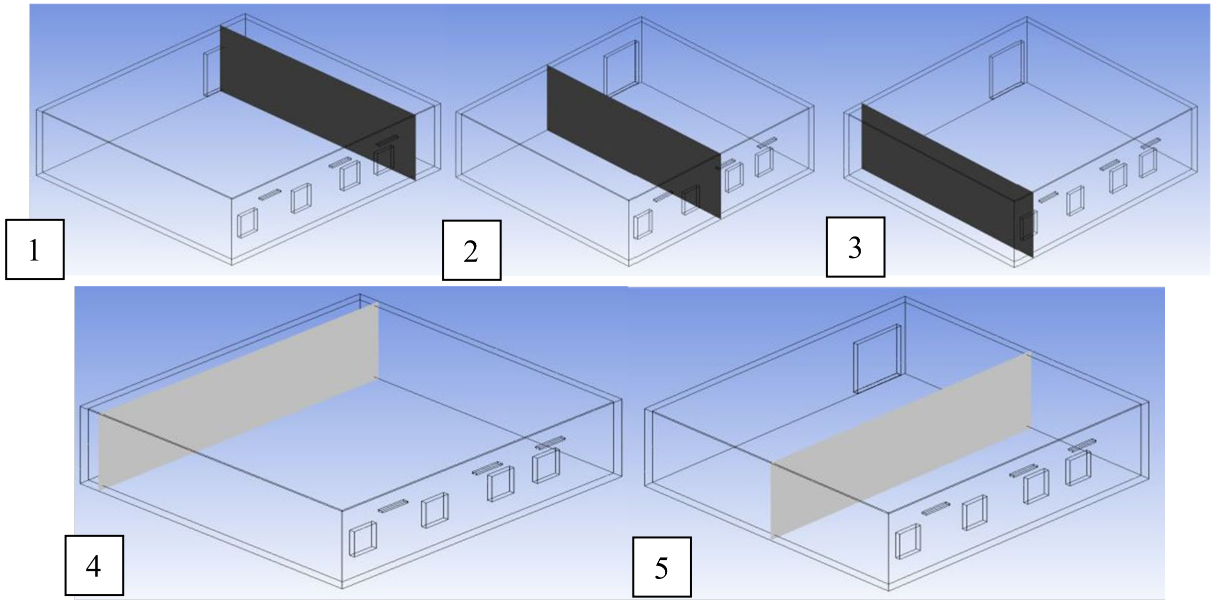


Figure 6. Plane Location for CFD simulation post-processing and analysis

3.1 Velocity Distribution

The simulation results of the velocity distribution are streamlined and also the velocity contour on each different plane which describes the velocity of the air flow that occurs in the entire contents of the room. The speed profile analyzed uses a plane that has been created because the plane created serves to show changes in the speed distribution that occur. To be able to determine the speed of air flow that occurs in the classroom, the probe feature is used in the simulation.

Table 4. Airflow Velocity Data

Velocity inlet (m/s)	coordinate X (m)	coordinate Y (m)	coordinate Z (m)	Velocity (m/s)
3.2	0.65	3.15	-9	0.793
3.6	0.65	3.15	-9	0.869
4	0.65	3.15	-9	0.945
4.5	0.65	3.15	-9	1.142
47	0.65	3.15	-9	1.232

https://doi.org/10.24912/ijaste.v1.i2.473-483

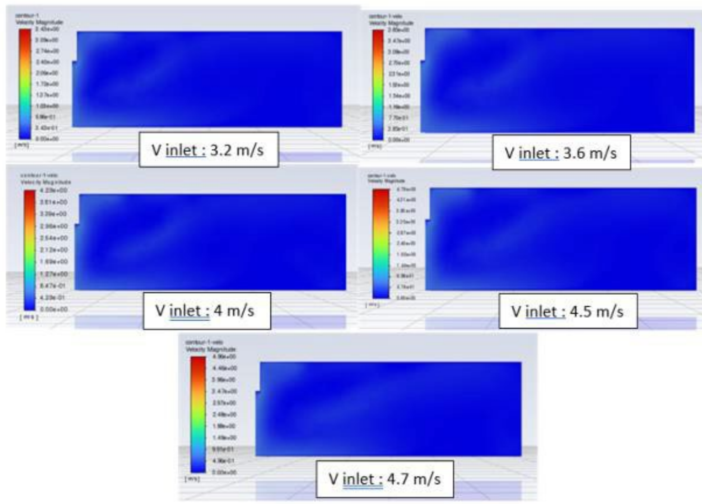


Figure 7. Velocity Distribution Profile Plane 1

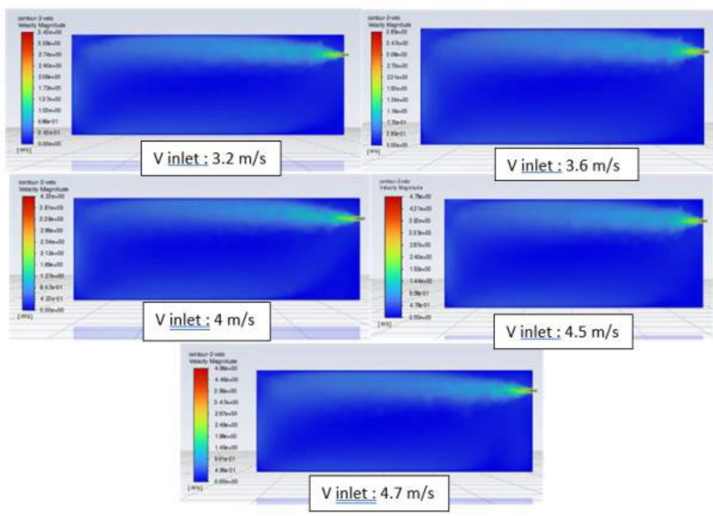


Figure 8. Velocity Distribution Profile Plane 2

There is an effect of air movement effect where when the air temperature coming out of the inlet hole is smaller than the temperature in the room, the increase in air velocity can affect cooling when there is a decrease in air temperature. The results obtained are different at each velocity. The difference in speed can be seen clearly on plane 1, plane 2 and also the streamline speed table above where the greater the velocity coming out of the AC inlet, the smaller the air affected by gravity, therefore the greater the velocity of the air coming out of the AC inlet, the more the air flow in the classroom can be well distributed.

3.2 Pressure Distribution

The results of the pressure distribution simulation are streamlined and the pressure contour on each different plane that describes the pressure changes that occur in the entire contents of the room. The pressure profile is analyzed using 2 planes, namely plane 2 and plane 4. This is because the two planes experience the largest changes in pressure distribution among the other planes.

https://doi.org/10.24912/ijaste.v1.i2.473-483

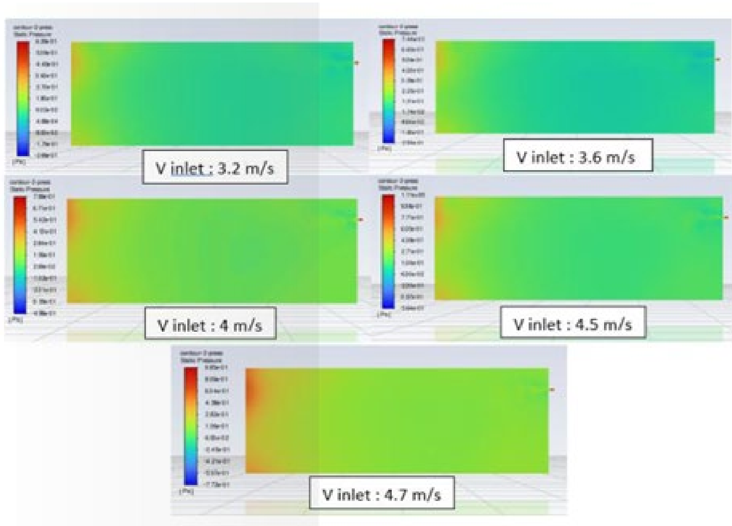


Figure 9. Pressure Distribution Profile Plane 2

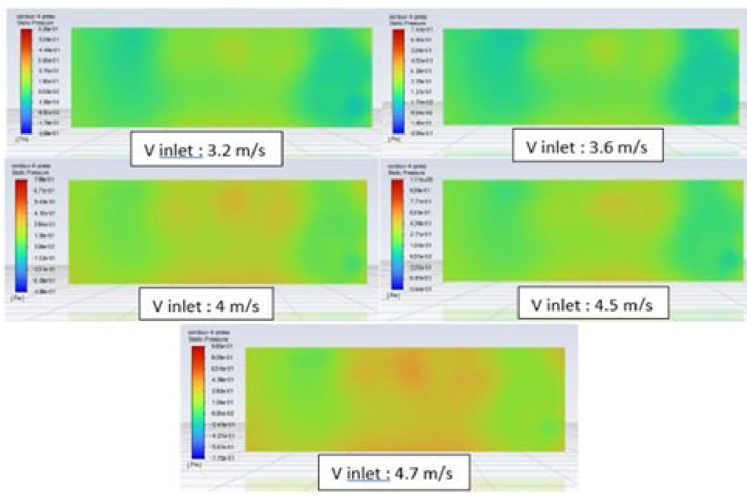


Figure 10. Pressure Distribution Profile Plane 4

It can be seen in plane 2 and plane 4 that the greater the air that is expelled from the inlet side, the smaller the pressure in the fluid, this is because the air moves towards the outlet so that the pressure in the outlet area decreases. The simulation results are in accordance with the Bernoulli equation where high pressure air flows into areas of low pressure.

3.3 Temperature distribution

Based on the simulation results regarding the temperature distribution profile, it can be seen in each plane indicating a mean radiant temperature effect where each object around has a radiant temperature. And on plane 2 it can be seen that the greater the speed released by the AC inlet, the faster the air spreads in the classroom with a comfortable temperature. Plane 3 indicates an similar temperature distribution ranging from 297.8 K to 301.5 K with the highest temperature being at the front and rear student desks because they are parallel to the AC.

https://doi.org/10.24912/ijaste.v1.i2.473-483

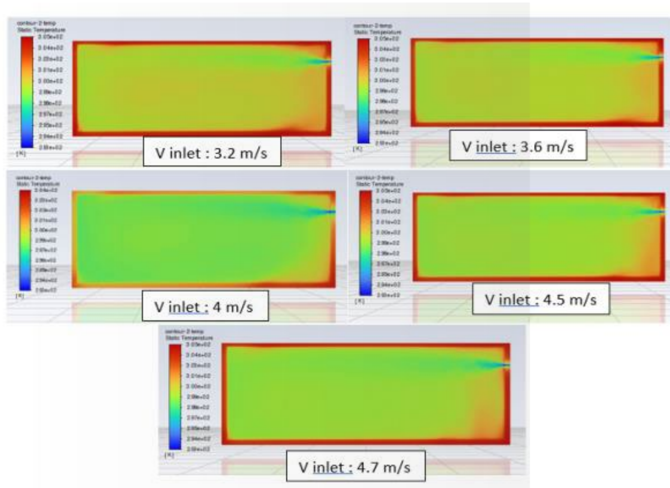


Figure 11. Temperature Distribution Profile Plane 2

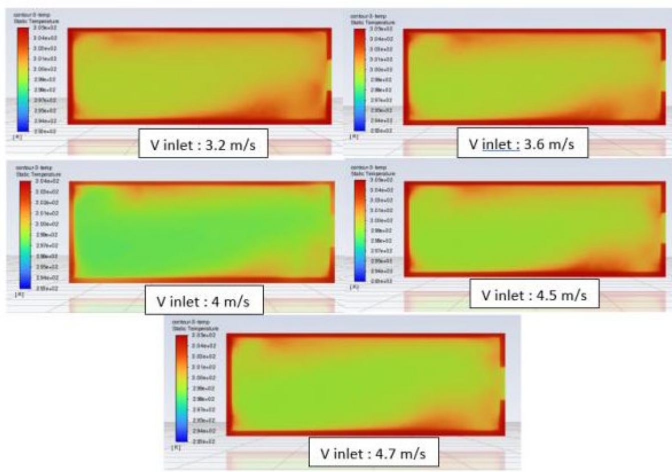


Figure 12. Temperature Distribution Profile Plane 3

According to ASHARE 55 2020, Human thermal comfort is defined as a state of mind that expresses satisfaction with the surrounding environment. Thermal comfort and good air quality in the classroom not only have a positive impact on student health in the classroom, but also help improve student concentration and learning performance. Therefore, it can be concluded that in the first configuration when viewed against the ASHRAE 55 standard in 2020, the results in the first configuration consisting of 3 distributions in the form of pressure, speed and temperature do not meet the ASHRAE 55 criteria because the air flow velocity in the classroom is more than 0.8 m/s which ranges from 0.86 m/s to 1.23 m/s which exceeds the air velocity standard value.

However, when viewed from the air circulation for the prevention of COVID-19, this first configuration is included in the criteria for good circulation in the spread of COVID-19 because it meets 3 requirements for good ventilation, such as:

1. Ventilation rate: the amount and quality of the outside air flow that enters the room
2. Airflow direction: this airflow aims to channel air from a clean place.
3. Airflow system: air distribution to all parts of the room in order to increase the dissolution and reduction of pollutants from the room.

With the configuration of the outlet in the form of a door, the spread of COVID in the classroom can be minimized due to good air exchange.

https://doi.org/10.24912/ijaste.v1.i2.473-483

4. CONCLUSION

According to the distribution profile results in the simulation, it can be concluded that the configuration with the outlet door is not in accordance with the ASHRAE 55 standard in 2020. This is because the air flow velocity in the room ranges from 0.869 m/s to 1,232 m/s which is not in accordance with the ASHRAE 55 standard. that is not to exceed 0.8 m/s.

Based on all the simulation results carried out by 5 different speed fan configurations, the best configuration according to the ASHRAE 55 standard in 2020 is the configuration with the 1st fan speed (3.2 m/s). In this configuration, it can be seen that the room temperature is between 298.7 K to 304.9 K and the indoor air flow velocity is 0.793 m/s. These results are also based on the consideration of the COVID-19 pandemic, which requires air circulation in order to break down the aerosol concentration of the COVID-19 virus itself.

REFERENCES

[1] C. Buratti, D. Palladino, and E. Moretti, "Prediction of Indoor Conditions and Thermal Comfort Using CFD Simulations: A Case Study Based on Experimental Data," *Energy Procedia*, vol. 126, pp. 115–122, 2017, doi: 10.1016/j.egypro.2017.08.130.

[2] "Pentingnya Ventilasi yang Baik Pada saat Pandemi Covid-19." https://upk.kemkes.go.id/new/pentingnya-ventilasi-yang-baik-pada-saat-pandemi-covid-19 (accessed Feb. 23, 2022).

[3] K. Juniar, "Judul : kevin juniar," 2022.

[4] R. Qin and C. Duan, "The principle and applications of Bernoulli equation," *J. Phys. Conf. Ser.*, vol. 916, no. 1, Oct. 2017, doi: 10.1088/1742-6596/916/1/012038.

[5] S. Darmawan, "Reynolds number effects on swirling flows intensity and reattachment point over a backward-facing step geometry using STD k-ε turbulence model," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 852, no. 1, 2020, doi: 10.1088/1757-899X/852/1/012073.

[6] K.-J. Bathe, *Computational fluid and solid mechanics 2003[Recurso electrónico] :] proceedings, Second MIT Conference on Computational Fluid and Solid Mechanics, June 17-20, 2003*. 2003. [Online]. Available: http://0-www.sciencedirect.com.jabega.uma.es/science/book/9780080440460

[7] A. Fluent, "Ansys Fluent Theory Guide," *ANSYS Inc., USA*, vol. 15317, no. November, pp. 724–746, 2013.

https://doi.org/10.24912/ijaste.v1.i2.473-483