Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 812-820
© Universiti Tun Hussein Onn Malaysia Publisher's Office

RPMME

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Numerical Study of Airflow Effect on Temperature

Mohammad Fazlan Harun[1], **Norasikin Mat Isa**[1]

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, MALAYSIA.

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.087
Received 20 July 2021; Accepted 25 Nov. 2021; Available online 25 December 2021

Abstract: Thermal comfort can be defined as the state in which an individual felt a sense of well-being in relation to the thermal conditions in an occupied space. The variety of personal preferences of individuals in feeling the thermal condition makes a definitive definition of thermal comfort difficult to describe. Computational fluid dynamics has now made it possible to simulate the airflow behaviour using simulation applications. This study is capable to provide an understanding of the airflow behaviour in air conditioning room. Upon completion of this study, the airflow of the air conditioning system can be simulated. It is expected to determine the possibility whether the velocity of airflow has a significance change on the air distribution thus affecting the thermal conditions. CFD ANSYS Fluent simulation method was used to simulate and analyze the velocity and temperature distribution in the test rig which is a room. There are three value of air velocity that have been manipulated in this simulation which are 0.2 m/s, 0.5 m/s and 0.9 m/s. The result obtained show that all velocity streamline shows a similar pattern except that with higher velocity of air supply at inlet, the more streamline created thus can cover the test rig more comprehensively. In term of air velocity distribution, the air velocity decreases as it goes from the upper horizontal plane to the lower horizontal plane, as seen by the velocity contour formed. This is due to the opening of the inlet, which is located at the top-centre of the domain. As a result, the areas with the highest air velocity are unquestionably those closest to the inlet. Even while the temperature contour is noticeably different in colours, indicating that there is still a variation along the temperature value on the plane, the difference is still very little and insignificance differ only around 0.1 Celsius. Optimum placement of inlet and outlet causes temperature distribution to be uniform throughout the room, resulting in cold air spreading evenly.

Keywords: Airflow, Temperature, CFD

1. Introduction

Air conditioner is a device that regulates temperature by removing hot air from a certain closed space to provide a cool air sensation to bring thermal comfort to the occupant. The majority of individuals spend more than 90% of their time each day inside a building [1]. Many performance and

**Corresponding author: sikin@uthm.edu.my*

health issues connected to indoor environmental conditions might be avoided with better air conditioning design system [2]. To ensure the optimum efficiency and performance of the AC system, the development of future indoor environmental control in residential buildings should incorporate centralised AC systems [3]. Air flow can be defined as the act of migration of air particle operates in a fluid manner which particle will flow from higher pressure area to lower pressure area. Thermal comfort is occasionally interpreted as the condition of an individual perceives a feeling of prosperity concerning the thermal conditions in an occupied space [4]. Definitive definition of thermal comfort is hard to be defined as it is closely related to the complexity personal preference of individual in experiencing the thermal condition. ASHRAE Standard 55 (Thermal Environmental Conditions for Human Occupancy) is an industry standard that serve as a guideline in creating an environment that achieve a thermal comfort. The level of thermal comfort also can be estimated by using Fanger's comfort equation [5]. Nowadays, engineering application had made it possible to simulate the airflow distribution through the usage of computational fluid dynamics. Computational fluid dynamics is a branch of fluid mechanics that uses numerical analysis and algorithms to analyse fluid flows.

2. Methodology

Software Ansys (Fluent) will be used to analyse the particle flow and to simulate the airflow distribution in the room. The initial phase in pre-processing involves three fundamental basic procedures in CFD simulations. The second phase is to solve, and the final step is post-processing.

2.1 Computational Domain

The room parameter as in Figure 1, where the research will be conducted is modelled based on the room Hikmah 3, floor 2 of Tunku Tun Aminah library, UTHM. The geometry of the room had been draw using the Solidwork software before been imported into the Ansys software. The room consist of 1 door, 1 window, 2 ceiling lamp, 1 inlet diffuser located at the top center of the room ceiling and 1 outlet at the back-right of the room ceiling. This room has dimensions of 3.07 m x 2.78 m x 332 m.

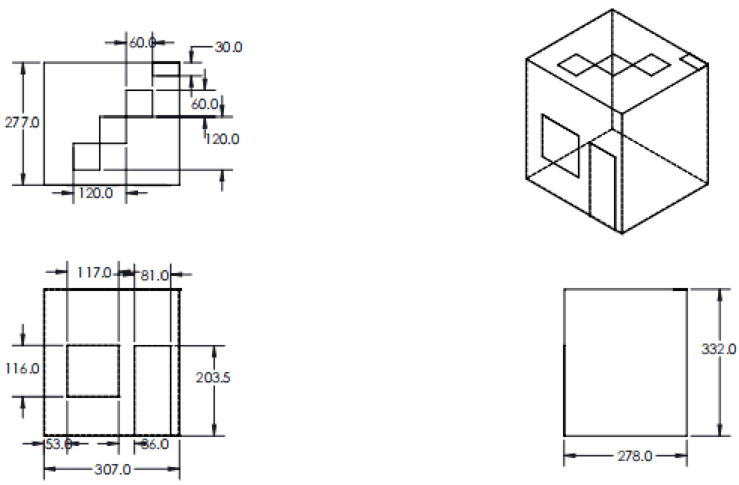

Figure 1: Room parameter

2.2 Meshing

The grid has cells of various shapes as shown in Figure 2, and the default setup for 3D geometry was employed for this investigation. The inlet and outlet location were specified and named. Later on, the mesh was generated. The smooth level adjusted from medium to high. Then, the mesh needed to be updated again as to include the new adjustment. The skewness is at 0.9 which is an acceptable mesh metric spectrum.

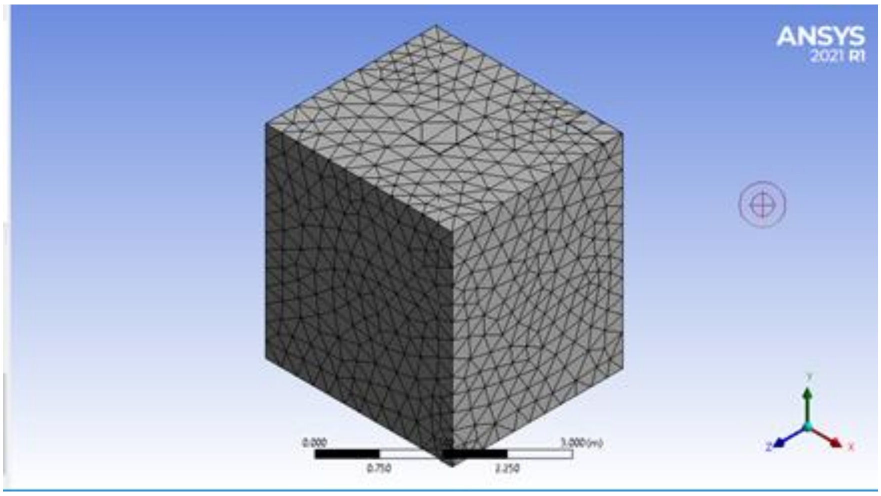

Figure 2: The meshing geometry

2.3 Boundary and Parameter Setup

Table 1: The boundary and parameter setup

Name selection	Parameter
	Reference frame = Absolute
Inlet	Temperature=297.15 kelvin
	Velocity = 0.2 m/s, 0.5 m/s, 0.9 m/s
	Pressure = 0 Pa
	Turbulent intensity = 5%
	Turbulent viscosity ratio = 10
	Backflow reference frame = Absolute
Outlet	Temperature = 300 kelvin
	Pressure = 0 Pa
	Pressure Profile Multiplier
	Backflow turbulent intensity = 5%
	Backflow turbulent viscosity ratio = 10
	Default as in Ansys database.
Wall	Wall motion = Stationary wall
	Shear condition = No slip
	Roughness models = Standard
	Roughness Constant = 0.5

. Table 1 shows all the boundary and parameter used in this study. The inlet air velocity has been supplied at 0.2, 0.5 and 0.9 m/s at constant temperature of 297.15 K equivalent to 24 degrees Celsius.

3. Results and Discussion

The outcome was extracted from steady state analysis. In this analysis, the velocity of airflow supply at constant temperature 297.15 Kelvin (24 Celsius) is changed from 0.2 m/s to 0.5 m/s and 0.9 m/s to observe the influence of change in velocity towards air distribution and temperature distribution. Three plane was constructed in both direction of horizontal (Figure 3) and vertical (Figure 4) to extract the necessary data.

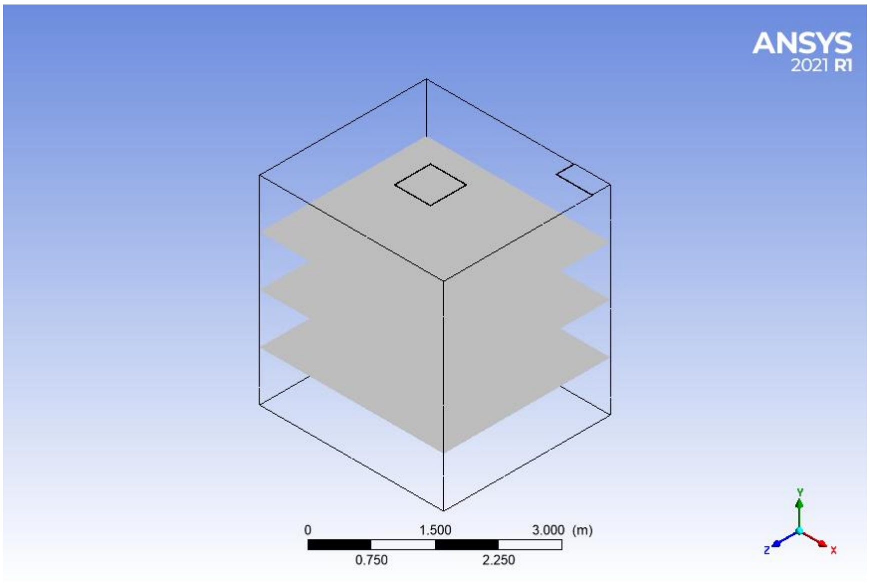

Figure 3: Horizontal planes

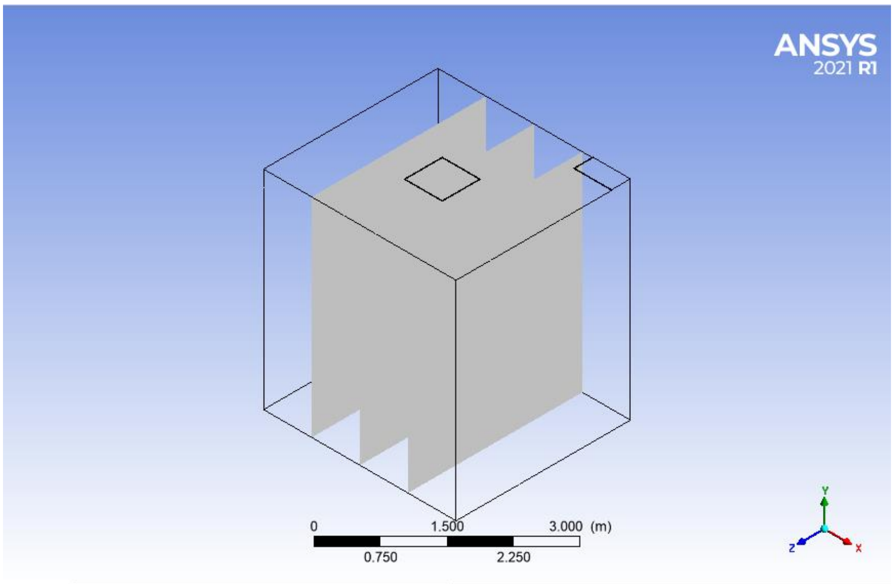

Figure 4: Vertical planes

3.1 Velocity streamline

All figure of 5, 6 and 7 show the velocity streamline when velocity inlet is supply at 0.2 m/s, 0.5 m/s and 0.9 m/s respectively, which cover almost all area in the test rig. It can be seen in all cases, the particle flow downwards from the opening at inlet to the region just above the floor and creating a circular motion upwards before eventually flow towards the outlet at the top right corner of the test rig. Generally, all velocity streamline shows a similar pattern except that with higher velocity of air supply at inlet, the more streamline created thus can cover the test rig more comprehensively.

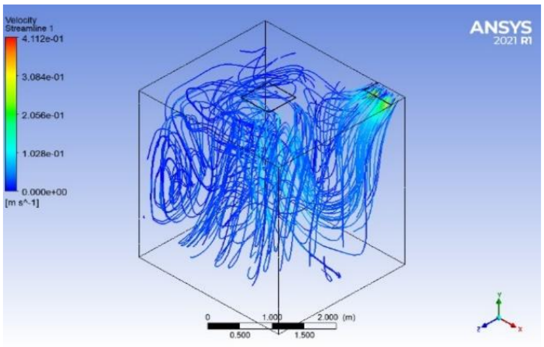

Figure 5: Velocity Streamline at velocity inlet 0.2 m/s

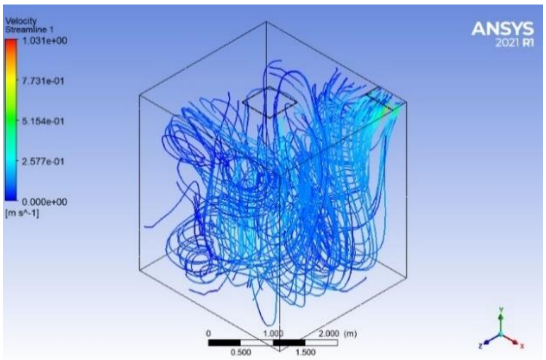

Figure 6: Velocity Streamline at velocity inlet 0.5 m/s

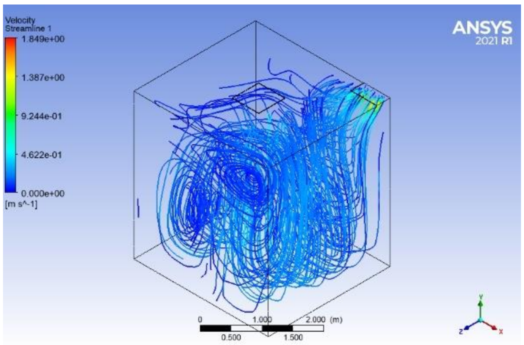

Figure 7: Velocity Streamline at velocity inlet 0.9 m/s

3.2 Velocity distribution

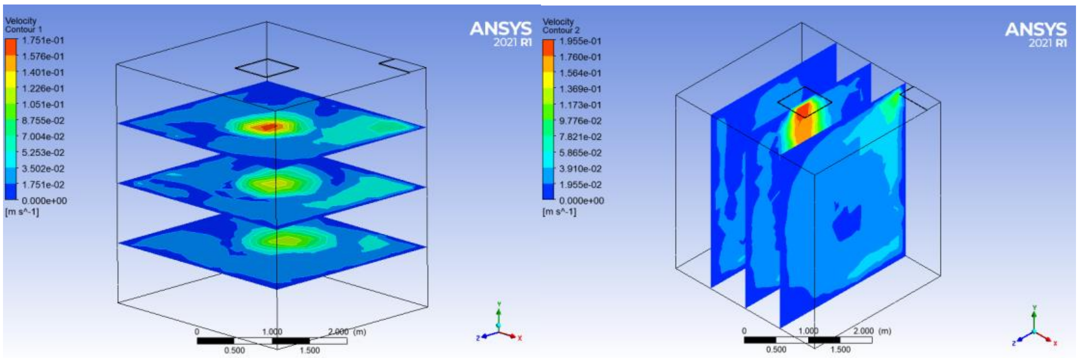

Figure 8: Velocity Contour at velocity inlet 0.2 m/s

Figure 8 shows the air velocity contour for velocity inlet 0.2 m/s on horizontal and vertical plane. On the upper horizontal plane, it can be seen that the area at the center has a higher air velocity range between 0.157 m/s to 0.1751 m/s and the value of the velocity contour decrease as it moves from center to the wall due to the air burst that coming from the inlet opening. The area under the outlet also shown a slightly higher value of velocity that range between 0.035 m/s to 0.07 m/s compare to other region that most has temperature below 0.0175 m/s. The vertical plane shows a more even velocity distribution on both the left and right side of the vertical plane in which most velocity below 0.05 m/s.

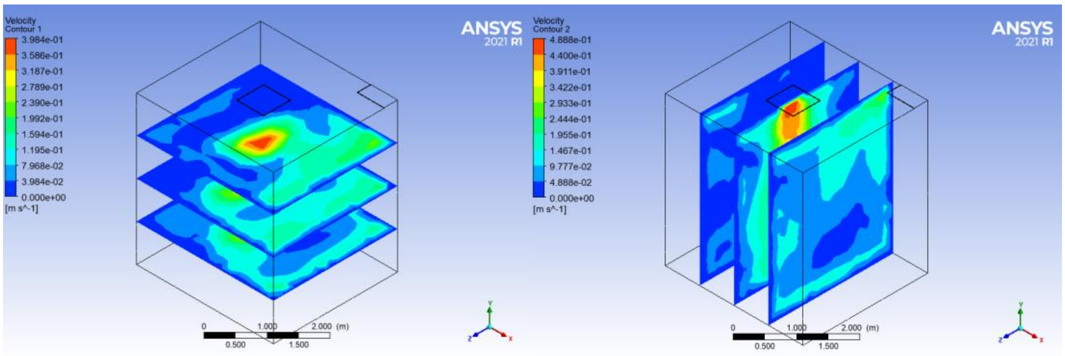

Figure 9: Velocity Contour at velocity inlet 0.5 m/s

Figure 9 shows the velocity contour at velocity inlet 0.5 m/s on both horizontal and vertical plane. Higher air velocity is found in locations near the opening inlet. The middle vertical plane shows the highest air velocity is at 0.48 m/s, while the upper horizontal plane records the maximum air velocity at 0.39 m/s. The middle horizontal plane shows a circulating air velocity along this plane is divided by two regions of velocity in which the right side has a higher value of velocity around 0.07m/s – 0.159 m/s compares to region on the left side which most have velocity below 0.03 m/s.

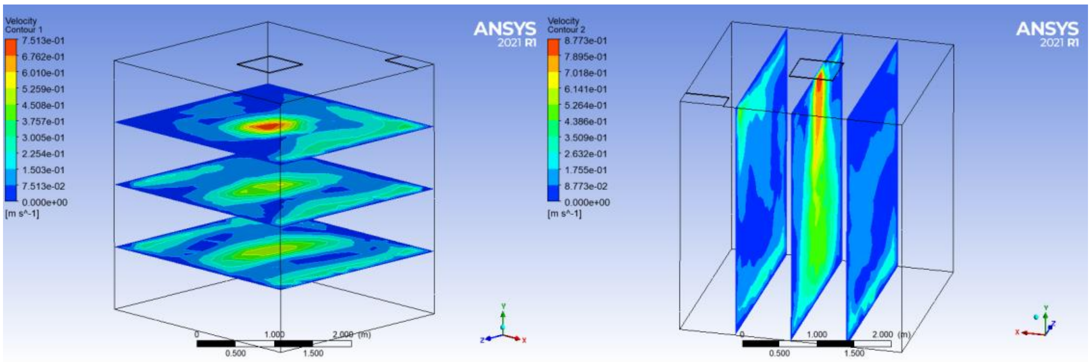

Figure 10: Velocity Contour at velocity inlet 0.9 m/s

Based on the Figures 10 above, the velocity distribution is more even on vertical plane on left and right compared to the vertical plane at the middle. The vertical plane on right and left shows the most part is covered with velocity value range between 0.26 and below. The velocity contour on the middle vertical plane also shows a smooth distribution of velocity with major region has a range between 0.15 m/s to 0.525 m/s. On the horizontal plane, it can be observed that the middle and lower horizontal plane show a similar velocity contour with higher velocity at center and both right and left side of the wall.

3.3 Temperature distribution

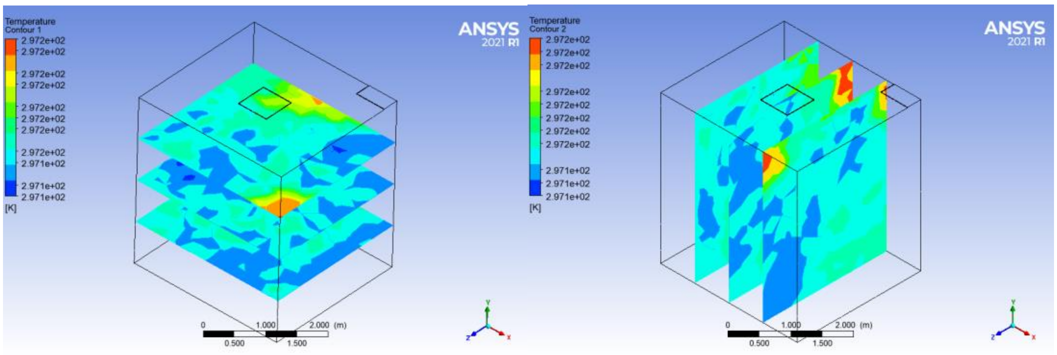

Figure 11: Temperature distribution at velocity inlet 0.2 m/s

Figure 11 shows temperature distribution at velocity inlet 0.2 m/s on horizontal and vertical plane. On the upper horizontal plane there are two parts that has a higher temperature in relative to other region that is at the middle back and at the front right corner. Vertical plane of right side also indicates that the lower part at the front region has the coolest temperature relatively. Though it can be seen the temperature contour has variety of colour indicating vary in temperature value, the value is too small that the temperature distribution is considered even at which temperature ranged between 297.1 Kelvin to 297.2 Kelvin.

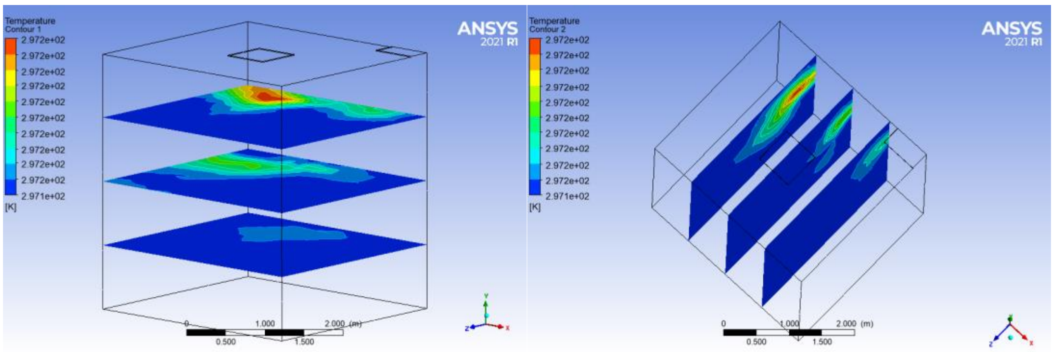

Figure 12: Temperature distribution at velocity inlet 0.5 m/s

Figure 12 shows temperature distribution at velocity inlet 0.5 m/s on horizontal and vertical plane. As can be seen, the temperature distribution become more even as it moves from upper horizontal plane to lower horizontal plane based on the temperature contour. The left vertical plane shows a higher temperature relatively and become colder as it moves to the right plane to the opening of the outlet. At the lower horizontal plane, the temperature contour is highly uniform indicating a good temperature distribution. At this velocity inlet of 0.5 m/s, it can be seen that the upper part near right wall is not efficiently cooled. However, as this level is not in human sitting and standing region, it is not significance to be corrected. Overall, temperature distribution at velocity inlet 0.5 m/s is even at temperature ranged between 297.1 K to 297.2 K.

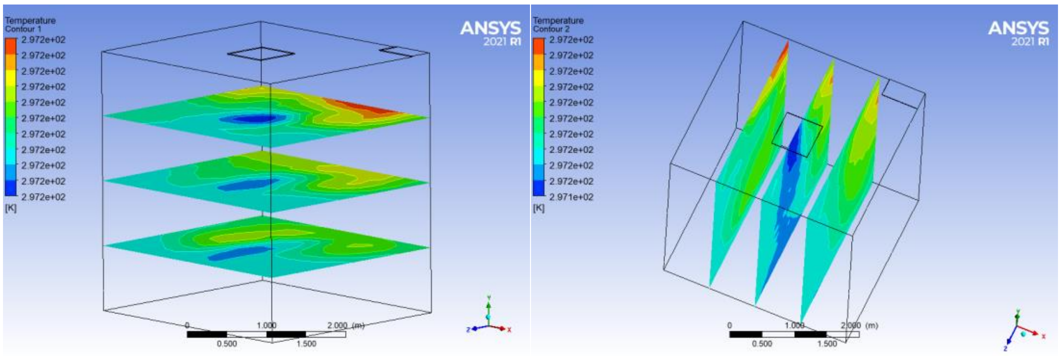

Figure 13: Temperature distribution at velocity inlet 0.9 m/s

Figure 13 shows temperature distribution at velocity inlet 0.9 m/s on horizontal and vertical plane. On the middle vertical plane, coolest temperature contour region formed due to the direction of cold air bursts from the inlet, resulted in relatively a lowest temperature along the center of the room. The region at the back edge shows an uncooled area relatively compared to the front region as the cold air escape through the outlet attached at the corner back. The horizontal plane on the middle and on the lower plane show a similar pattern where temperature at the front is cooler than temperature at the back relatively. The vertical plane on the left and right side also indicates an identical temperature contour pattern as blasts of cold air from the inlet have not reached this part. Generally, eventhough the temperature contour is visibly different in colours indicate there is still a difference along the temperature value on the plane, the amount is still very small that is it varies only between 23.95 celsius to 24.05 Celsius.

4. Conclusion

Based on the results of the numerical simulations, at various velocity all velocity streamline shows a similar pattern except that with higher velocity of air supply at inlet, the more streamline created thus can cover the test rig more comprehensively. In term of air velocity distribution, the air velocity decreases as it goes from the upper horizontal plane to the lower horizontal plane, as seen by the velocity contour formed. This is due to the opening of the inlet, which is positioned at the domain's top centre. As a result, the areas with the highest air velocity are unquestionably those closest to the inlet. Even while the temperature contour is noticeably different in colours, indicating that there is still a variation along the temperature value on the plane, the difference is still little and insignificance, ranging only between 23.95 and 24.05 degrees Celsius. Correct placement of interior air conditioning units causes temperature distribution to be uniform throughout the room, resulting in cold air spreading evenly.

Acknowledgement

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

References

[1]. Vakiloroaya, V., Samali, B., Fakhar, A., & Pishghadam, K. "A review of different strategies for HVAC energy saving," Energy Conversion and Management, 77, 738–754, (2014).

[2]. Mendell, M. J., Fisk, W. J., Kreiss, K., Levin, H., Alexander, D., Cain, W. S., Girman, J. R., Hines, C. J., Jensen, P. A., Milton, D. K., Rexroat, L. P., & Wallingford, K. M., "Improving the health of workers in indoor environments: Priority research needs for a National Occupational Research Agenda", American Journal of Public Health, 92(9), (2002).

[3]. Aste, N., Adhikari, R. S., & Manfren, M., "Cost optimal analysis of heat pump technology adoption in residential reference buildings," Renewable Energy, (2013).

[4]. F. Bode, I. Nastase, P. Danca, A. Meslem and P. Danca, "The influence of the Inlet angle of vehicle air diffuser on the thermal comfort of passengers," *2017 International Conference on ENERGY and ENVIRONMENT (CIEM)*, Bucharest, 2017, pp. 442-446.

[5]. M. Bingxin, S. Jiong and W. Yanchao, "Experimental design and the GA-BP prediction of human thermal comfort index," *2011 Seventh International Conference on Natural Computation*, Shanghai, 2011, pp. 771-775.