Journal of Advanced Research Design

Journal homepage:
https://akademiabaru.com/submit/index.php/ard
ISSN: 2289-7984

Prediction of Air Flow in a Cinema under Different Diffuser Location

Mohd Hafifi Roslan[1], Ummikalsom Abidin[1,*], Nor Azwadi Che Sidik[2]

[1] Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
[2] Malaysian-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100 Kuala Lumpur, Wilayah Persekutuan Kuala Lumpur, Malaysia

ARTICLE INFO

Article history:
Received 21 September 2021
Received in revised form 24 July 2021
Accepted 3 January 2022
Available online 7 January 2022

Keywords:
Overhead air diffuser; underfloor air diffuser; computational fluid dynamics

ABSTRACT

The utilization of different types of air distribution systems in the installation of building air conditioners has a positive impact on effective cooling. This effort is also capable of enhancing the internal environmental quality and the welfare of the occupants. A numerical method employing a 3D model using Computational Fluid Dynamics (CFD) investigates the airflow temperature and velocity distribution in an air-conditioned cinema or theater. Numerical simulations are executed using the commercial CFD software ANSYS Fluent. This study aims to predict and identify the flow characteristics, temperature and velocity distribution of the current overhead air diffuser system (OHAD) and the new arrangement of the underfloor air diffuser system (UFAD) in an air conditioning system. In the system evaluation, both the OHAD and UFAD systems were modeled and compared. The results indicated that the use of the CFD numerical technique can accurately predict the airflow temperature and velocity distribution in the conditioned space of a cinema. Furthermore, the UFAD system was capable of creating smaller vertical variations in air temperature and a more comfortable environment than the OHAD system under the supply temperature and velocity of 18 °C and 0.8 m/s. The highest temperature achievable using the UFAD system was 22 °C, falling within Malaysia's ASHRAE standard recommendation of 18 - 27°C.

1. Introduction

The importance of reducing building energy consumption has increased since the greenhouse effect has become a serious problem. For space heating and cooling; heating, ventilation and air-conditioning (HVAC)'s air distribution strategy has a strong influence not only on internal thermal comforts, but also for energy costs. The air distribution system also has direct influence on space organization, floor height, inland layout and construction cost. All of the previous research achievements also show that the increasing developments of computational fluid dynamics (CFD) in the recent years have opened the possibilities of low-cost yet effective methods for improving HVAC systems in design phase, with less experiment required. The advantage and disadvantage on conventional designs of HVAC systems, with or without using CFD techniques [1,19].

* *Corresponding author.*
E-mail address: ummi@utm.my

Task/ambient conditioning (TAC) systems allow occupants to control temperature, air flow, and in some cases lighting and sound to meet their individual needs. This technology has recently been gaining a foothold in the United States. It is often implemented in conjunction with under floor air distribution, which opens up opportunities for a number of efficiencies in building design and operation [2]. Under floor air distribution (UFAD) systems have been applied as an alternative to conventional air distribution (CAD) systems in recent years. This system was first introduced in the 1950's to cool computer rooms. A major paradigm of the UFAD system is that its air distribution characteristics are different from overhead CAD systems. It has been suggested that new load calculation and design methods need to be developed to properly design and implement this air distribution technology [8]. It is found that under floor system gives better performance than overhead system in contaminant removal and significantly in energy saving while maintaining the same thermal comfort condition [3,20].

The used numerical technique could accurately predict the air flow and temperature distribution in the high-rise conditioned space. UFAD system is capable of creating smaller vertical variations of air temperature and a more comfortable environment and energy saving than Overhead Air Diffuser system (OHAD) system, the supply air velocity and temperature, number of diffusers and height of the space have a significant impact on thermal comfort. The percentage of energy saving due to using UFAD system increases with increasing the theater height [4,22]. In UFAD systems, the exhaust/return-split configuration is believed to be more promising than the exhaust/return- combined arrangement in terms of energy efficiency. However, how to properly position the return vents still remains an unanswered question when the thermal comfort and indoor air quality (IAQ) need to be simultaneously considered [5,23]. In a study by Raluca Teodosiua *et. al.,* [6] results showed a decent agreement between measurements and numerical values in terms of velocity profile at the inlet. In addition, the overall flow development in the air jet is well represented numerically.

Air infiltration through building entrances is one of the main sources of energy loss in modern buildings. Previous studies have shown that air curtains, when used at building entrances, can reduce infiltration related energy loss significantly [7,21]. Floor swirl diffuser is one of the commonly used diffusers in UFAD system. Geometrical design of the floor swirl diffuser will affect the airflow pattern of the indoor environment [9,24]. Rahul Khatria *et al.,* [10] concluded that minimum time taken and maximum temperature drop was achieved with the installation of both the equipment on the same side of the room.

According to Hurnik *et al.,* [11] identification of the evolution of the air mean velocity distribution with distance from the supply opening were studied. In the distance of 15 to 30 equivalent diameters, the jet behaves as a free turbulent axisymmetric jet; its momentum flux is conserved and fixed at approximately 80 % of the inlet momentum flux. At higher distances from the inlet, the confinement effect of partitions is observed. The CFD results validation and reporting methods applicable for the benchmark data are proposed.

Air flow mal-distribution has been identified as one of the causes for performance degradation of air to refrigerant heat exchangers. The high computational cost of CFD makes it difficult to comprehensively study the effect of multiple parameters impacting the air flow distribution and the heat exchanger performance. An integrated CFD-segmented heat exchanger model based on a momentum resistance model is developed that is computationally efficient and accurate. The CFD simulation process is fully automated using script based open-source CFD code, Open FOAM [12].

Gan *et al.,* [13], used CFD to predict the indoor environment of a mechanically ventilated room and overall ventilation effectiveness of air distribution systems. It was found that the most effective air distribution system for heating operation differs from that for cooling. It was shown that an air distribution system that results in upward displacement ventilation performs better than others in terms of indoor air quality and energy use but may cause local thermal discomfort.

The above literature revealed that numerical studies succeed in predicting indoor airflow pattern and temperature distribution with good accuracy and less difficulty, and consumed time. Moreover, the numerical solutions can give more details in full study field and complete parametric study. Detailed comprehensive study on the effects of the different operating and design parameters on the air flow pattern, temperature distribution and thermal comfort can be studied through CFD. The importance of reducing the use of building energy is increasing.

For space heating and cooling, the HVAC air distribution strategy has a strong influence not only on the thermal comfort of room air but also on energy costs. The air distribution system also directly affects the composition of the room, floor height, room layout, and construction cost. Therefore, based on this statement the objectives of the current study are to develop numerical model for analyzing the flow in a cinema as well as to predict the flow characteristics and temperature distribution under different current OHAD and new method UFAD.

1.1 Mathematical Formulation and Numerical Method

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics. The air how considered in this study has very low velocity and the change in pressure is very small as well. Therefore, the flow is assumed to be incompressible. The three-dimensional steady flows with heat transfer continuity, momentum and energy equations can be written in Cartesian tensor notation as follows,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

The fact that the rate of change of momentum is equal to the sum of the forces gives the momentum equations,

$$\begin{aligned}
\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{u}) &= \frac{\partial(-\rho + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \\
\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v \mathbf{u}) &= \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial(-\rho + \tau_{xx})}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} + S_{My} \\
\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w \mathbf{u}) &= \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial(-\rho + \tau_{zz})}{\partial z} + S_{Mz}
\end{aligned} \tag{2}$$

The fact that the rate of change of energy is equal to the sum of the rate of heat addition to and the work is done on the fluid gives the energy equation,

$$\begin{aligned}
\frac{\partial(\rho E)}{\partial t} + \nabla \cdot (\rho E \mathbf{u}) = -\nabla \cdot (\rho \mathbf{u}) + \Bigg[&\frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{zx})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \\
&\frac{\partial(v\tau_{yz})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z}\Bigg] + \nabla \cdot (\lambda \nabla T) + S_E
\end{aligned} \tag{3}$$

The terms on the left-hand side represent the rate of change of energy and the net rate of flow of energy out of the volume, respectively. The terms on the first two rows on the right-hand side represent the total rate of work done by surface stresses. The last two terms represent the rate of heat addition due to heat conduction across the boundaries of the volume and the rate of change of energy due to sources [16].

1.2 Turbulence Modelling

The Reynolds number (Re) is a measure of the relative importance of inertial forces and viscous forces [14]. It is expressed as in below,

$$\mathrm{Re} = \frac{\rho U L}{\mu}$$

The governing equations of the steady mean flow are called the Reynolds-Averaged Navier-Stokes (RANS) equations,

$$u(t) = \overline{u} + u'(t) \quad v(t) = \overline{v} + v'(t) \quad w(t) = \overline{w} + w'(t) \quad p(t) = \overline{p} + p'(t) \tag{5}$$

Reynolds stresses are proportional to the mean velocity gradients according to Eq. (6),

$$-\rho\overline{u_i' u_j'} = \mu_t \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{6}$$

The RANS equations include additional stress terms called Reynolds stresses. A turbulence model is needed to account for the Reynolds stresses and close the system of equations [15].

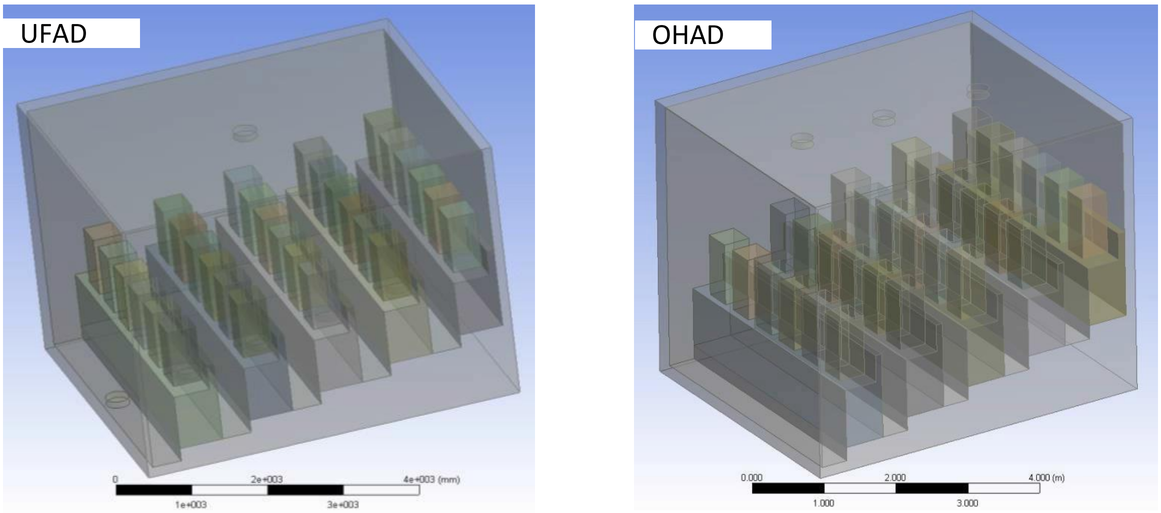
CFD using ANSYS FLUENT was used in the present study to simulate three-dimensional turbulence flow of indoor air distribution in theater by using the realizable *k-ε* turbulence model at steady-state condition. The governing equations are discretized by using the finite volume method. The near-wall region was solved by using enhanced wall treatment. The pressure-velocity coupling was achieved by SIMPLE algorithm. First-order upwind spatial discretization scheme was considered for turbulent dissipation rate and turbulent kinetic energy, second order pressure, second-order upwind momentum and energy and energy equations because it provides fast convergence and better results in the flow field and temperature distribution. The solution is considered converged when the normalized residual of continuity, momentum, turbulence and energy is less than 10^{-3}, 10^{-4}, 10^{-5} and 10^{-6}, respectively for determining accurate results. The *k-ε* turbulence model with, near wall treatment use standard wall function. This selected because of this model is unsteady and turbulent flow. Energy sources using people indoor usually sit or do some lighter physical labor. The metabolic rate of sitting is 57 W/m^3 [17].

Whereas the full 3D CFD analysis of the under-floor airflow yields useful results, a 1D idealization of the flow can be used to get valuable insight have derived such a 1D model and provided an analytical solution of the governing equations. The validity of the model is demonstrated with reference to a full three-dimensional CFD solution [16].

2. Physical Model, Computational Domain and Boundary Conditions

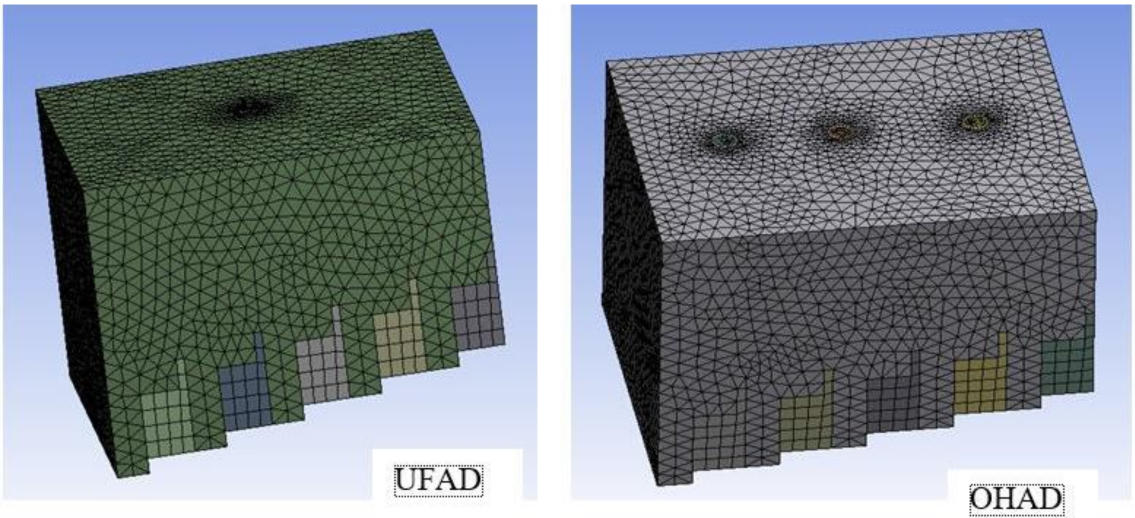
The studied physical model has been selected to have typical operating conditions and geometrical configuration to those found in the theaters. The selected section (i.e. repeated portion) from the total theater was chosen as a computational domain to minimize the simulation time. The section dimension is 6 m length x 4 m width x 4m height as shown in Figure 1. According to the international building codes, the theater sections accommodate five rows of five seats. The conditioned air is delivered to the theater from supply diffusers and returned through the ceiling return diffuser. Thirty simulators to simulate 30 seated human bodies were placed on the seats. Human bodies were simplified as rectangular cuboids of dimensions 0.4 m x 0.4 m x 0.9 m. The supply air diffusers are defined as inlet uniform velocity and the return diffusers are set as pressure outlet boundary condition. All the vertical walls are defined as symmetry and adiabatic wall conditions are applied for the floor, ceiling and seats.

In the base line case study (i) the finished ceiling height is varying from 4 m at the front and 3m at the rear, (ii) five supply diffusers are used and distributed on the floor such that one diffuser is located in the middle of each row at front of seat, (iii) one return diffuser is placed in the center of the ceiling of the studied section, (iv) circular supply diffusers of cross sectional area of 0.01 m^2 and circular return diffuser of 0.4 m diameter are considered and (v) air is supplied to the theater at 18 °C with a velocity of 0.8m/s. The grid generation of computational domain is described in Figure 1, where unstructured tetrahedral cells are chosen.



(a) (b)

Fig. 1. Design model (a) UFAD and (b) OHAD



(a) (b)

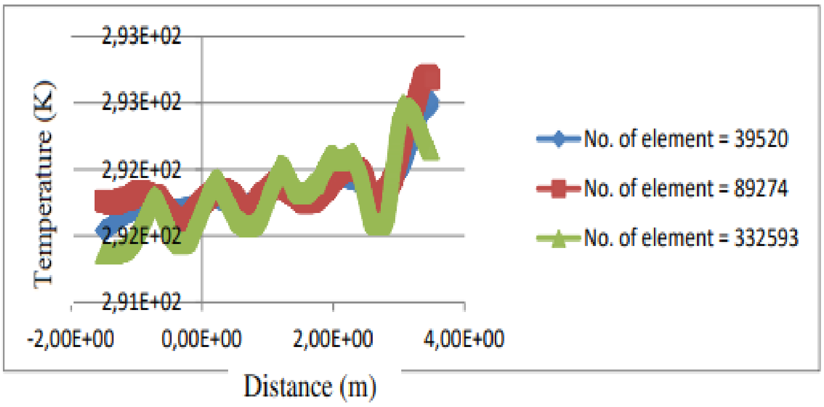
Fig. 2. Meshing of (a) UFAD and (b) OFAD

2.1 Grid Independence Test

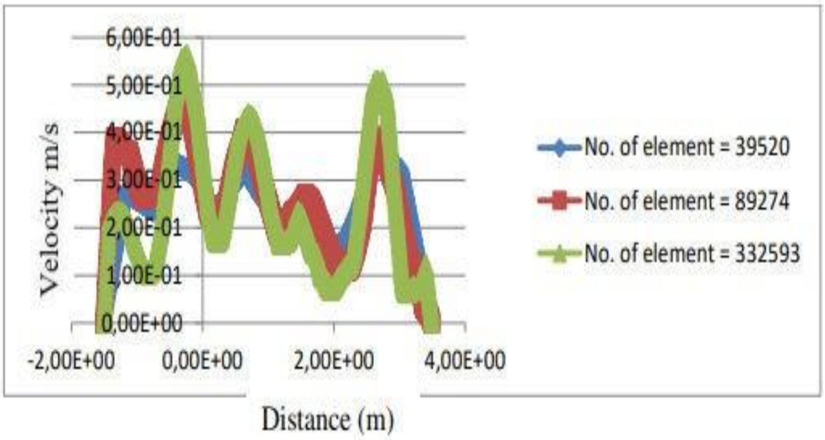
Models UFAD are built for the scale model with the same dimensions mentioned in Figure 1 (a). Three different number of cells (39520, 89274 and 332593). For models OHAD are built for the scale model with the same dimensions mentioned in Figure 1 (b). Three different number of cells (35473, 80212 and 244981). Both number of cells are used to check the model solution convergence. To carry out the present study for accurate simulations and less time consumptions. The solution was considered grid independent and the grid size of 89274 and 80212.

2.2 Generate Meshing

Modelling of design was completed using Solid Work 2016 version, the next procedure modelling design will be export to ANSYS Fluent for generating the mesh. During the process to generate a mesh for the model have any setup need to change. For this model, sizing needs to insert on surface modelling. Sizing for this model using 0.1 m for the element size and the unstructured tetrahedral cells are chosen. Figure 2 (a) and (b) above show the setup mesh for this model.

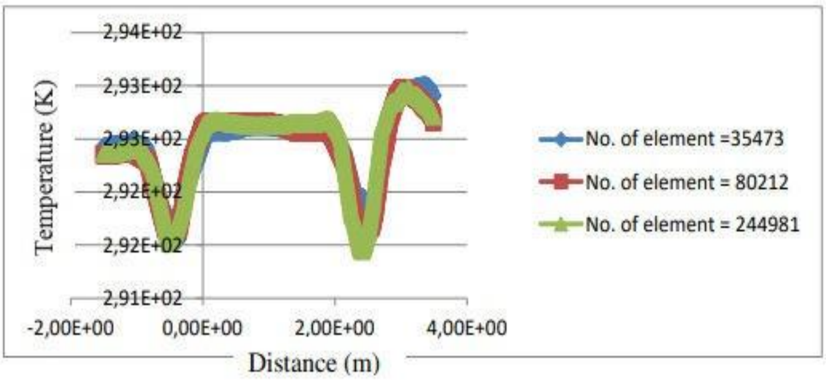


(a)

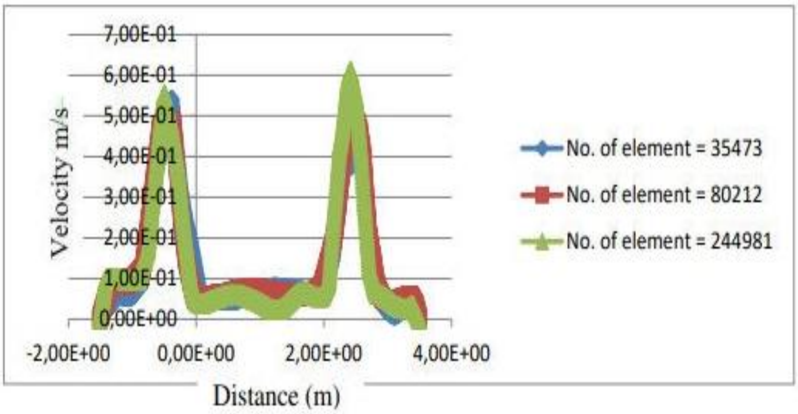


(b)

Fig. 3. Grid independence test (a) UFAD temperature distribution and (b) UFAD velocity distribution



(a)

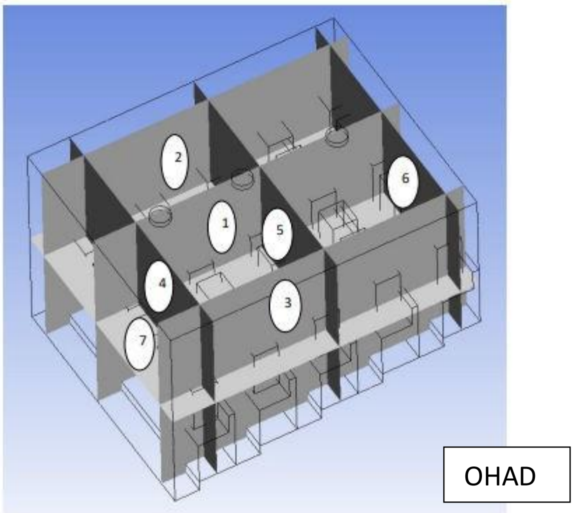


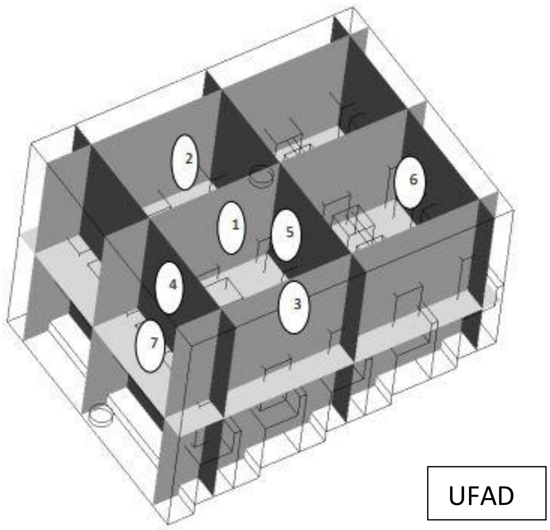
(b)

Fig. 4. Grid independence test (a) OHAD temperature distribution and (b) OHAD velocity distribution

3. Methodology

The aims for this study is to determine, the temperature and airflow distribution at several planes for both the OHAD and UFAD system design as shown in Figure 5. Three vertical side view x-axis (Plane 1, Plane 2 and Plane 3), three vertical front view z-axis (Plane 4, Plane 5 and Plane 6) and a horizontal top view x-axis are (Plane 7) are selected for this study.





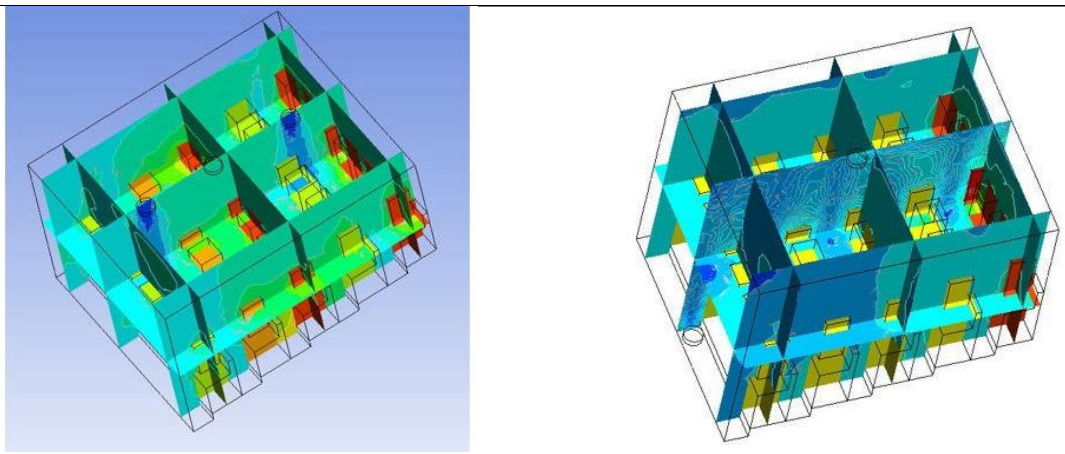
(b) (b)

Fig. 5. Plane distribution air flow (a) OHAD and (b) UFAD systems

4. Results and Discussion

4.1 Numerical Analysis

Figure 6 shows the contour result simulation on OHAD and UFAD system. The purpose of this study is to observe the high temperature location. Temperature needs to be maintained at 18°C (291K) and 0.8 m/s followed by the previous study recommendation [7]. Contour profile of UFAD system shows a lower temperature distribution in comparison with OHAD system. Detail simulation results will be explained in the next section of this paper.



(a) (b)

Fig. 6. Contour profile result (a) OHAD and (b) UFAD systems

4.2 Temperature and Velocity Distribution Contour

4.2.1 Temperature distribution contour

In UFAD system, air is diffused at ground level and gained heat due to thermal loads inside the room rising its temperature and moves it upwards under the action of the supplying velocity and forced convection affects. In the case where air is diffused at low temperature, it will cause a sense of coldness which is an inconvenient issue. If the supply air is diffused at high temperature, it reaches people's occupied levels at high uncomfortable temperature which is also inconvenient according to ASHRAE Standard [18]. Therefore, the supply temperature should be precisely chosen to satisfy the required thermal conditions.

Temperature distributions over the different selected planes and lines are represented in Figure 8 - 14. For OHAD and UFAD model. In general, all figures show the temperature increases slowly with increasing height measured from the supply level. The gradient of the temperature increases with height at the upper level where the heat source (occupants) exist is much higher than the gradient in the lower part of the theater. The well distribution of temperature for UFAD system shows a better cooling or lower surface temperature of occupant in comparison with OHAD system as shown from different planes as in Figure 8 - 10. This effect is clearly seen for occupants from row 1 - 4 with surface temperature slightly below 22 °C. Furthermore, occupants in row 5 is still in comfort cooling effect with the surface temperature of 22 °C. Occupants in OHAD system experiences less conducive environment due to the effect of unbalance temperature distribution. For OHAD system, occupants below the diffuser which is at row 1 and 4 will feel prolonged cooling. On the contrary, the middle rows occupants will feel slightly warmer where the maximum occupant surface temperature is 23 °C. The comfort level of front and back rows occupants of OHAD and UFAD system are shown in Figure 11 and 12 respectively. Occupants in front and back row of UHAD system experience the same comfort as the temperature is not exceeding 22 °C. On the other hand, the less comfort effect is experiences by the OHAD system of front rows' occupants.

Figure 14 shows the comparison contour of temperature distribution between OHAD and UFAD system of plane 7 or the top plane. All occupants in all rows of UFAD system is in comfort temperature due to well temperature distribution in the space. In UFAD system, middle, second and third row occupants will experience cooler temperature. On the other hand, mix occupants comfort zone for OHAD system is observed. Occupants in third and last row experiencing slightly higher comfort temperature in comparison to occupants in other rows. In conclusion, all the planes in UFAD system shows that the temperature in the occupied level is 22 °C where it is lies within the ASHRAE standard recommendation. Therefore, UHAD system is a better option in comfortable temperature range in the cinema or theatre as stated by ASHRAE Standard [18].

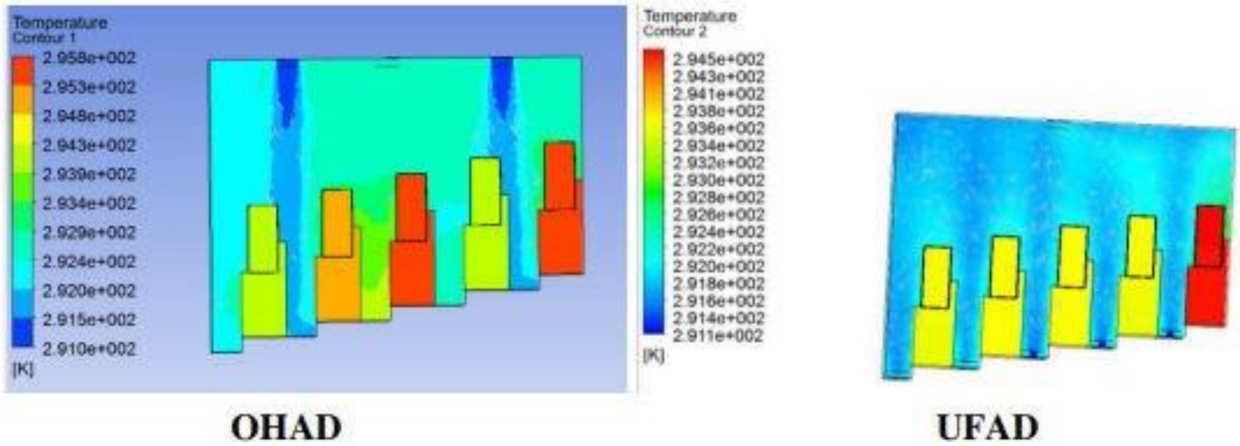
4.2.2 Velocity distribution contour

In ASHRAE comfort standard, the air velocity is an important factor in the calculation of thermal comfort indices. Moreover, the standard also restricts the extent to which air velocity can be used to achieve comfort by limiting it to a specified maximum level of 0.8 m/s in summer according to previous researches. Researchers concluded that high supply air velocity is needed for more mixing in the occupied zone for UFAD system [25]. Figure 8 - 14 show that the velocity distribution contours for OHAD and UFAD system. The figures show that both OHAD and UFAD system will reach the optimum supply velocity of 0.8 m/s where the temperature in the occupied level (about 22 °C) lies within Malaysia ASHRAE standard recommendation which is 18 – 27 °C.

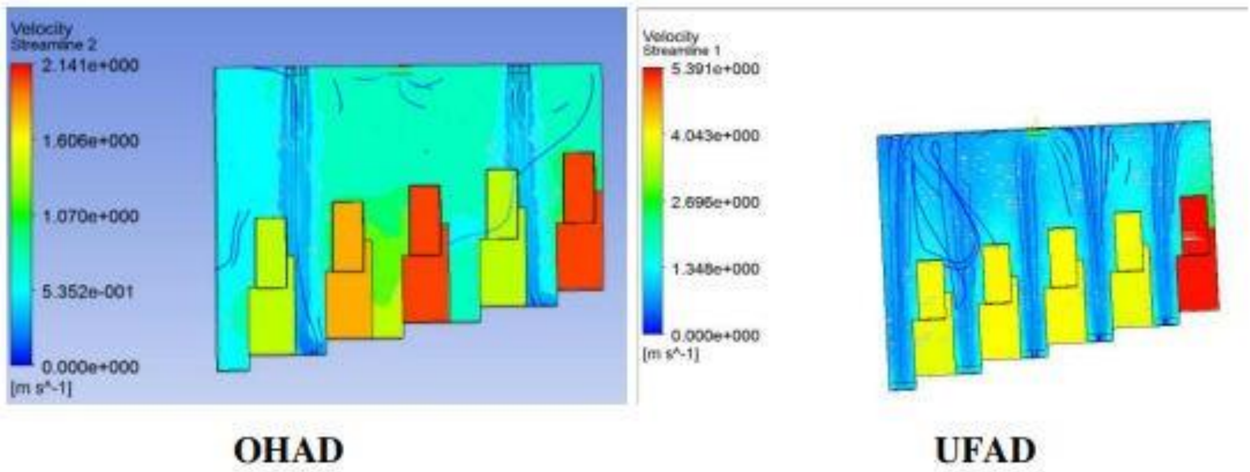
Figure 8 - 13 illustrates a well distributed velocity distribution contour for UFAD system in comparison to OHAD system. However, Figure 14 shows that velocity of UFAD system is better distributed at the fourth and last row in the cinema.

It is also noticed from the figures that if the supply velocity decreased or increased from 0.8 m/s, the temperature in the occupied zone increases (reaches above 27 °C) exceeding the comfort level.

i. Side View



(a)



(b)

Fig. 8. Comparison contour (a) Temperature distribution OHAD and UFAD system Plane 1 and (b) Velocity magnitude OHAD and UFAD system Plane 1

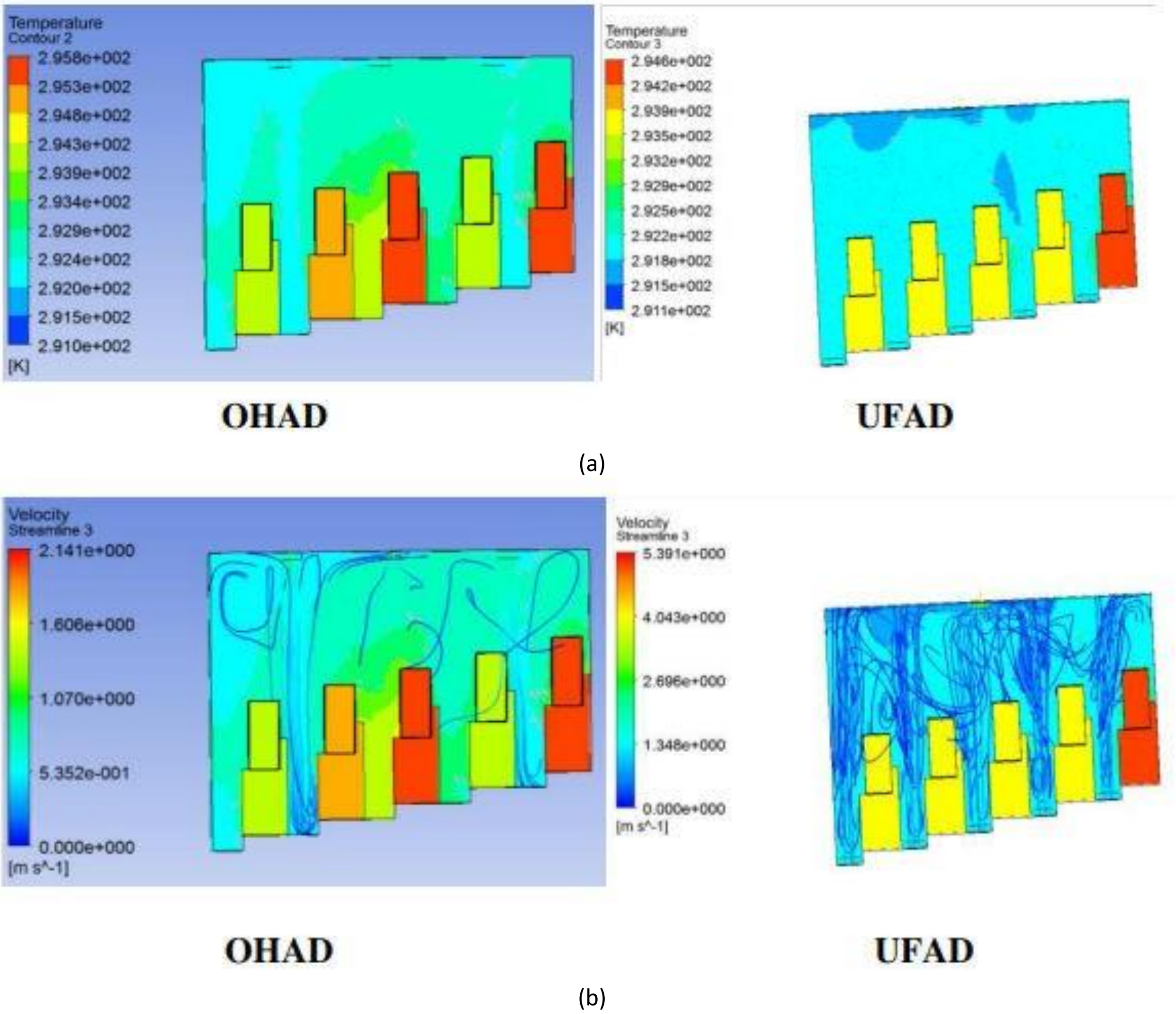
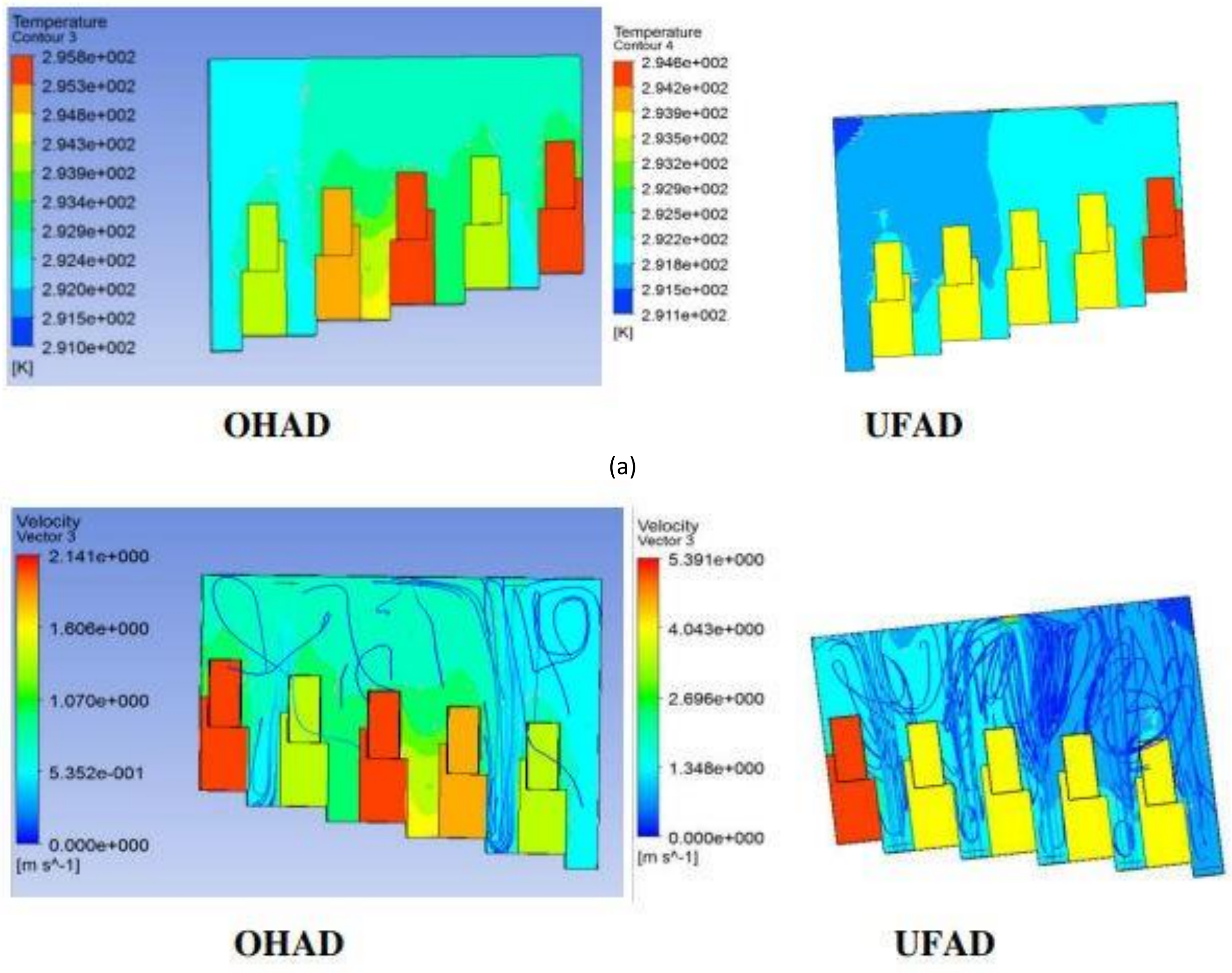


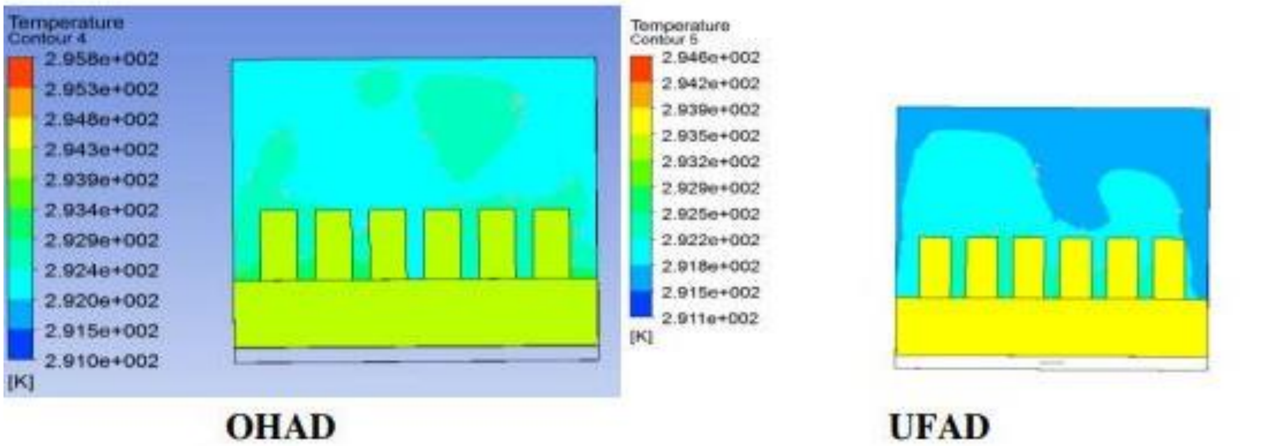
Fig. 9. Comparison contour (a) Temperature distribution OHAD and UFAD system at Plane 2 and (b) Velocity magnitude OHAD and UFAD system at Plane 2



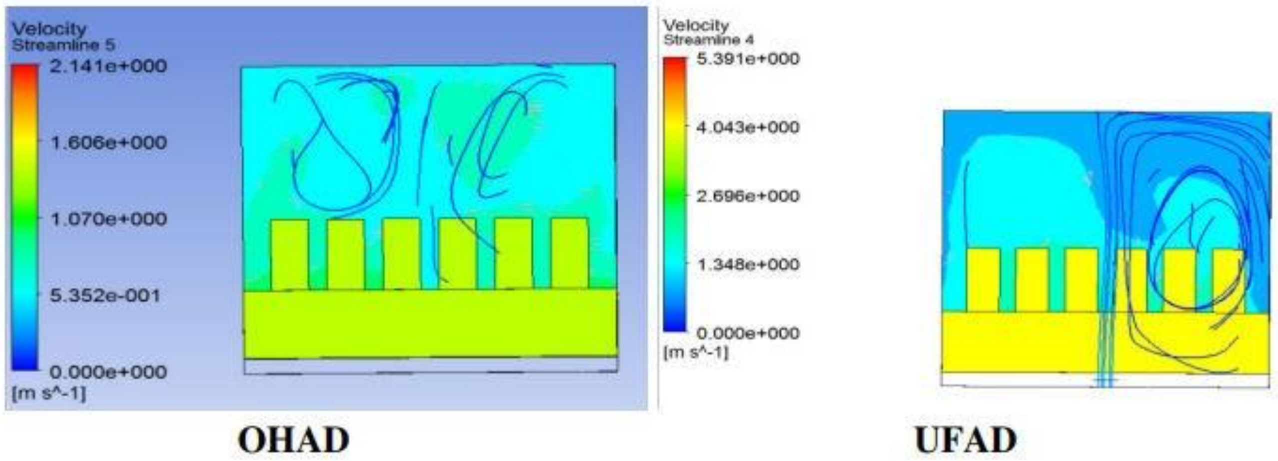
(b)

Fig. 10. Comparison contour (a) Temperature distribution OHAD and UFAD system at Plane 3 and (b) Velocity magnitude OHAD and UFAD system at Plane 3

ii. Front View



(a)



(b)

Fig. 11. Comparison contour (a) Temperature distribution OHAD and UFAD system Plane 4 and (b) Velocity magnitude OHAD and UFAD system Plane 4

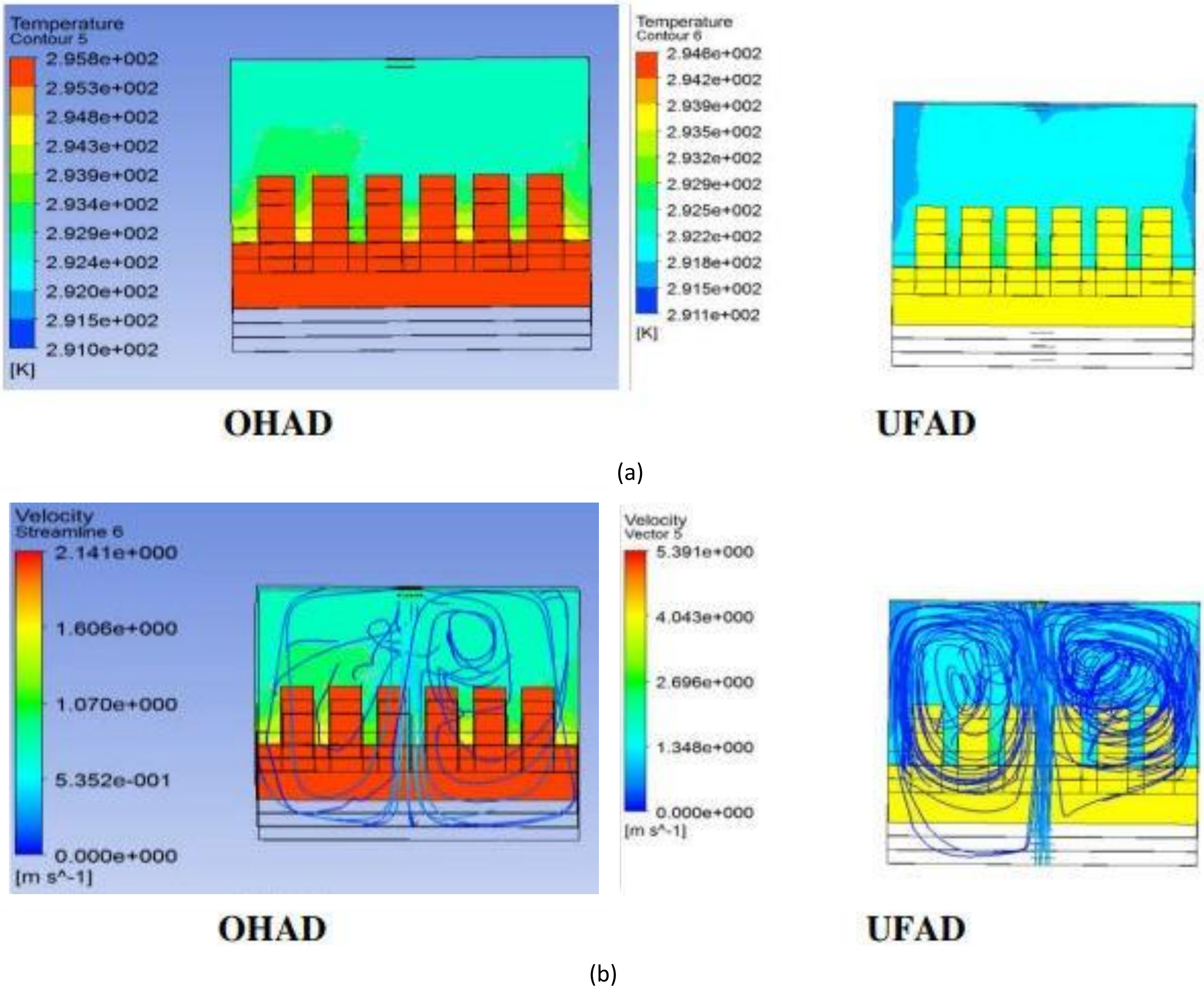
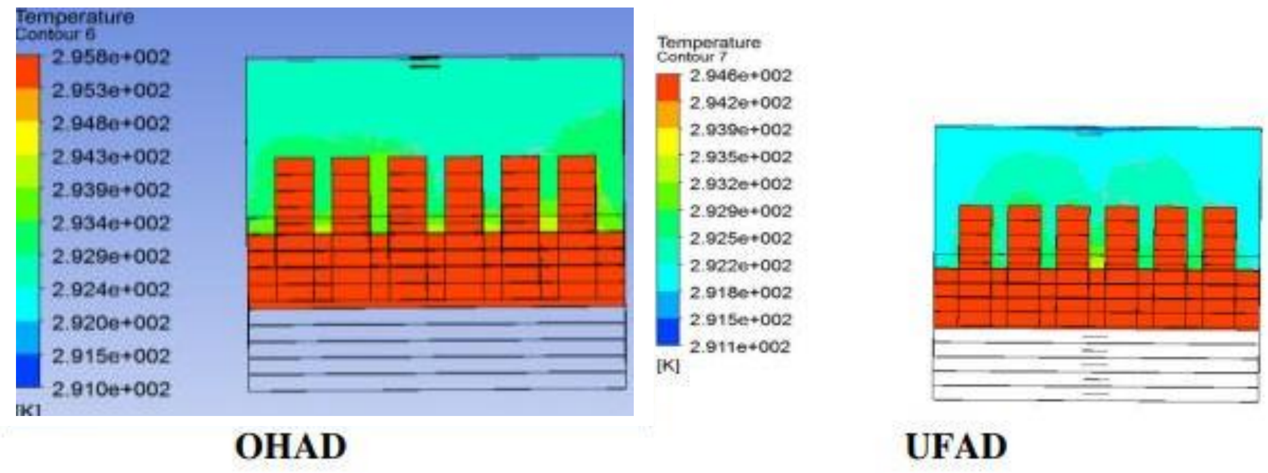
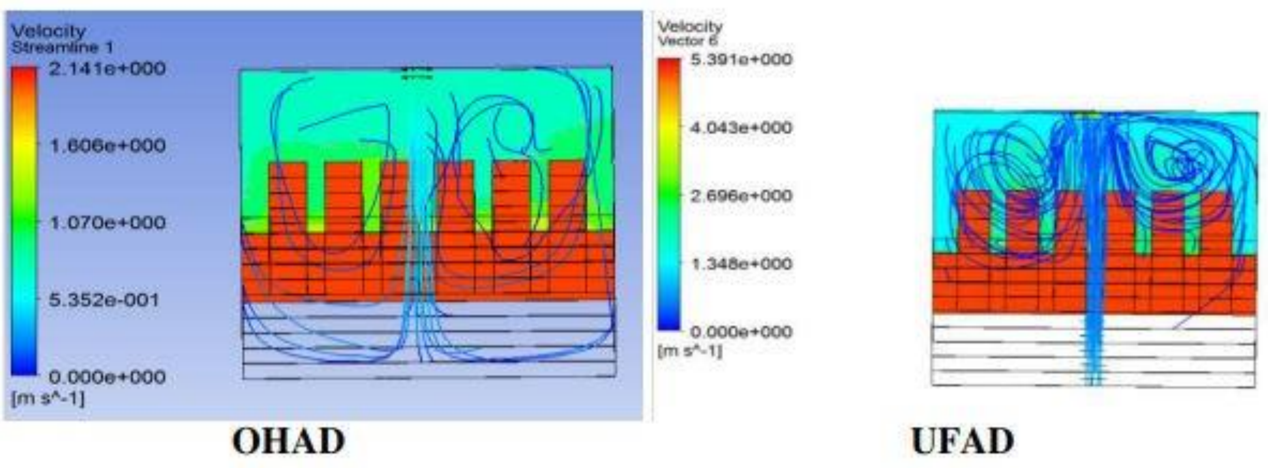


Fig. 12. (a) Comparison contour temperature distribution OHAD and UFAD system at Plane 5 and (b) Comparison contour velocity magnitude OHAD and UFAD system at Plane 5

iii. Back View



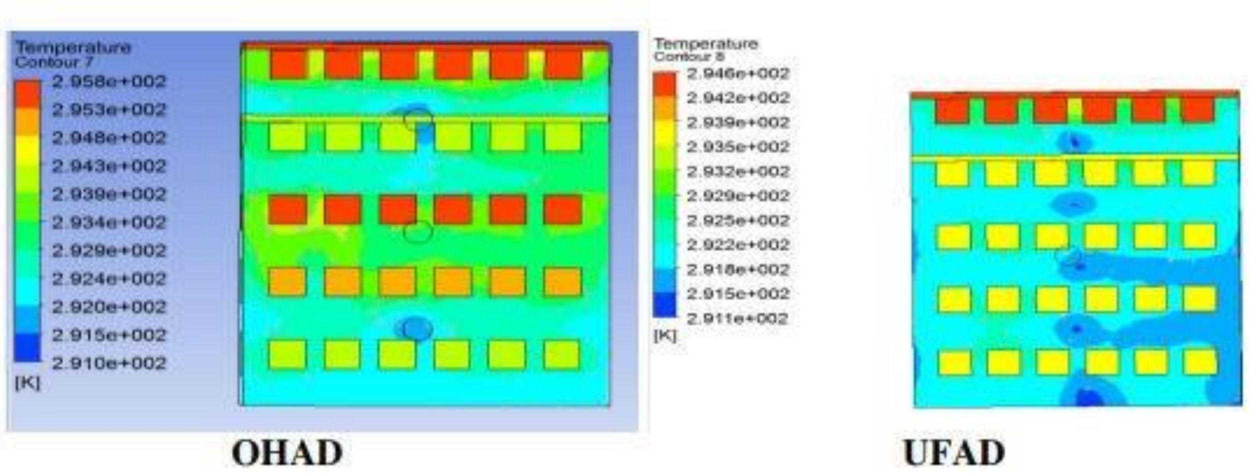
(a)



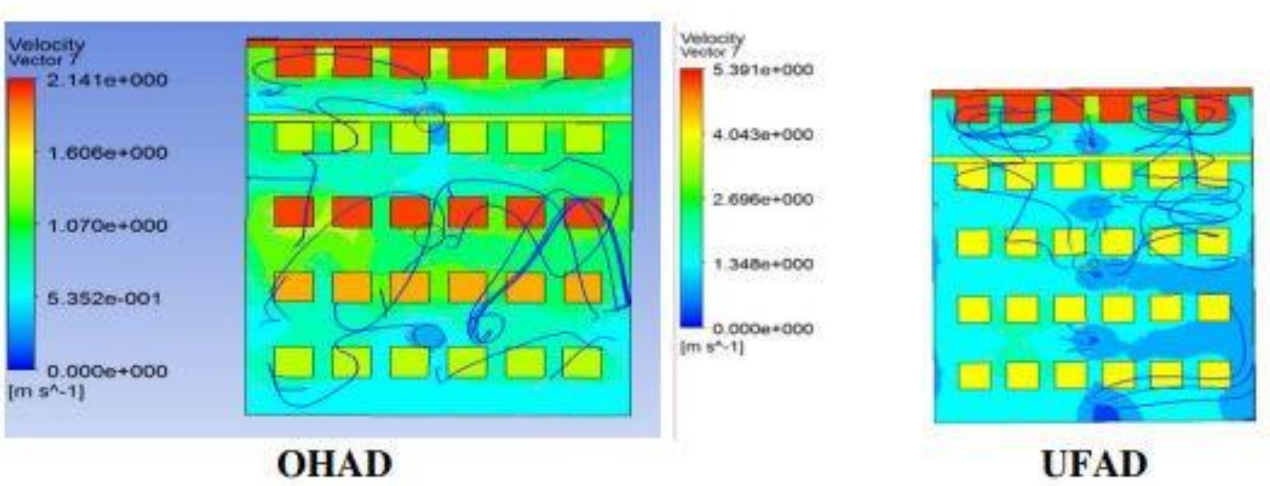
(b)

Fig. 13. Comparison contour (a) Temperature distribution OHAD and UFAD system at Plane 6 (b) Comparison contour velocity magnitude OHAD and UFAD system at Plane 6

iv. Top View



(a)

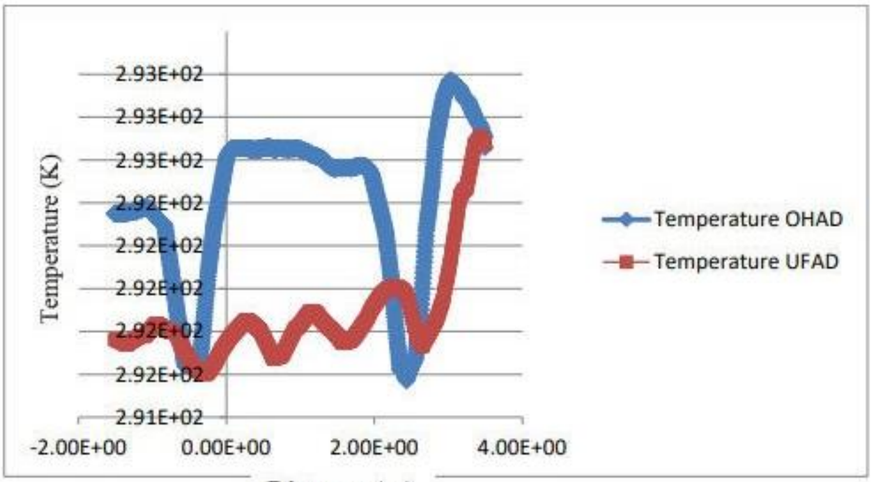


(b)

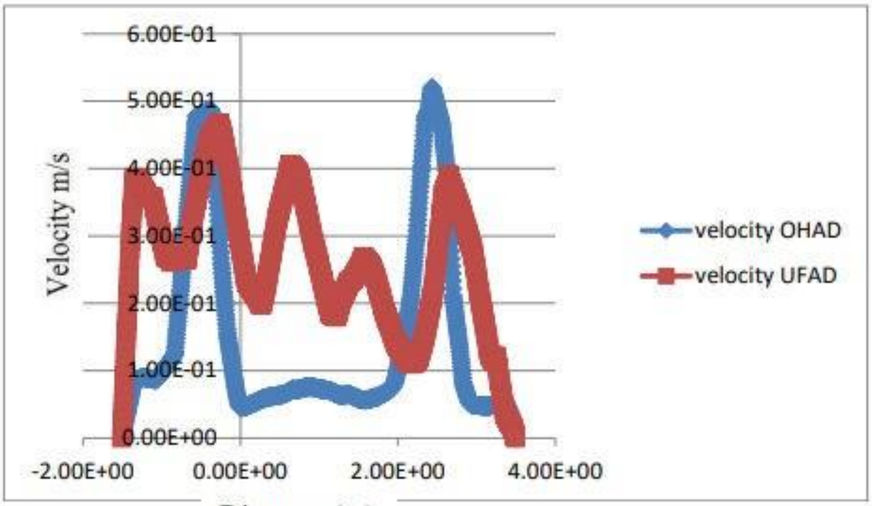
Fig. 14. Comparison contour (a) Temperature distribution OHAD and UFAD system Plane 7 (b) Velocity magnitude OHAD and UFAD system Plane 7

4.3 Comparison Result OHAD vs UFAD System Temperature and Velocity with Distance

Figures 15 - 20 show the temperature and velocity with distance graphs for the UFAD and OHAD systems at the 7 planes (Plane 1, Plane 2, Plane 3, Plane 4, Plane 5 and Plane 6 and Plane 7). The figures show that for UFAD system the flow is uniform and stratified in the occupied zones. In addition, the temperature levels of plane in case of using UFAD system are lower than OHAD system. As a result, for UFAD system, lower temperature variation along the distance is achievable. In contrary, for OHAD model, air at the occupied zone is slightly warmer in temperature. Velocity with distance graph for the UFAD and OHAD systems as shown in Figure 15 - 20. All the results show that there is no discomfort level of air flow velocity occurs in the whole space within ASHRAE recommendation where the threshold limit for discomfort due to air velocity is 0.8 m/s. All the figures show that better temperature, velocity distributions and comfort levels can be obtained using UFAD system.

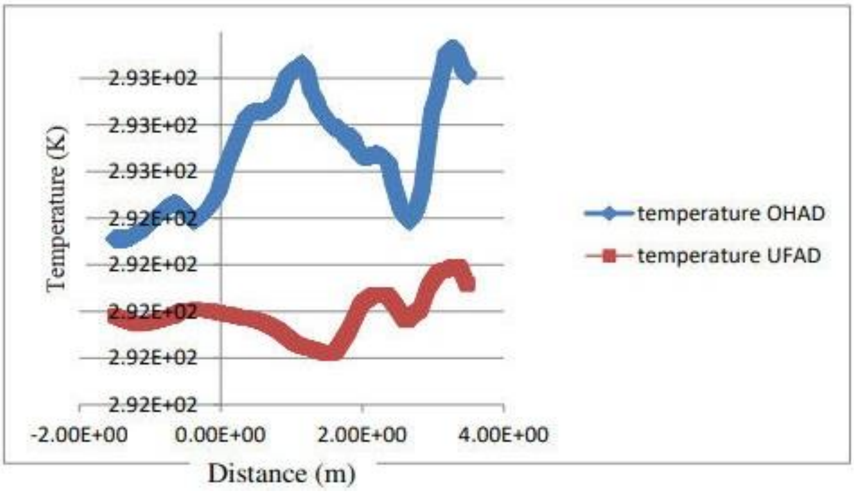


(a)

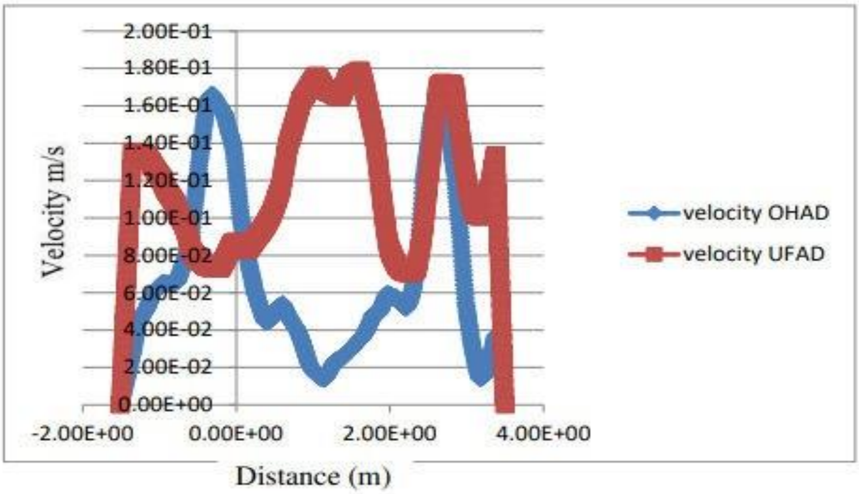


(b)

Fig. 15. Comparison (a) Temperature distribution OHAD and UFAD system graph Plane 1 and (b) Velocity magnitude OHAD and UFAD system graph Plane 1

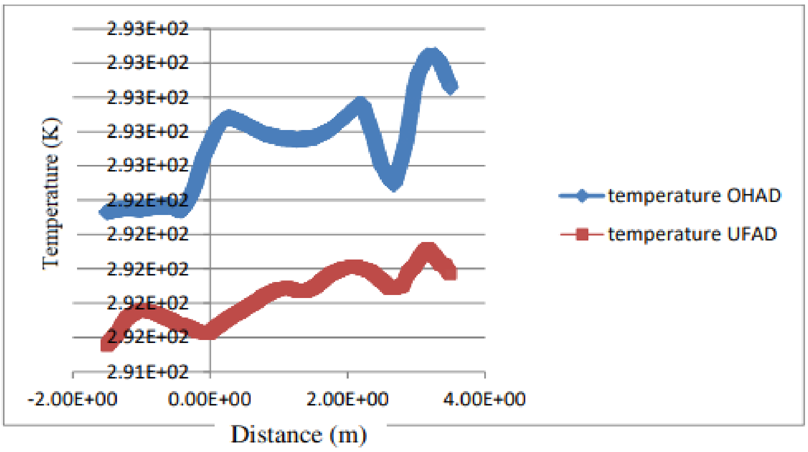


(a)

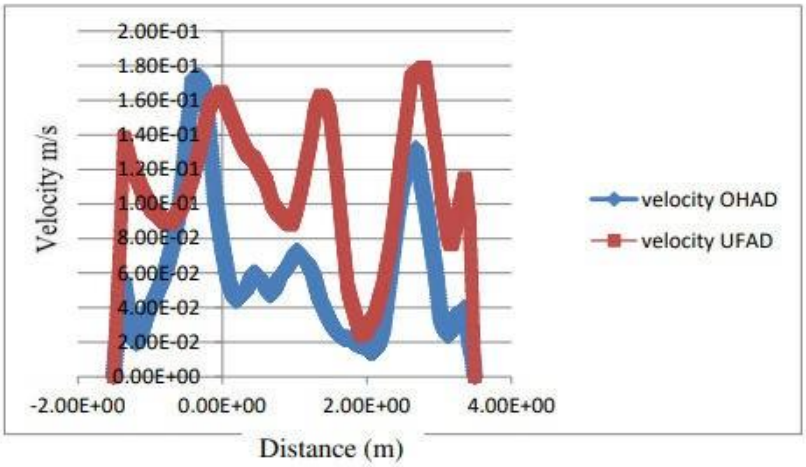


(b)

Fig. 16. Comparison (a) Temperature distribution OHAD and UFAD system graph at Plane 2 and (b) Velocity magnitude OHAD and UFAD system graph at Plane 2



(a)



(b)

Fig. 17. Comparison (a) Temperature distribution OHAD and UFAD system graph at Plane 3 (b) Velocity magnitude OHAD and UFAD system graph at Plane 3

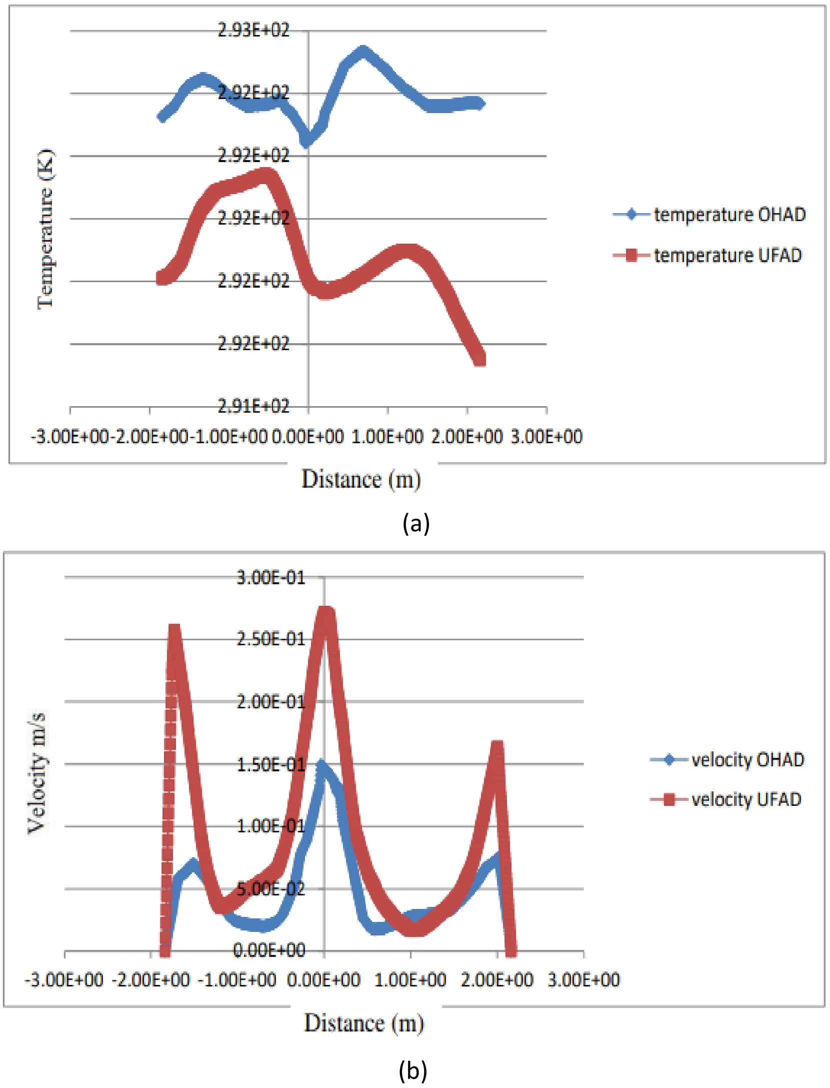
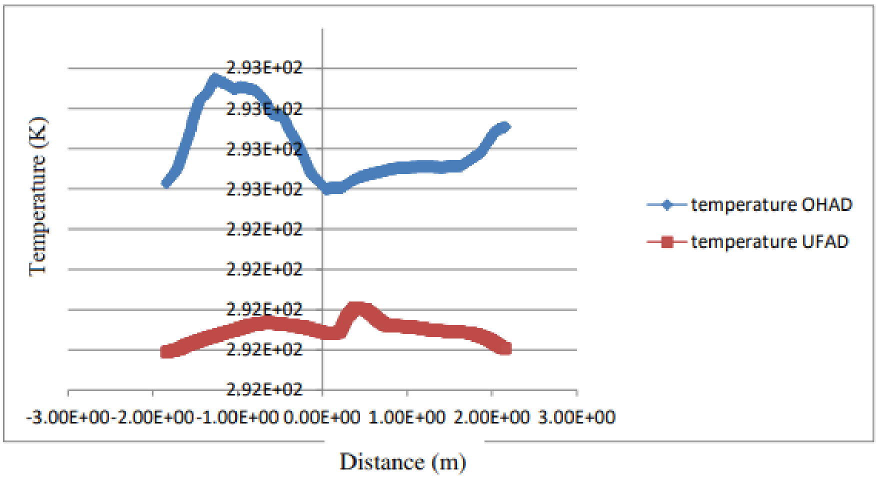
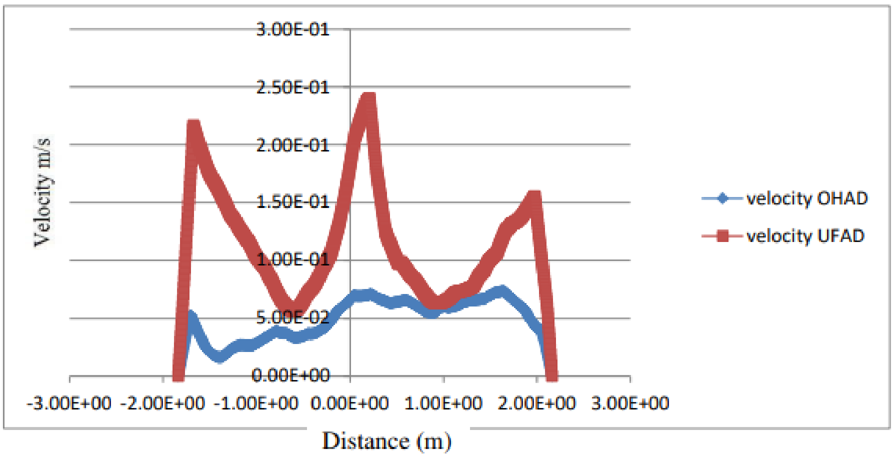


Fig. 18. Comparison (a) Temperature distribution OHAD and UFAD system graph Plane 4 and (b) Velocity magnitude OHAD and UFAD system graph Plane 4

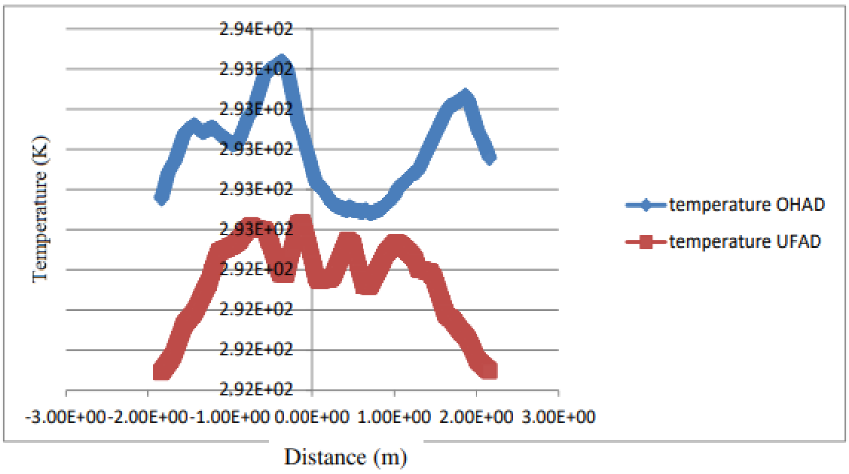


(a)

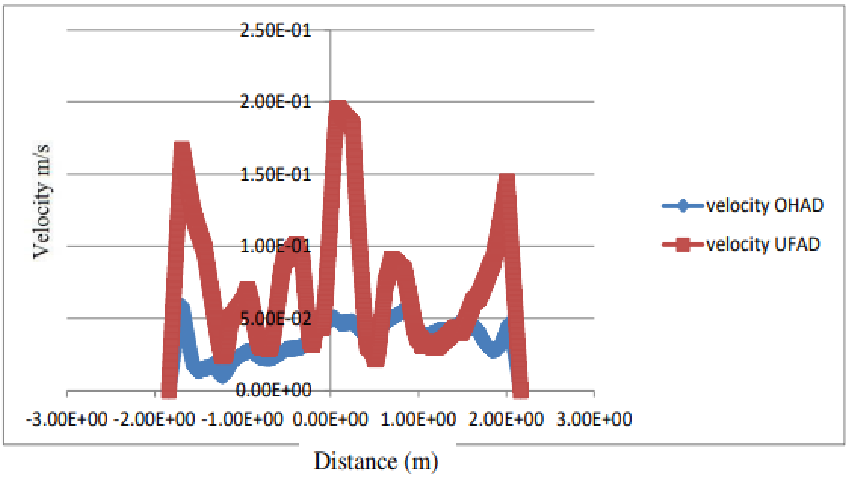


(b)

Fig. 19. Comparison (a) Temperature distribution OHAD and UFAD system graph Plane 5 (b) Velocity magnitude OHAD and UFAD system graph Plane 5



(a)



(b)

Fig. 20. Comparison (a) temperature distribution OHAD and UFAD system graph Plane 6 (b) velocity magnitude OHAD and UFAD system graph Plane 6

5. Conclusion

In this paper, the airflow temperature and velocity distribution within an air-conditioned cinema or theater were numerically investigated under given geometrical, design conditions and configurations. A comparative study between the underfloor air distribution (UFAD) system and the overhead air distribution systems (OHAD) was conducted to assess their suitability for cinema applications, particularly in providing occupants with a comfort zone. The noteworthy contribution of this work lies in the successful application of the numerical technique, which accurately predicted the airflow temperature and velocity distributions within the air-conditioned space of a cinema.

The key finding of the study indicates that the UFAD system is capable in providing effective air temperature and velocity distribution compared to an OHAD system in a cinema application. The airflow temperature and velocity distribution clearly demonstrate the positive impact of the UFAD system, resulting in a more comfortable environment in the cinema. The UFAD system leads to finer temperature distributions and lower mean temperatures. In this study, utilizing a supplied temperature and velocity of 18 °C and 0.8 m/s results in the highest temperature of 22 °C, falling within Malaysia's ASHRAE standard recommendation of 18 – 27 °C.

References

[1] Haines, Roger W., and Douglas C. Hittle. *Control systems for heating, ventilating, and air conditioning*. Springer Science & Business Media, 2006.

[2] Bauman, Fred S. "Giving occupants what they want: guidelines for implementing personal environmental control in your building." (1999).

[3] Ho, Son H., Luis Rosario, and Muhammad M. Rahman. "Comparison of underfloor and overhead air distribution systems in an office environment." *Building and Environment* 46, no. 7 (2011): 1415-1427. https://doi.org/10.1016/j.buildenv.2011.01.008

[4] Fan, Yaming, Xiangdong Li, Yihuan Yan, and Jiyuan Tu. "Overall performance evaluation of underfloor air distribution system with different heights of return vents." *Energy and Buildings* 147 (2017): 176-187. https://doi.org/10.1016/j.enbuild.2017.04.070

[5] Goubran, Sherif, Dahai Qi, Wael F. Saleh, Liangzhu Leon Wang, and Radu Zmeureanu. "Experimental study on the flow characteristics of air curtains at building entrances." *Building and Environment* 105 (2016): 225-235. https://doi.org/10.1016/j.buildenv.2016.05.037

[6] Teodosiu, Raluca, Damien David, Cătălin Teodosiu, Gilles Rusaouën, Viorel Ilie, and Chi-Kien Nguyen. "Experimental and numerical investigation of mechanically ventilated rooms." *Energy Procedia* 112 (2017): 252-260. https://doi.org/10.1016/j.egypro.2017.03.1094

[7] Nada, S. A., H. M. El-Batsh, H. F. Elattar, and N. M. Ali. "CFD investigation of airflow pattern, temperature distribution and thermal comfort of UFAD system for theater buildings applications." *Journal of building engineering* 6 (2016): 274-300. https://doi.org/10.1016/j.jobe.2016.04.008

[8] Peng, Lei, Peter V. Nielsen, Xiaoxue Wang, Sasan Sadrizadeh, Li Liu, and Yuguo Li. "Possible user-dependent CFD predictions of transitional flow in building ventilation." *Building and Environment* 99 (2016): 130-141. https://doi.org/10.1016/j.buildenv.2016.01.014

[9] Yau, Yat Huang, Kai Sin Poh, and Ahmad Badarudin. "A numerical airflow pattern study of a floor swirl diffuser for UFAD system." *Energy and Buildings* 158 (2018): 525-535. https://doi.org/10.1016/j.enbuild.2017.10.037

[10] Khatri, Rahul, Abhay Pratap Singh, and Vaibhav Rai Khare. "Identification of Ideal Air Temperature Distribution using different location for Air Conditioner in a room integrated with EATHE–A CFD based approach." *Energy Procedia* 109 (2017): 11-17. https://doi.org/10.1016/j.egypro.2017.03.036

[11] Hurnik, M., M. Blaszczok, and Z. Popiolek. "Air distribution measurement in a room with a sidewall jet: a 3D benchmark test for CFD validation." *Building and Environment* 93 (2015): 319-330. https://doi.org/10.1016/j.buildenv.2015.07.004

[12] Lee, Moon Soo, Zhenning Li, Jiazhen Ling, and Vikrant Aute. "A CFD assisted segmented control volume based heat exchanger model for simulation of air-to-refrigerant heat exchanger with air flow mal-distribution." *Applied Thermal Engineering* 131 (2018): 230-243. https://doi.org/10.1016/j.applthermaleng.2017.11.094

[13] Gan, Guohui. "Evaluation of room air distribution systems using computational fluid dynamics." *Energy and buildings* 23, no. 2 (1995): 83-93. https://doi.org/10.1016/0378-7788(95)00931-0

[14] Wibron, Emelie. "CFD Modeling of an air-cooled data center." (2015).

[15] Nada, S. A., M. A. Said, and M. A. Rady. "CFD investigations of data centers' thermal performance for different configurations of CRACs units and aisles separation." *Alexandria engineering journal* 55, no. 2 (2016): 959-971. https://doi.org/10.1016/j.aej.2016.02.025

[16] Day, Tony. "Data center cooling." U.S. Patent 7,867,070, issued January 11, 2011.

[17] Höppe, Peter. "Different aspects of assessing indoor and outdoor thermal comfort." *Energy and buildings* 34, no. 6 (2002): 661-665. https://doi.org/10.1016/S0378-7788(02)00017-8

[18] ASHRAE, Standard. "55: Thermal Environmental Conditions for Human Occupancy, American Society of Heating Refrigeration and Air Conditioning Engineers." *Inc., Atlanta*(2004).

[19] Yuan, Ruoyang, Tom Fletcher, Ahmed Ahmedov, Nikolaos Kalantzis, Antonios Pezouvanis, Nilabza Dutta, Andrew Watson, and Kambiz Ebrahimi. "Modelling and Co-simulation of hybrid vehicles: A thermal management perspective." *Applied Thermal Engineering* 180 (2020): 115883. https://doi.org/10.1016/j.applthermaleng.2020.115883

[20] Abdolzadeh, Morteza, Ehssan Alimolaei, and Marcelo Pustelnik. "Numerical simulation of airflow and particle distributions with floor circular swirl diffuser for underfloor air distribution system in an office environment." *Environmental Science and Pollution Research* 26, no. 24 (2019): 24552-24569. https://doi.org/10.1007/s11356-019-05651-8

[21] Qi, Dahai, Sherif Goubran, Liangzhu Leon Wang, and Radu Zmeureanu. "Parametric study of air curtain door aerodynamics performance based on experiments and numerical simulations." *Building and Environment* 129 (2018): 65-73. https://doi.org/10.1016/j.buildenv.2017.12.005

[22] Shokrollahi, Samaneh, Mohammad Hadavi, Ghassem Heidarinejad, and Hadi Pasdarshahri. "Multi-objective optimization of underfloor air distribution (UFAD) systems performance in a densely occupied environment: A combination of numerical simulation and Taguchi algorithm." *Journal of Building Engineering* 32 (2020): 101495. https://doi.org/10.1016/j.jobe.2020.101495

[23] Fan, Yaming, Xiangdong Li, Yihuan Yan, and Jiyuan Tu. "Overall performance evaluation of underfloor air distribution system with different heights of return vents." *Energy and Buildings* 147 (2017): 176-187. https://doi.org/10.1016/j.enbuild.2017.04.070

[24] Yau, Yat Huang, Kai Sin Poh, and Ahmad Badarudin. "A numerical airflow pattern study of a floor swirl diffuser for UFAD system." *Energy and Buildings* 158 (2018): 525-535. https://doi.org/10.1016/j.enbuild.2017.10.037

[25] Kim, Gon, Laura Schaefer, Tae Sub Lim, and Jeong Tai Kim. "Thermal comfort prediction of an underfloor air distribution system in a large indoor environment." *Energy and Buildings* 64 (2013): 323-331. https://doi.org/10.1016/j.enbuild.2013.05.003