CFD Letters 12, Issue 3 (2020) 66-77

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

CFD Analysis on Effect of Air Inlet and Outlet Location on Air Distribution and Thermal Comfort in Small Office

Open Access

Ihab Hasan Hatif[1], Azian Hariri[1,*], Ahmad Fu'ad Idris[1]

[1] Industrial and Indoor Environment Research Group (IIERG), Centre for Energy and Industrial Environment Studies (CEIES), Faculty of Mechanical and Manufacturing Engineering Universiti Tun Hussein Onn Malaysia 86400 Parit Raja, Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 23 January 2020
Received in revised form 18 March 2020
Accepted 23 March 2020
Available online 29 March 2020

ABSTRACT

Air distribution occupies an important position in the building design process as building occupants expect good standards of indoor air quality. Indoor air may uncomfortable because of its temperature, speed, direction, and volume flow rate. It doesn't matter how efficient the ventilation equipment is if the air not distributed well. The main aim of this study was to improve air distribution inside the office and avoided the local temperature difference, extract the warmed and contaminated air before it disperses across the room. A numerical study was performed into the effects of the location of inlet air and outlet air in relation to the room heat sources on air distribution and thermal comfort. A concept of combined indoor heat sources with the exhaust outlet was employed in this investigation. Also, in this research, the developed CFD models were thoroughly validated. This system was adopted for use in office spaces, where the exhaust outlet was located near the heat source. Four different locations studied in four cases with eight lines: four in the different corners and four around the heat source were analysed also the study take Air Distribution Performance Index and ventilation effectiveness as an indicator to the improvement in air distribution. The results showed a significant improvement in (ADPI) from 33% to 83% and the indoor thermal environment was improved using the new locations of the inlets and outlets.

Keywords:
Thermal comfort; mixing ventilation; air distribution; CFD

1. Introduction

The study of ventilation in buildings is essential to improve the quality of the indoor environment. Moreover, the effects of indoor factors such as airspeed and temperature distributions can increase the thermal comfort and benefit the air indoor quality [1]. Numerous analysts showed that higher ventilation rates can be helpful in work office environments [2,3] Ventilation involves the removal of noxious indoor air and the supply and distribution of fresh (outside) air into the indoor environment. Fresh air is necessary for buildings to provide oxygen for respiration and alleviate odors [4]. To evaluate the effectiveness and the strategy of ventilation, it is

* *Corresponding author.*
E-mail address: azian@uthm.edu.my (Azian Hariri)

https://doi.org/10.37934/cfdl.12.3.6677

necessary to quantitatively study the local thermal comfort with different air distribution systems. Dynamic building energy and CFD simulations showed that a change in the visible level of control over the thermal environment [5] and identify appropriate airspeed and air temperature could reduce cooling energy consumption in buildings and saving the total energy cost without losing the thermal comfort of the occupants [6]. Macpherson definite the physical variables affecting thermal sensation (air temperature, air velocity, relative humidity means radiant temperature) [7]. In real circumstances, thermal comfort may not be completely seen just by these variables [8].

Ventilation is achieved through the installation of systems, which could be through natural ventilation [9], mechanical ventilation or hybrid ventilation [10]. The use of mechanical systems helps to increase ventilation rates and it can be designed or adjusted to deliver a specific flow rate. It may also include options to condition and clean the incoming air with cooling and filtering equipment. However, it is associated with energy consumption that comes at a cost [11]. Mechanical ventilation can categorize into several methods: mixing ventilation, displacement ventilation, personalized ventilation, hybrid air distribution, stratum ventilation, protected occupied zone ventilation, local exhaust ventilation, and piston ventilation [12].

Mixing ventilation show better results than displacement ventilation, if considering the expression of thermal comfort and condensation danger. This emphasizes the sturdy effect of the air movement on heat and mass transfer phenomena [13]. The outlet location and diffuser characteristics are very important for indoor air quality and thermal comfort. Because in most cases room air is not well mixed, the thermal comfort at any given point can be different, and the mean value for thermal comfort is not enough. In the case of mixing ventilation, the air is supplied such that the room air is fully mixed. Thus, contaminant concentration is evenly distributed throughout the whole room and the concentration of pollutants is diluted by the incoming ventilation air. In some cases, supply air may not mix with the room air but, instead, flow directly to the extract air opening. This ‘short-circuiting’ reduces the effectiveness of ventilation and should be avoided. Air distribution, designed to supply clean air where, when, and as much as needed. To choose the location and type of supply air diffuser and the location and type of the return air grilles. makes it possible to efficiently achieve thermal comfort, control exposure to contaminants, provide high-quality air for breathing and minimizing the risk of airborne cross-infection while reducing energy use. exposure to indoor pollution may increase due to inefficient air distribution [14].

In this study, the effects of the location of inlet and outlet air in office room where the warm and pollutant air is extracted and their relation to room heat sources on thermal comfort were investigated numerically by using computational fluid dynamic (CFD) software. In addition, the study by using CFD will give better understanding of the location of ventilation inlet and outlet prior to good air distribution acceptable thermal comfort. This work is significant in improving our capacity to predict in a fine way all the processes of indoor ventilation, with specific modification numerical simulations of the air flow process in building. Complementary results obtained through CFD simulations will provide an effort to expansion a deeper understanding of these complex flows of the air distribution. In addition, the CFD simulation is one of the alternative approaches in determining the uncertainties that were unable to be achieved by experimental works in term of reduction of cost as well as time constrain.

2. Methodology

2.1 Case Description

The effect of different locations of the inlet and outlet air ventilation and their relationship with the heat sources and air distribution on the indoor environment were numerically investigated in

Modeling of an office room with mixing ventilation by using ANSYS CFD. The model room (4.8 X 4.8 X 2.65) m^3 Had two inlets via (0.4 X 0.4) m^2 and two outlets via (0.55 X 0.43) m^2 and the other outlet had 0.3 m diameter. Inlet one was in the middle of the top of the room. The distance between the center of inlet one and inlet two was 0.19 m and the distance between the midpoint of the room and the center of outlet one or two was 0.45 m. The center of another outlet was in the centerline of the room and it was about 0.2 m from the edge of the end of the room ceiling. The rest of the air outlet and air inlet locations listed In Table 1. Figure 1 shows the locations of the inlet and outlet to each case.

Table 1
Case study description listed the locations of the inlet and the outlet

Case	Inlet location		Outlet location		
study	Inlet 1	Inlet 2	Outlet 1	Outlet2	Outlet 3
Case 2	X= 3.6	X= 1.2	X= 1.2	X= 3.6	X= 2.4
	Y= 2.5	Y= 2.5	Y= 2.5	Y= 2.5	Y= 2.5
	Z= 1.2	Z= 3.6	Z= 1.2	Z= 3.6	Z= 2.4
Case 3	X= 3.6	X= 1.2	X= 2.4	X= 2.4	X= 2.4
	Y= 2.5	Y= 2.5	Y= 2.5	Y= 2.5	Y= 2.5
	Z= 2.4	Z= 2.4	Z= 4.3	Z= 0.5	Z= 2.4
Case 4	X= 2.4	X= 2.4	X= 4.3	X= 0.5	X= 2.4
	Y= 2.5	Y= 2.5	Y= 2.5	Y= 2.5	Y= 2.5
	Z= 1.2	Z= 3.6	Z= 2.4	Z= 4.3	Z= 2.4

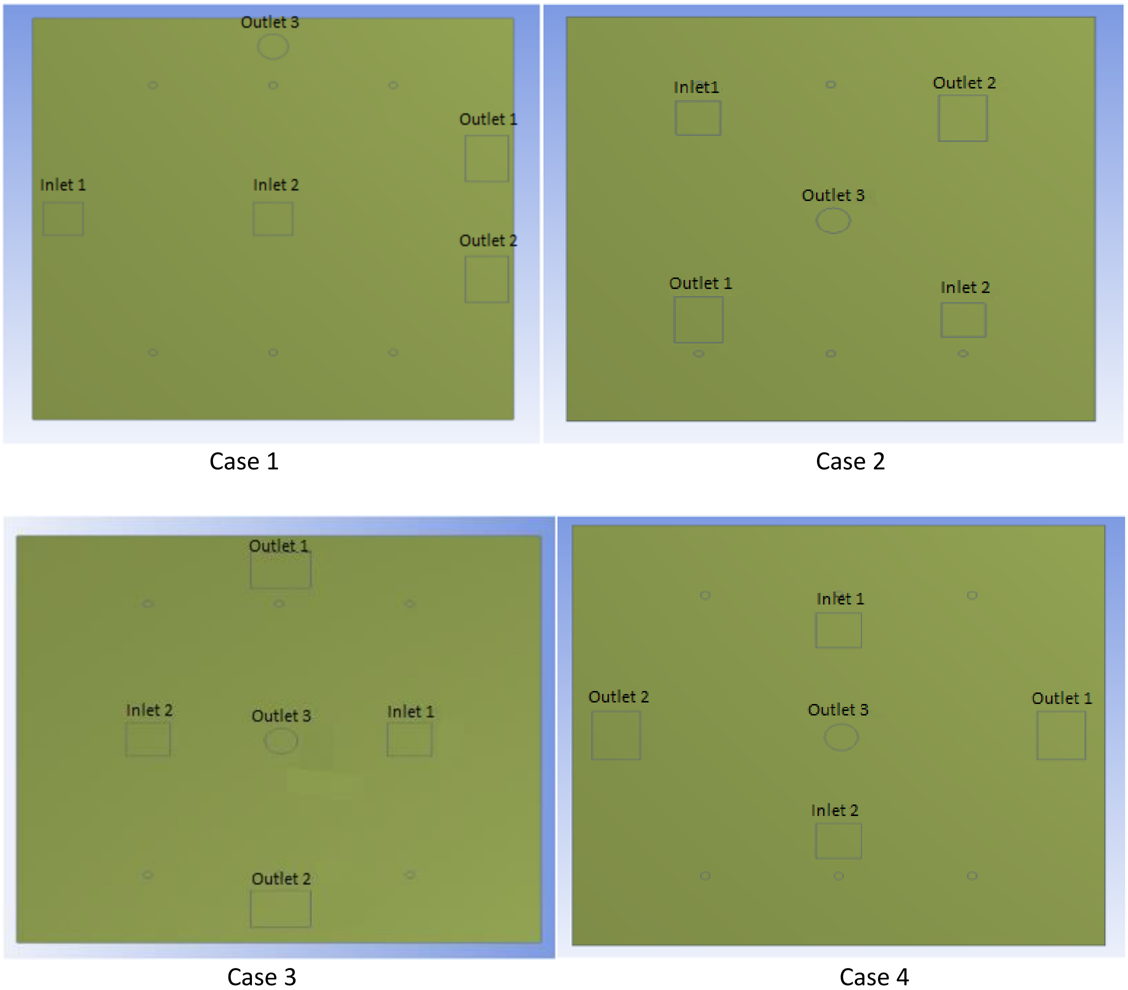


Fig. 1. The locations of inlet and outlet to each case study

The occupant in the room was simulated by four boxes human shape seated [15] and was heated by 60 w/m2 human activity such as seated, reading, or writing [16]. A laptop shape with a dimension of 0.45 m X 0.45 m X 0.05 m, located on the meeting table in the middle of the room. The table had a dimension of 0.18 m long and 0.11 m wide and 0.05 m thick and 0.70 m height. However, the table was simulated without legs, and the chairs were not included because the geometrical was quite complicated and to decrease the computational grid and grid size which would reduce computation time. Boolean was created around generated heat sources such as human body, laptops and lamps Figure 2. Assessment of thermal comfort including air temperature and air velocity were performed in this study the comparison in air temperature distribution in four positions in different corners and five levels of heights inside the occupied zone. The effect of heat sources in the air temperature in four position and five levels of heights near to the heat sources (human and laptop) Figure 3. The simulation results in the present study for the tested room was adopted using many computational runs at various planes and for all the domain.

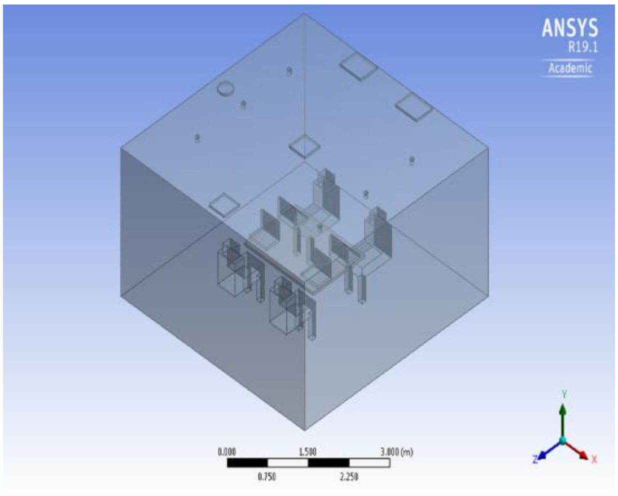


Fig. 2. Physical model

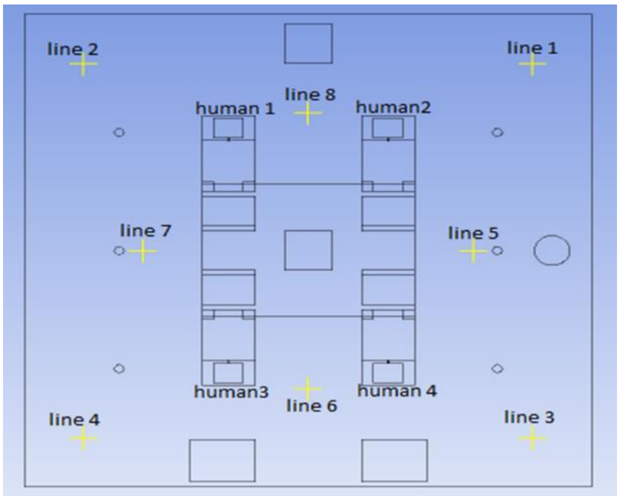


Fig. 3. Schematic diagram of the lines and occupant position in the room

2.2 Modelling and Setup

In the process of setup, all boundary conditions and parameters for room were set. Firstly, the domain should be established for all room geometry. Fluid type and its properties were added in the domain. Parameters were set to air as a continuous fluid while the wall was designated as a solid. In addition, the model should be selected. The turbulence model calculation is important. Before turbulence model was select in modeling solver, the (Re) for predicting the transition laminar to turbulent flow Reynolds number can use for this flow situations. Eq. (1) shows the Reynold number equation.

$$Re = \frac{\rho v D}{\mu} \tag{1}$$

where Re: Reynold number, ρ: density of the air 1.225 (kg/m^3), v: velocity 0.9 m/s, D: is linear dimension (m), μ: is the dynamic viscosity 1.7894 x 10^{-5} (kg m/s)

From the calculation, the result of Re was greater than 24600. The type of fluid model was turbulence model. Therefore, in the setup, renormalization group (RNG) k-ε model were chosen. Many researches have used (RNG) k-ε because it can give more accurate results for the turbulent airflow comparison between other eight different turbulence models [17,18]. Therefore, in this study RNG k-ε model was selected. There are many turbulence models that are accepted by many researchers to be applied for cases such as the one being considered. RNG is one of these models that provides more accurate results in the simulation of air distribution around the human body.

Another modelling setup energy equation that the air movement and detached mass was directed to momentum and energy equations using a finite volume method.

Momentum equation

$$\frac{\partial(\rho u)}{\partial t} + \nabla.(\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau xx}{\partial x} + \frac{\partial \tau yx}{\partial y} + \frac{\partial \tau zx}{\partial z} + \rho f x \qquad (2)$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla.(\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau xy}{\partial x} + \frac{\partial \tau yy}{\partial y} + \frac{\partial \tau zy}{\partial z} + \rho f y \qquad (3)$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla.(\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau xz}{\partial x} + \frac{\partial \tau yz}{\partial y} + \frac{\partial \tau zz}{\partial z} + \rho f z \qquad (4)$$

Energy equation

$$\frac{\partial}{\partial t}\left[\rho\left(e + \frac{V2}{2}\right)\right] + \nabla.\left[\rho\left(e + \frac{V2}{2}\right)V\right] = \frac{\rho\dot{q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) - \frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} + \partial(u\tau xx)}{\partial x} + \frac{\partial(u\tau yx)}{\partial y} + \frac{\partial(u\tau zx)}{\partial z} + \frac{\partial(v\tau xy)}{\partial x} + \frac{\partial(v\tau yy)}{\partial y} + \frac{\partial(v\tau zy)}{\partial z} + \frac{\partial(w\tau xz)}{\partial x} + \frac{\partial(w\tau yz)}{\partial y} + \frac{\partial(w\tau zz)}{\partial z} + \rho f \cdot V \qquad (5)$$

ρ = air density, V = velocity, f = force, P = pressure, τ = shear stress, $\dot{q}$ = heat flux, e = energy.

continuity equation:

$$\partial\rho\partial t + (\rho \boldsymbol{u} i)\ \partial x i = 0 \qquad (6)$$

Eq. (6) represents the time dependent three-dimensional continuity equation for a compressible fluid. For an incompressible flow, the density (ρ) is assumed to be constant, and hence there is no change of the density with time. Therefore, Eq. (6) becomes

$$(\boldsymbol{u} i)\partial x i = 0 \qquad (7)$$

The pressure-velocity connection outline for the solution used was a pressure-based coupling algorithm. Standard pressure and second order for momentum, turbulence kinetic energy in the governing calculations were used by adopting an upwind scheme. Table 2 shows the main parameters and were used as input for the simulation. The table also shows the modeling selection in the setup simulation. The boundary condition for computer, human, lamps, wall ceiling and roof were selected as shown in Table 2.

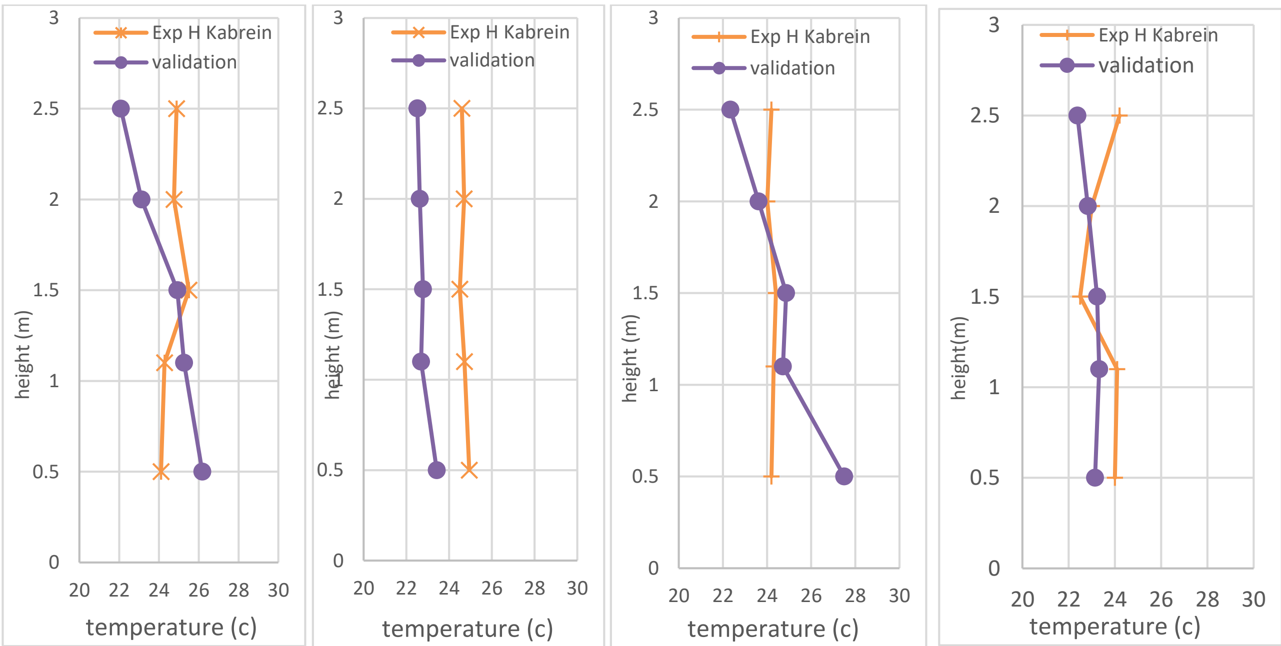
Table 2
Main parameter selected and modelling domain selection

Physics Performance	CFD
Solver Performance	Fluent
Models	
Energy equation	On
Viscose turbulence flow	(RNG) k-ε model
Species model	Species transport
Boundary Condition	
Computer	Heat flux 258.9 W/m2
Occupant	Heat flux 60 W/m2
Supply air	Velocity inlet 0.9 m/s, 0.85 m/s @292k
Lamps	Heat flux 166.6 W/m2
Outlet	Pressure outlet
Wall, ceiling and floor	Adapted

3. Validation of The CFD Algorithm

3.1 Validation

An experimental and CFD modelling for thermal comfort and CO2 concentration in office building was performed by H Kabrein [19].This work was chosen as the validation model of this study. In this study, the published model was modelled with similar geometrical parameters and settings the simulation software ANSYS 19.1 was used for simulation prediction. The results from the validation model were compared to H Kabrein experimental results which showed good agreement. The error ratio between the validation line and the measured line was 5.9%. Figure 4 shows the simulation results validated to experimental for H Kabrein [19]. It found that the simulation results are in good agreement with experimental by Kabrein.



Line 1: X=1, Z= 2.4 (m) Line 2: X=2.4, Z=1 (m) Line 3: X=3.8, Z=2.4 (m) Line 4: X=2.4, Z=3.8 (m)

Fig. 4. Exp Kabrein *et al.,* [19] work and validation temperature profile

3.2 Grid Independence

As a result of the room and equipment complexity, ANSYS CFD software was used in this study to generate a tetrahedral mesh the significant role in the simulation study was to check the mesh quality by grid independence test. To make sure that this result not sensitive to the quantity of mesh the grid independence test helpful for this purpose. The grid on all sides of the occupants and other objects were designed to be good enough to solve the boundary layer and capture thermal environment behaviour. In other words, the solution is not depending on the mesh. In this study, the grid size had been chosen by comparing the simulation results for different mesh sizes as listed in Table 3. Figure 5 shows that increasing grid cells from mesh two to mesh three, there is no considerable change in the temperature and velocity. thus, mesh 2 with 2083707 cells had been chosen to be sufficed mesh for the rest of the simulations.

Table 3
Mesh independence test

Type of mesh	Cells number
Mesh 1	1288922
Mesh 2	2083707
Mesh 3	2869325

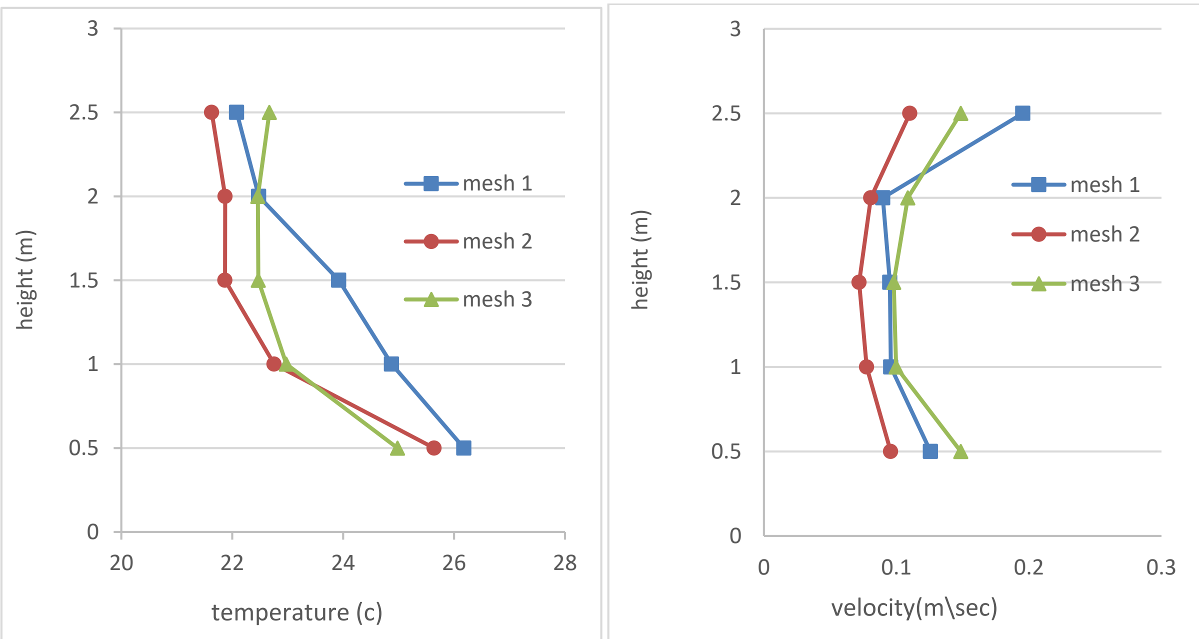


Fig. 5. Mesh independence test for temperature profile °C velocity profile (m/sec)

4. Result and Discussion

4.1 Temperatures Distribution

The temperature difference between the different places inside the room should not be more than 3 °C and the effective draft temperature is between -1.5 and + 1 ºC Figure 6 shows the locations of four lines through points 1, 2, 3 and 4, two being on the four corners inside the occupied zone. Figure 7 shows the predicted temperature difference at ΔT (average for the maximum temperature and average for the minimum temperature) for all case studies. Although the thermal comfort was satisfactory. In case 1, there is a large difference in temperature at 2.3 ºC which means that the difference between the temperature in the four points is large because of the

CFD Letters
Volume 12, Issue 3 (2020) 66-77

location of the air inlets. While the case 2 and case 3 shows better result than case 1 with the least difference in temperature differences while case 4 , where the entry sites in the middle of the room near the sources of heat the better result between all cases and the smallest difference between the temperatures inside the room, which means case 4 has better thermal distribution and more thermal comfort Figure 8 shows a comparison between the amount of heat at the head level of every human in each case. The case 1 has the highest temperature variation between humans. This is due to the location of human 4, human 3 near the cold air inlet, human 2, human 1 near the exit, One and a human 3 which is closer to the air outlet. The figure shows that the fourth case was the lowest variance between the temperature of each person.

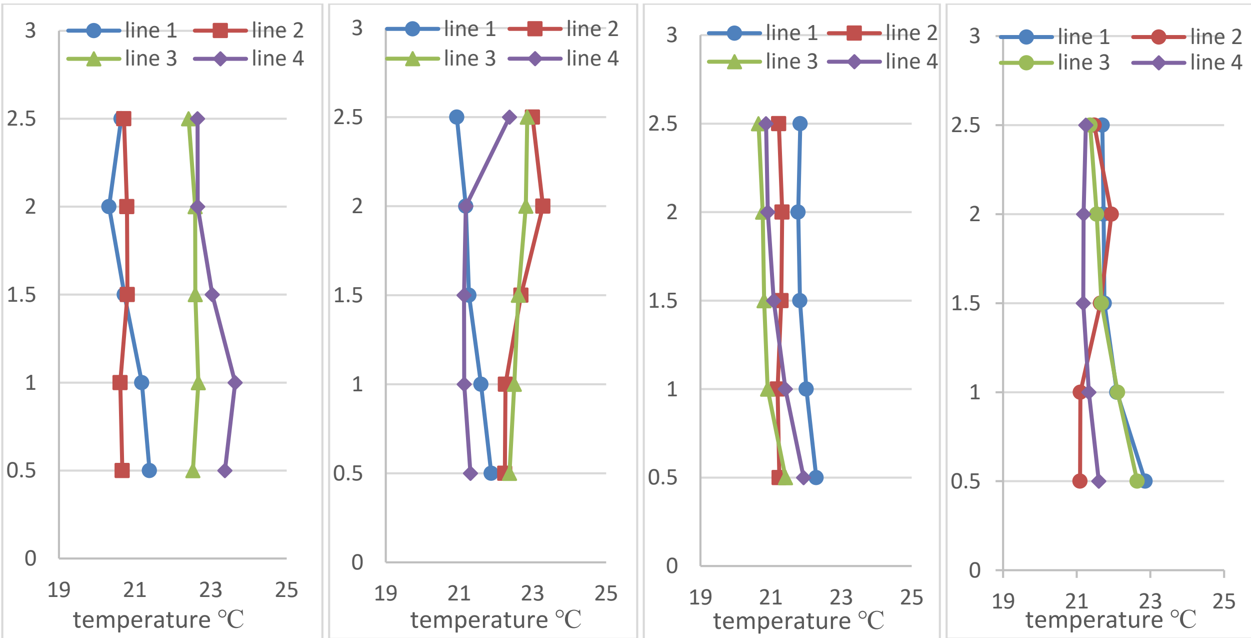


Fig. 6. Simulation of temperature profile in four line in different corners

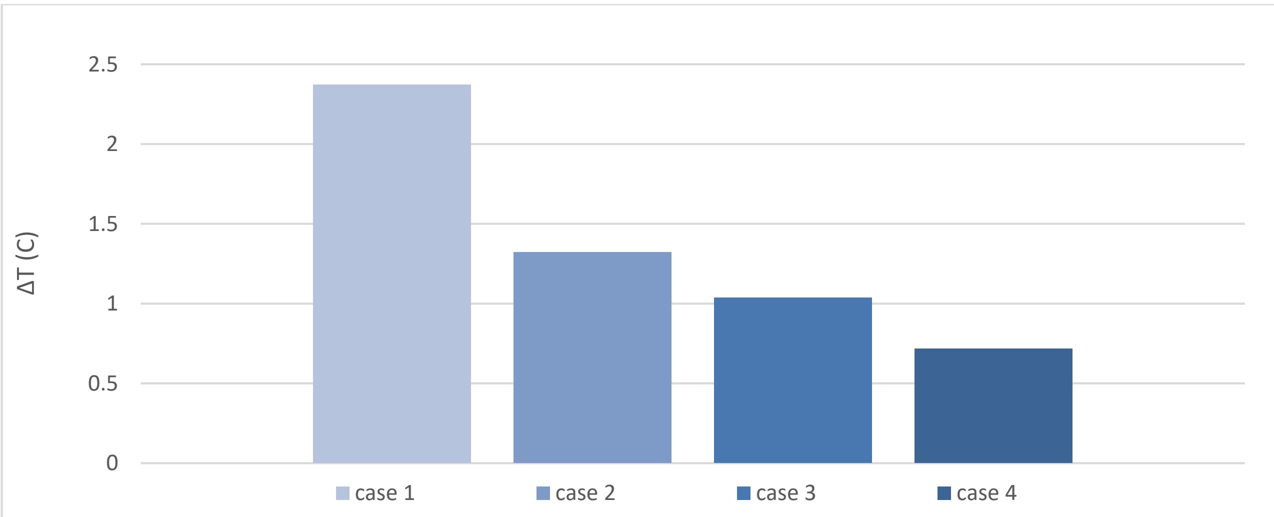


Fig. 7. The different in maximum average temperature and minimum average temperature

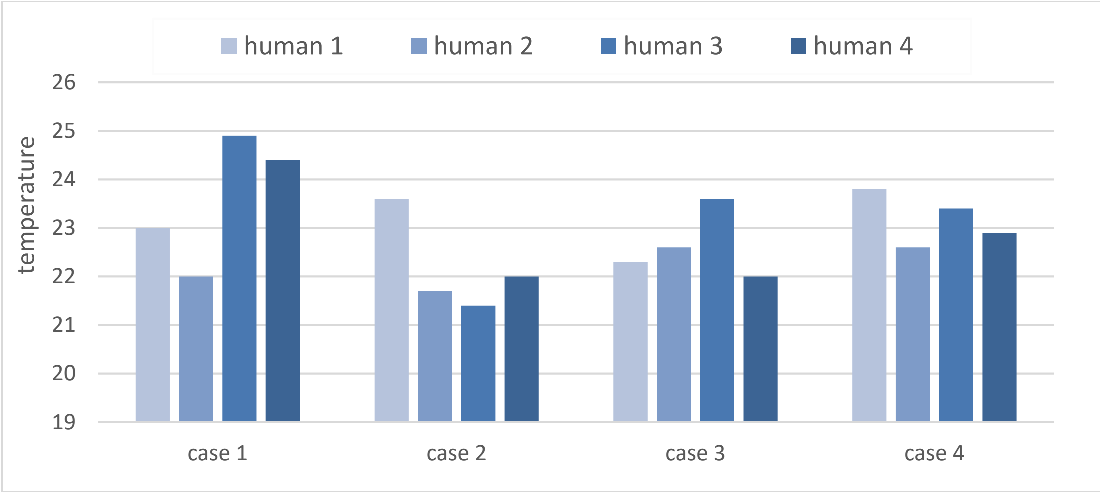
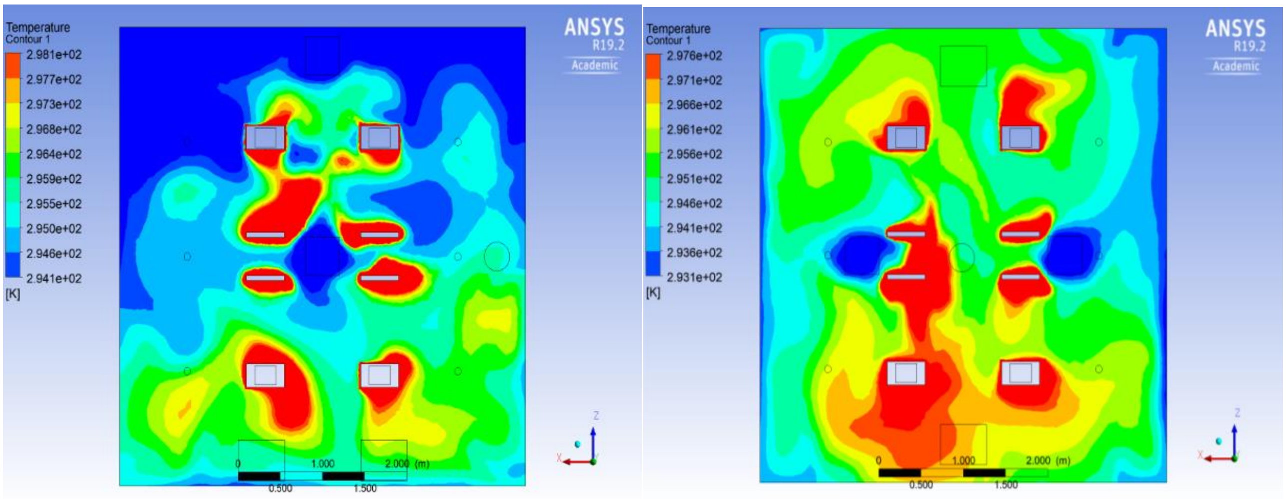


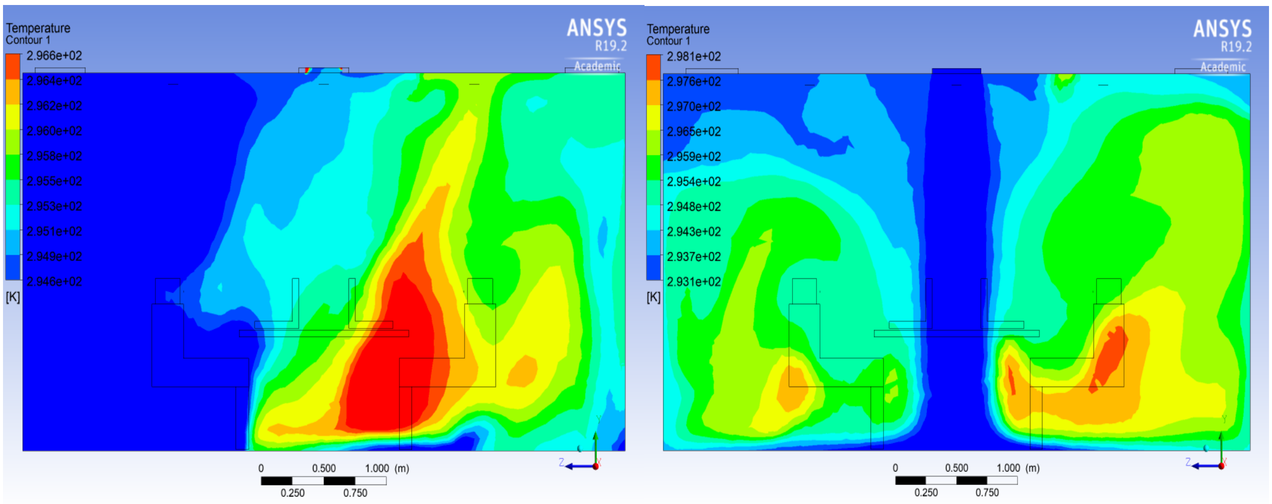
Fig. 8. The air temperature gradient at point located at head level of each person

Figure 9 a and b display the comparison for the contours of air temperature distribution in horizontal plane and height 1 m near to the occupied between case 1 and case 4.



(a) Case 1, plane(X,Z) Y=1m

(b) Case 4, plane (X,Z) Y=1m



(c) Case 1, plane(Z,Y) X=4.3m

(d) Case 4, plane (Z,Y) X=4.3m

Fig. 9. The air temperature distribution for case 1 and case 4

Figure 9 c and d shows the comparison in the side view in plane (Z,Y) from the figure it can be observed that in case 1 there are two area one is low temperature where the inlet air located and the other high temperature where the outlet air located while in case 4 the air temperature is distributed uniformly in all area.

4.2 Air Velocity Distribution

The air velocity could be a significant parameter for human thermal comfort because the increased air velocity will lead to a cooling effect. However, when the air velocity is too high, this may cause discomfort sensation. ASHRAE standard recommended that air velocity should not be more than 0.2 m/s and not recommend for the less air velocity. The air velocity in these cases was calculated numerically in eight lines for five levels of heights also the air velocity inside the room was investigated at locations around the occupants. The maximum air velocity was recorded in the case 3 and case 4. where the velocity in line 6 and line 8 which located on the same side of the air inlet is higher than line 1 and line 2 which located in the opposite sit side for case 3 the maximum velocity in line which located near to the humans. However, the four lines become very close to each other in the level near to the floor. For case 1 and case 2 the all lines close to each other and the average air velocity inside the chamber, ranged between 0.13 to 0.2 m/s. At a height of 1 m, the air velocity concentrated between human 1 and human 4, also, on both sides of room near to the sidewall. The air spread down and the height of an airstream due to the reflection of the airway in the ground at the area between inlet 1 and inlet 2. Figure 10 shows the air velocity above each person's head. In all cases, all the results were satisfactory and within the tolerable temperature range between 20 ºC to 26 ºC and air velocity less than 0.2 (m/sec).

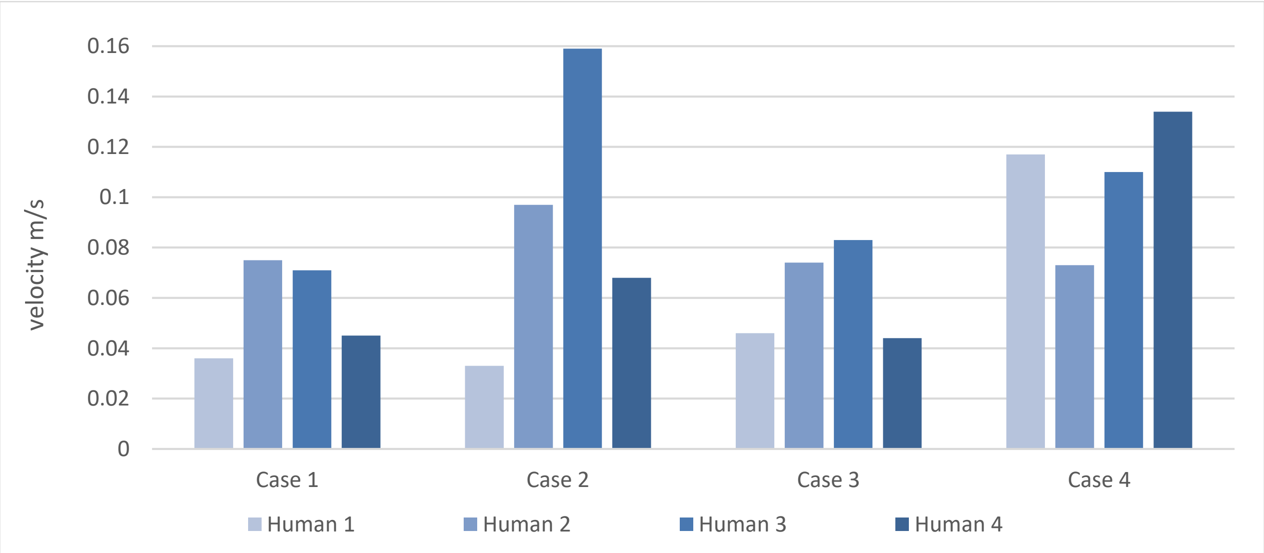


Fig. 10. The air velocity at point located at the head level of each person

4.3 ADPI and Effectiveness of Ventilation

Air Distribution Performance Index (ADPI) and effectiveness temperature were determined to indicaces the amount of the improvement of air distribution as illustrated in Table 4 variables were illustrated for each of the cases at different height level and for different points in four corners within occupied zone.

Table 4
Air Distribution Performance Index (ADPI) effectiveness temperature

	Case 1	Case 2	Case 3	Case 4
ADPI%	33.33	66.6	75	83.33
Effectiveness	1.034	1.039	1.11	1.26

5. Conclusions

The air distribution system has a major influence on various indoor environmental system parameters such as thermal comfort, energy efficiency, indoor air quality. This study was conducted by CFD simulation in an office room to improve the indoor environment system and thermal comfort through the arrangement of air distribution systems with different locations of the inlet and outlet air inside the environmental chamber represent the office room. However, the advantage of the numerical simulation is the ability to depict the air velocity and air temperature distribution at any layers or location in the room. The assessment of air distribution depends on the purpose of the airflow distribution which might need different indices like air distribution index ADPI and ventilation effectiveness.

In this investigation, the effects of the location of the inlet and outlet air ventilation on the thermal indoor comfort and vertical and horizontal temperature different inside the occupied zone in an environmental chamber were investigated and the result was concluded as follows

i. A significant improvement achieved when the outlet was combined with the room heat sources in case 2 and 3 where outlet 3 located in the centre of the room between the heat sources. Also, in case 3 where the outlet 1 and outlet 2 was closed and combined with the lamps.
ii. The result of the investigation also showed that the location of the inlet and outlet of air in different corners inside the room as in case 2 more efficient and effective than case 1 where the locations of the inlet in a side and the outlet on another side of the room. also, a significant improvement was made when delivering the supply air to a specific point in a room like a case 3 when the supply air was delivered to the human. case 4 showed the lowers temperature different and best ventilation effectiveness, overall a better indoor thermal environment and air distribution in term of thermal comfort was achieved by combining the indoor heat sources withe the outlet and select location far as possible from the inlet which will increase the ventilation effectiveness and avoid short-circuit.

Acknowledgement
The authors would like to thank the Centre of Energy and Industrial Environment Studies (CEIES) Faculty of Mechanical and Manufacturing Engineering in Universiti Tun Hussein Onn Malaysia, also our acknowledgement to the Computational Fluid Dynamics laboratory (CFD) lab of UTHM.

References

[1] Chen, Wenhua, Shichao Liu, Yunfei Gao, Hui Zhang, Edward Arens, Lei Zhao, and Junjie Liu. "Experimental and numerical investigations of indoor air movement distribution with an office ceiling fan." *Building and Environment* 130 (2018): 14-26. https://doi.org/10.1016/j.buildenv.2017.12.016

[2] Tarantini, Mariantonietta, Giovanni Pernigotto, and Andrea Gasparella. "A co-citation analysis on thermal comfort and productivity aspects in production and office buildings." *Buildings* 7, no. 2 (2017): 36. https://doi.org/10.3390/buildings7020036

[3] Al Horr, Yousef, Mohammed Arif, Amit Kaushik, Ahmed Mazroei, Martha Katafygiotou, and Esam Elsarrag. "Occupant productivity and office indoor environment quality: A review of the literature." *Building and environment* 105 (2016): 369-389. https://doi.org/10.1016/j.buildenv.2016.06.001
[4] Hajdukiewicz, Magdalena, Marco Geron, and Marcus M. Keane. "Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room." *Building and Environment* 70 (2013): 73-89. https://doi.org/10.1016/j.buildenv.2013.08.020
[5] Yun, Geun Young. "Influences of perceived control on thermal comfort and energy use in buildings." *Energy and Buildings* 158 (2018): 822-830. https://doi.org/10.1016/j.enbuild.2017.10.044
[6] Muhieldeen, M. W., and Y. C. Kuang. "Saving Energy Costs by Combining Air-Conditioning and Air-Circulation using CFD to Achieve Thermal Comfort in the Building." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 58, no. 1 (2019): 84-99.
[7] Yao, Runming, Baizhan Li, and Jing Liu. "A theoretical adaptive model of thermal comfort–Adaptive Predicted Mean Vote (aPMV)." *Building and environment* 44, no. 10 (2009): 2089-2096. https://doi.org/10.1016/j.buildenv.2009.02.014
[8] Rupp, Ricardo Forgiarini, Natalia Giraldo Vásquez, and Roberto Lamberts. "A review of human thermal comfort in the built environment." *Energy and Buildings* 105 (2015): 178-205. https://doi.org/10.1016/j.enbuild.2015.07.047
[9] Kamar, Haslinda Mohamed, Nazri Kamsah, and J. L. Kam. "Indoor air of a double-storey residential house in Malaysia." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 31, no. 1 (2017): 11-18.
[10] Kim, Jungsoo, Federico Tartarini, Thomas Parkinson, Paul Cooper, and Richard De Dear. "Thermal comfort in a mixed-mode building: Are occupants more adaptive?." *Energy and Buildings* 203 (2019): 109436. https://doi.org/10.1016/j.enbuild.2019.109436
[11] Teodosiu, Catalin, Viorel Ilie, and Raluca Teodosiu. "Numerical prediction of thermal comfort and condensation risk in a ventilated office, equipped with a cooling ceiling." *Energy Procedia* 85 (2016): 550-558. https://doi.org/10.1016/j.egypro.2015.12.243
[12] Cao, Guangyu, Hazim Awbi, Runming Yao, Yunqing Fan, Kai Sirén, Risto Kosonen, and Jianshun Jensen Zhang. "A review of the performance of different ventilation and airflow distribution systems in buildings." *Building and Environment* 73 (2014): 171-186. https://doi.org/10.1016/j.buildenv.2013.12.009
[13] Pantelic, Jovan, and Kwok Wai Tham. "Adequacy of air change rate as the sole indicator of an air distribution system's effectiveness to mitigate airborne infectious disease transmission caused by a cough release in the room with overhead mixing ventilation: a case study." *HVAC&R Research* 19, no. 8 (2013): 947-961. https://doi.org/10.1080/10789669.2013.842447
[14] Evola, G., A. Gagliano, L. Marletta, and F. Nocera. "Controlled mechanical ventilation systems in residential buildings: Primary energy balances and financial issues." *Journal of Building Engineering* 11 (2017): 96-107. https://doi.org/10.1016/j.jobe.2017.04.010
[15] ASHRAE, ANSI. "Standard 55-2004, thermal environmental conditions for human occupancy, atlanta: american society of heating, refrigerating, and air-conditioning engineers." *Inc., USA* (2004).
[16] Mahyuddin, Norhayati, Hazim B. Awbi, and Emmanuel A. Essah. "Computational fluid dynamics modelling of the air movement in an environmental test chamber with a respiring manikin." *Journal of Building Performance Simulation* 8, no. 5 (2015): 359-374. https://doi.org/10.1080/19401493.2014.956672
[17] Chen, Q. "Comparison of different k-ε models for indoor air flow computations." *Numerical Heat Transfer, Part B Fundamentals* 28, no. 3 (1995): 353-369. https://doi.org/10.1080/10407799508928838
[18] Zaki, Sheikh Ahmad, Nur Farhana Mohamad Kasim, Naoki Ikegaya, Aya Hagishima, and Mohamed Sukri Mat Ali. "Numerical Simulation on Wind-Driven Cross Ventilation in Square Arrays of Urban Buildings with Different Opening Positions." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 49, no. 2 (2018): 101-114.
[19] Kabrein, H., A. Hariri, A. M. Leman, M. Z. M. Yusof, and A. Afandi. "Experimental and CFD modelling for thermal comfort and CO2 concentration in office building." In *IOP Conference Series: Materials Science and Engineering*, vol. 243, no. 1, p. 012050. IOP Publishing, 2017. https://doi.org/10.1088/1757-899X/243/1/012050