Experimental and CFD modelling for thermal comfort and CO2 concentration in office building

H Kabrein\(^1,a\), A Hariri\(^1,b\), A M Leman\(^2\), M Z M Yusof\(^1\) and A Afandi\(^1\)

\(^1\)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
\(^2\)Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia

E-Mail: hashimkabrein@gmail.com\(^a\), azian@uthm.edu.my\(^b\)

Abstract. Computational fluid dynamic CFD was used for simulating air flow, indoor air distribution and contamination concentration. Gases pollution and thermal discomfort affected occupational health and productivity of work place. The main objectives of this study are to investigate the impact of air change rate in CO\(_2\) concentration and to estimate the profile of CO\(_2\) concentration in the offices building. The thermal comfort and gases contamination are investigated by numerical analysis CFD which was validated by experiment. Thus the air temperature, air velocity and CO\(_2\) concentration were measured at several points in the chamber with four occupants. Comparing between experimental and numerical results showed good agreement. In addition, the CO\(_2\) concentration around human recorded high, compared to the other area. Moreover, the thermal comfort in this study is within the ASHRAE standard 55-2004.

1. Introduction
Computational Fluid Dynamics (CFD) is a numerical technique to solve fluid flow issues. Today’s CFD able to perform use very sizable amount of calculations to simulate the behaviour of fluids in advanced environments and geometries [1]. The ventilation rates influence the concentration of carbon dioxide and also affected of uniformly of CO\(_2\) distribution. Numerical analyses can be helpful in engineering to predict of air distribution and the average CO\(_2\) concentration in selected samplings location in a place [2]. The numerical analysis can be positively used to estimate distribution of carbon dioxide exhaled by occupants in a room [3]. However, the CFD modelling should first identify the ventilation positions which air inlet, outlet, the room layout, number of occupants and types of equipment used. According to Bulińska et al. (2014) [3], numerical analysis can be positively used to estimate the distribution of carbon dioxide exhaled by occupants in a room.

Under a similar air supply volume and particle possession conditions, the displacement ventilated area has a lower deposition rate and bigger number of infiltrating particles than the mixing ventilation. Also, the average particle concentration within the displacement case is larger than the mixing one [4]. In the same condition, between stratum ventilation and displacement ventilation, the particle concentration in breathing zone were less in stratum ventilation compared to displacement ventilation [5]. Mixing ventilation had better results compared to displacement ventilation in terms of thermal comfort and condensation risk. This emphasises the sturdy influence of the air movement on heat and mass (humidity) transfer phenomena that turn up in airy rooms equipped with cooling ceilings [6]. Comparing between types of ventilations for gaseous contaminants, the results showed that the stratum
ventilation was better than the displacement ventilation in terms of the inhaled air quality by the occupant. However, when the source located around the office walls, and the supply air temperature is 19 °C the ventilation is better than to stratum ventilation for the inhaled air quality [7]. Study by Zhang & Chen, (2006) (8) showed the numerical analysis could be used to estimate the particle distribution during different ventilation system such as ceiling and side wall air supply system and under floor air distribution (UFAD) has more influence, that due to the profile of air flow inside the room. Also, he found that (UFAD) system has high efficiency for particle removal and efficiency reduction the air contamination. Catalina, & Kuznik, (2009) [9] conducted the study about chilled ceiling panel, his CFD results showed that the air velocity discomfort was identified at feet/ankle zone but good values of air velocity were found of the rest of the test place. In addition, the results of air temperature as a vertical profile showed asymmetry is less than 1 °C and the maximum difference between globe temperature and the air temperature was 0.8 °C. Non uniform temperature distribution in office building appears with high record temperature, especially around the ceiling and above the ventilation zone. Also the air velocity in the centre of the office was 2 m/s that due the higher air flow rate compared to other place. The CO₂ concentration in the occupied zone was controlled within the target concentration level, but with approximately 20% defence with the CO₂ concentration at any point [10]. (Predicted mean vote) PMV is the greatest widely used thermal comfort index today. Six factors influence PMV index value are: air temperature, air velocity, metabolism of the human body from activity, clothing insulation, air humidity and mean radiant temperature [11]. Acceptable thermal comfort is measured by PMV and (Predicted Percentage Dissatisfied) PPD. The fluctuation of mean air temperature must be not too much and around 24 °C [12].

Increasing the air velocity, will not improve thermal comfort if the relative humidity and air temperature are too high. To get a comfortable sensation, the mean air velocity should be less than 1.5 m/s in the occupied zone. Whereas, the air velocity more than 0.2 m/s in the feet and ankles’ level will effect comfort of the occupants [9]. According to Mahyuddin & Awbi, (2010) [2], the accurate estimation of CO₂ concentration in the room must be using several sensors’ in varying level vertical or horizontal. The re-normalization k-ε (RNG k-ε) model can be used in a numerical simulation of stratium ventilation, the current model showed good results were compared between numerical and experimental data [13,14]. Q. Chen, (1995) [15], provides eight different kinds of turbulence model and he concluded that for all these models one is more accurate that is re-normalization group (RNG) k-ε model.

The main objectives of this study are to investigate the impact of air change rate in CO₂ concentration and to estimate the profile of CO₂ concentration in the offices building.

2. Method
This study used the results measured from experimental to model CFD simulation. The experimental method in this study was carried out using the environmental control chamber to simulate a typical office environment. The chamber dimension is 4.8 m x 4.8 m x 2.65 m. A schematic diagram of the chamber setup is shown in figure 1, the walls and the ceiling were of polyurethane insulation board of 10 cm thickness, the design of the joint, the groove and the tongue that made from stainless steel metal (5 mm) and located in the chamber on both sides. The chamber was equipped with HVAC system, compressor (10 HP), outdoor condenser and fan coil unit in the mixing room, which is located on the top of the chamber roof. The main supply duct was distributed in two main branches to provide air through ceiling diffusers (0.4 m x 0.4 m); moreover, there are two holes (0.55 m x 0.45 m) provided on the return air duct suspended on the top of the ceiling connecting the chamber and mixing box. The outdoor fresh air intake was in the mixing room through the flexible duct, exhaust fan, and damper. The damper was connected by a flexible duct to warrant that sufficient supply of outdoor fresh air compatible to ASHRAE standard 62.1 2007 [16]. Another damper was also connected to exhaust fan at the top of the ceiling to control the exhaust air.
The air flow through the system was adjusted to 560 CFM (951 m$^3$/h) to achieve a face velocity 0.9 m/s through any diffusers, with air changes rate about 3 h$^{-1}$ (typical air change rate for an office building) as suggested by ASHRAE standard 55-2004, in this study the chamber occupancy by four occupants the fresh air volume was controlled of 84 CFM (142.7 m$^3$/h). The chamber environmental condition was adjusted between 23 ºC to 26 ºC and 78% for RH, which is the indoor environmental condition that suggested in Malaysia by the studies of [17,18]. The control of the air temperature and relative humidity by using AHU system, through temperature and relative humidity monitoring system to adjusting the temperature.

The chamber study was conducted in two cases. The first case involved in recording the environmental condition; daily in two months, simulating typical work hours from 8:00 am to 5:00 pm with four occupants in the chamber. Thermal environmental parameters were recorded. The second case involved the measuring CO$_2$ gas concentrations in the chamber. The temperature and relative humidity were recorded using four sensors (KIMO 100) that connected to the PC device using temperature and relative humidity monitoring system. Whereas, air velocity was measured using VelociCalc plus model 8324/8384. The instrument located at five levels of height. The first one at 0.5 m above the floor that represented the occupant seated at the knee high, while 1.1 m high presented the occupant level at sitting condition, and point is 1.5 m at standing level [11]. Additional points, 2 m, 2.5 m. The measurement of CO$_2$ concentration was recorded using an IAQCalc device model 8760/8762 at five levels of heights also in the chamber, the measurement at any location was repetitive three times to confirm the reliability of the test. The air temperature and relative humidity were recorded every 5 minutes interval using the chamber monitors / sensors.

2.1. Numerical method

2.1.1. Computational geometry and modeling

In this study Ansys fluent 16.1 software was chosen to run CFD simulation for case study. The model geometry in this study (length x, high y, width z) is 4.8 *2.65 * 4.8 m as show in figure 1. The fresh air was supplied into the mixing room through an inlet located at the top of the mixing room ceiling, with dimension 0.3 m diameter. The exhaust air from the chamber through an outlet located on the top of ceiling with a dimension 0.3 m diameter, the centre of the circle (x= 0.3 m, z= 2.4 m), two holes inlet and 2 holes return air located in the top of the ceiling, rectangular (0.4 * 0.4 m) and (0.55 *0.45 m2) inlet and return air respectively, the centre of inlet holes 1 and 2 positions are (x= 2.4 m, z= 0.28 m) and (x= 2.4 m, z= 2.4 m) respectively. Thus the centre of return air holes 1 and 2 positions are (x= 1.7 m, z= 4.52 m) and (x= 3.10 m, z= 4.52 m) respectively. The people in the test chamber were simulated by the human seated shape diminution in figure 2 and heated by 60 W /m$^2$/ person; depending on the human in offices building by ASHRAE standard 55-2004. The position of people in the test chamber (x= 1.5 m and 3.3 m) and (z= 1.03 m and 3.77 m). Human generated CO$_2$ through nostril and simulated by hole 12 mm diameter and height of 1.1 m above the floor [19], the calculation exhalation exit velocity can be calculated using the following equation 1,

$$V_{exit} = \frac{Q}{S} = \frac{Q}{((\pi D^2)/4)}$$

Where: Q is flow rate around 8.5 l/m, S = cross section area, D = nostril diameter in this study 12 mm [19]. The exhalation through the nose was recognized to be directed downwards from the horizontal level with an angle of approximately 45º. However, the flow rate (Q) at a sitting level is around 8.5 l/m then the exit exhalation velocity is 1.25 m/s.
A laptop shape 0.45 m X 0.45 m X 0.05 m, located in the table meeting in the middle of the room was simulated with three 60 W light bulbs. The laptops position 0.9 m above the floor and 0.05 m front of human, however, because the geometrical is quite complicated the legs of the table and chairs are not included that to decrease the computational grid [19]. 6 Fluorescent light with 0.07 m diameter and 0.12 m height installed in the ceiling and positioned at (x, z) (1.2 m, 1.2 m), (1.2 m, 2.4 m), (1.2 m, 3.75 m), (3.6 m, 1.25 m), (3.6 m, 2.40 m), (3.6 m, 3.75 m). The light simulated with 12 W light bulbs and the total 6*12 W.

The record of temperature, velocity and carbon dioxide concentration along five plumb lines in the chamber are measured point by point and line to line. Figure 3 shows the sensors line distribution in the plan view in room.
2.1.2. Meshing
The overall outer mesh was dominated by a tetrahedrons mesh. Furthermore, the sizing of the mesh was adjusted in certain locations in order to get smooth and high quality of mesh. The grid sizing was applied to the human, inlets and laptops. The number of grids was about 2.4 million. In addition, grid independence solution was checked by using a fine mesh compared with coarse mesh. It was achieved by doubling the number of grid i.e. a fine mesh (2.4 million) and coarse mesh (1.2 million). The residual converge criteria of energy is set to $10^{-6}$, while the turbulence kinetic energy, $k$ turbulent dissipation rate, $\varepsilon$, and carbon dioxide, $c$ were set to $10^{-3}$.

2.1.3. Modelling and setup
Many resent research have used (RNG) $k-\varepsilon$ because it can provide more accurate results for the turbulent airflow computation between eight different turbulence models that investigated by Chen [15]. In this study (RNG) $k-\varepsilon$ was chosen. In addition, another modelling setup energy equation that the air movement and detached mass was directed to momentum and energy equations using a finite volume method.

General turbulence equations are as the flowing:

Momentum equation:

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho uV) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$  \hspace{1cm} (2)

$$\frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho vV) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$  \hspace{1cm} (3)

$$\frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho wV) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z$$  \hspace{1cm} (4)

Energy equation:

$$\frac{\partial}{\partial t} \left[ \rho \left( e + \frac{V^2}{2} \right) \right] + \nabla \cdot [ \rho \left( e + \frac{V^2}{2} \right) V] = \rho \ddot{q} + \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right) - \frac{\partial (up)}{\partial x} - \frac{\partial (vp)}{\partial y} - \frac{\partial (wp)}{\partial z}$$

$$+ \frac{\partial (\tau_{xx})}{\partial x} + \frac{\partial (\tau_{yx})}{\partial y} + \frac{\partial (\tau_{zx})}{\partial z} + \frac{\partial (\tau_{yy})}{\partial y} + \frac{\partial (\tau_{zy})}{\partial z} + \frac{\partial (\tau_{zz})}{\partial z}$$

$$\rho f_V$$ \hspace{1cm} (5)

Where: $\rho = \text{air density}$, $V = \text{velocity}$, $f = \text{force}$, $P = \text{pressure}$, $\tau = \text{shear stress}$, $\dot{q} = \text{heat flux}$, $e = \text{internal energy}$.

3. Validation
The article published by Tian et al (2010) [7], has been chosen as the validation model of this study. Figure 4 showed the schematic diagram of Tian model, which has resulted from his experimental and simulation study. The reason for selecting this model from this article is because the model was about an office room which has a several heat sources and gasses contamination. In addition, this model has focused on CO$_2$ concentration and distribution in the room.

In this study re-built the published model in ANSYS 16.1, and run with a setting close to the published model. The details of geometry of this model show in the Figure 4, the room (3.9 x 2.9 x 2.6 m), the air inlet (0.21 x 0.17 m) rectangular located in the middle of the right wall, (0.6 x 0.6 m) perforated ceiling exhaust with 15.3% as the effective area ratio. The human body was simulated by box (0.25 x 0.4 x 1.2 m), a personal computer simulated by box (0.4 x 0.4 x 0.4 m), located on the table. The air inlet is 1.19 m/s and total heat load is 399 W; in the validation model, the results were illustrated in Figure 5, for nine positions. Therefore, the results from validation model were compared with Tian’s experimental and numerical results which show good agreement. The error ratio between validation line and measured line and between measured line and simulation line are 6.6% to 8.7% and 8.0% to 9.0% respectively. The simulation results are in acceptable agreement with experimental data.
Therefore, this re-built model from the distributed article can be applied to case study once modification in geometry and boundary condition that is described within the numerical method.

Figure 5 shows the numerical results compared with experimental and numerical simulation for [7] regarding of CO$_2$ concentration. It is found that the simulation results are in acceptable agreement with experimental data and simulation by Tian (2010).

4. Results and discussion

4.1. Grid independence test

The significant role in the simulation study is to check the mesh quality by grid independence test. This test is to confirm that the results are not sensitive with the mesh quantity. In another word, the solution is independent of the mesh. In this study, the process has been done by adjusting the mesh in which by doubling the number of grid mesh. However, the number of the grid was about 1.2 million with a coarse mesh and has been doubled to 2.4 million by applying a more fine mesh. Table 1 shows the results from a different number of elements.

| Type of mesh | Number of elements | Air velocity m/s | Air temperature °C | CO$_2$ concentration ppm |
|--------------|--------------------|------------------|--------------------|--------------------------|
| Mesh 1       | 1288922            | 0.196698         | 297.390            | 694.75                   |
| Mesh 2       | 2450288            | 0.210158         | 298.258            | 711.50                   |
| Percentage difference | | 6.40% | 0.29% | 2.30% |

From Table 1, the solver can simulate a relatively faster using coarse mesh. However, due to small variation obtained from both types of mesh, the Mesh 1 was chosen for the rest of the analysis in order to save time.

![Figure 4. Geometric model designs of chamber (validation).](image-url)
Figure 5. Measured, simulated (Tian et al. 2010) and simulation validation CO$_2$ concentration profile for various positions (ppm)

4.2. Air temperature distribution

Figure 6. Experimental and simulation temperature profile in the chamber
Figure 7. Air temperature distribution in horizontal plan

Figure 8. CO$_2$ concentration top plane (X Z); Y=1.1 m

Figure 6 compares the simulation findings with the experiment. It is observed that the air temperature decreases with height points. However, in line 2 the temperature had shown a little increase near to the ceiling. Some points in line 1 and 2 showed an agreement in the temperature degree between simulations and measurement. The cooler air entered to the room from two diffusers located in centre of the ceiling and the air then distributed to various types of heating sources such as humans, computers. Moreover, the lamps on the ceiling also contribute to the heating of the incoming air. The thermal plumb in line 1 of the simulation illustrated some similar degrees compared with the measurement.

The temperature from simulation was recorded in the range of 23.5 ºC to 24.0 ºC. Whereas, in Line 2 the results were indicated some agreement in the temperature degrees between measurement and simulation. Meanwhile, some temperature degrees show overestimation at level less than 1.1 m. In line 3, the higher degree was recorded at 0.5 m height and it was 26.9 ºC, compared with 24.5 ºC from measurement at the same level. This difference in temperature may be due into the error of measuring instrument as [20] Siamak R. Ardkapan et al. (2014) also found. In line 4, we can notice that the minimum degrees in level 1.1 m was 22.5 ºC from measurement graph, whereas the result of the simulation at the same level was 24.0 ºC. However, from the results of the simulation, air temperature profile indicated that the office area has a varying distribution of temperature. Generally the temperature near the floor was slightly higher. However, in this study, the effect of air diffusers in the centre line of the ceiling chamber also significantly influences the variation of the air temperature distribution.

The error of air temperature percentage between experimental and simulation is from 4.0% to 5.9%.

Figure 7 shows the air temperature distribution in horizontal plan at 0.8 m above the floor. The results indicate that the temperature at level 0.8 m was higher than other levels, especially around the heat sources.

4.3. Air velocity distribution

The air velocity profile from the simulation was obtained from the inlet velocity in experimental data. In addition, the air velocity inside the chamber was investigated at many locations and several heights. In this simulation, four positions and five heights were recorded line by line through shifting the sensors’ point.

Figure 9 demonstrates the results of measurement and simulation for vertical air velocity at four plumes. In line 1, the air velocity in this line was lower than the other line (approximately 0.07 m/s), except in the level near to the ceiling, it was recorded 0.17 m/s at 2.5 m height. That is due to the air obtained from ceiling air flow inlet. Generally, the velocity in the lower level was low than the highest points.
The maximum average of air velocity recorded in line 4 that is located between two diffusers is 0.120 m/s, while the maximum velocity from the experimental results is 0.136 m/s. However, the distribution of air velocity depended on the type of ventilation and location of air diffuser. Ceiling ventilation affected the air distribution at low level of heights; the expected results of air velocity profile had already been discussed in the air temperature profile section which caused in reduction of the temperature in these levels.

4.4. CO$_2$ concentration distribution

The CO$_2$ concentration resulted from the simulation and experimental is shown in Figure 10. The figure indicates that the variation between these findings from the different background of CO$_2$ concentration, which is applied during the experiment and simulation. The background of CO$_2$ concentration in the chamber was measured during the working day, with the course of 16 hours for all the tests, which makes the time duration of each test is 4 hours.

Figure 10 shows that the CO$_2$ concentration in line 1 recorded the highest number at 0.5 m and decreased in the higher levels. In this line, the simulation graph recorded less value than the experimental results except at level 0.5 m above the floor that may be to the different background of CO$_2$ concentration. This is because in the real scenario the background of CO$_2$ varies with time, however, this variation was constant during the simulation process. From the simulation results, a little change of values occurred on line 2 and 4 at each level. Figure 8 shows the CO$_2$ concentration top plan at 1.1 m above the floor. The concentration of CO$_2$ recorded higher values in the centre of the chamber where the breathing zones at the level of 1.1 m was close to the nose of occupants. Whereas, the air velocity near to the walls was quite low, this means less concentration comparing with the CO$_2$ concentration at breathing zone.

This caused by the location of sensor holder of line 2 was near to return air hole, while for line 4 was located between two inlet diffusers. According to [10] Fan & Ito (2014), in the ceiling air supply system the concentration of CO$_2$ in the return air zone is relatively lower than the occupied zone. From the current result of the simulation of CO$_2$ the error ratio was 3.87 to 9%, however, this result agrees well with the findings of the experiment.
5. Conclusion
The research was based on experimental study which had the support of a numerical analysis on the air flow type by using the CFD technique. The air velocity obtained from the experimental and CFD showed that the air diffuser in the centre line of the ceiling chamber significantly infuses the lowering values of air velocity at 0.5 m level. Thermal comfort in this study is within the ASHRAE stander 55-2004, local code and also accepted by occupants in the workplace. Controlling the CO\textsubscript{2} concentration depends on the ratio of fresh air divided by the total volume. This ratio can when increased will decrease the carbon dioxide concentration in work place, which causes the increase of energy consumption. The area around occupants is more polluted comparing with the upper ventilated zone. That is due the fresh air supplied through the ceiling openings was diffused in the upper zone but was not effectively transported to the occupied zone. Moreover, the assessment of CO\textsubscript{2} concentration demonstrates the maximum concentration at low level and around the human body, whereas, the concentration of the ceiling ventilation is very well, on one condition, when the air distributed is accurate.

The numerical analysis by CFD simulation in this study also showed the improvement IAQ of the current approach. The results from the numerical analysis showed good agreed with the experimental results. However, the advantage of the numerical simulation is the ability to depict the air velocity and air temperature distribution at any layers or locations in the place.

Acknowledgement
The authors would like to thank the staff of the Thermal Environmental Control laboratory in the Faculty of Mechanical & Manufacturing Engineering for their technical support and the Office for Research, Innovation, Commercialization and Consultancy Management (ORICC), Universiti Tun Hussein Onn Malaysia for financial support for this research (Grant U176).

References
[1] Brown K, Kalata W and Schick R 2014 Optimization of SO2 scrubber using CFD modeling Procedia Eng. 83 170–80
[2] Mahyuddin N and Awbi H 2010 The spatial distribution of carbon dioxide in an environmental test chamber Build Environ. 45(9) 1993–2001
[3] Bulińska A, Popiołek Z and Buliński Z 2014 Experimentally validated CFD analysis on sampling region determination of average indoor carbon dioxide concentration in occupied space Build Environ. 72 319–31
[4] Zhao B, Zhang Z, Li X and Huang D 2004 Comparison of diffusion characteristics of aerosol particles in different ventilated rooms by numerical method ASHRAE Trans. 110(1) 88–95
[5] Inthavong K, Tian Z F and Tu J Y 2009 Effect of ventilation design on removal of particles in woodturning workstations Build Environ. 44(1) 125–36
[6] Teodosiu C, Ilie V and Teodosiu R 2016 Numerical prediction of thermal comfort and condensation risk in a ventilated office, equipped with a cooling ceiling Energy Procedia 85 550–8
[7] Tian L, Lin Z and Wang Q 2010 Comparison of gaseous contaminant diffusion under stratum ventilation and under displacement ventilation Build Environ. 45(9) 2035–46
[8] Zhang Z and Chen Q 2006 Experimental measurements and numerical simulations of particle transport and distribution in ventilated rooms Atmos Environ. 40(18) 3396–408.
[9] Catalina T, Virgone J and Kuznik F 2009 Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling Build Environ. 44(8) 1740–50
[10] Fan Y and Ito K 2014 Integrated building energy computational fluid dynamics simulation for estimating the energy-saving effect of energy recovery ventilator with CO2 demand-controlled ventilation system in office space Indoor Built Environ. 23(6) 785–803
[11] ASHRAE 2005 Standard 55 Thermal Environmental Conditions for Human Occupancy
[12] Tian L, Lin Z, Liu J and Wang Q 2008 Numerical study of Indoor Air Quality and thermal comfort under stratum ventilation Prog Comput Fluid Dyn An Int J 8(7/8) 541
[13] Lin Z, Tian L, Yao T, Wang Q and Chow T T 2011 Experimental and numerical study of room airflow under stratum ventilation Build Environ. 46(1) 235–44
[14] Lin Z, Lee C K, Fong K F and Chow T T 2011 Comparison of annual energy performances with different ventilation methods for temperature and humidity control Energy Build. 43(12) 3599–608
[15] Chen D Q, Muralidhar K and Munshi P 1995 Comparison of different k-ε models for indoor air flow computations Numer Heat Transf Part B 35 353–69
[16] ASHRAE 2007 Standard. Ventilation for Acceptable IAQ ANSI/ASHRAE Stand 621-2007 62(1) 41
[17] DOSH 2010 Industry Code of Practice on Indoor Air Quality Minist Hum Resour Dep Occup Saf Heal 1–50
[18] Jamaludin N, Mohammed N I, Khamidi M F and Wahab S N A 2015 Thermal Comfort of Residential Building in Malaysia at Different Micro-climates Procedia - Soc Behav Sci 170 613–23
[19] Mahyuddin N, Awbi H B and Essah E 2014 A computational fluid dynamics modelling of the air movement in an environmental test chamber with a respiring manikin J Build Perform Simul 1493 1–16
[20] Ardkapan S R, Johnson M S, Yazdi S, Afshari A and Bergsøe N C 2014 Filtration efficiency of an electrostatic fibrous filter: Studying filtration dependency on ultrafine particle exposure and composition J Aerosol Sci 72 14–20