Pertanika J. Sci. & Technol. 25 (S): 77 - 88 (2017)

SCIENCE & TECHNOLOGY

Journal homepage: http://www.pertanika.upm.edu.my/

Computational Simulation of Indoor Thermal Environment in a Tropical Educational Hall with Displacement Ventilation

Qi Jie Kwong[1]*, Hon Fai Chen[2] and Azli Abd Razak[3]

[1]*Faculty of Architecture, Planning and Surveying, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Malaysia*
[2]*FOSTEM, Inti International University, Bandar Baru Nilai, 71800 Nilai, Negeri Sembilan, Malaysia*
[3]*Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Malaysia*

ABSTRACT

Displacement ventilation (DV) has been found to be effective in cooling large indoor spaces, but its effectiveness in providing thermal comfort in tropical buildings requires more detailed analysis to be made. This study examines the thermal environment in a lecture hall cooled via DV using FloEFD, a computational fluid dynamics (CFD) software. A calibrated CFD simulation model was developed to replicate the actual atmospheric conditions of the DV-cooled lecture hall. Results indicate that some parts of the hall received more cooling and air movement than required which would result in local thermal discomfort, especially at the front seating areas. The CFD results were consistent with those of earlier studies which validated the simulation model. The findings can be used to determine the locations which are more prone to steep thermal gradients in addition to reducing thermal discomfort, which include suitable arrangements of occupants in the lecture hall when it is not fully occupied and resetting the diffusers' supply air temperature.

Keywords: Computational fluid dynamics (CFD), Displacement ventilation (DV), lecture hall, thermal comfort, thermal environment

ARTICLE INFO

Article history:
Received: 05 January 2017
Accepted: 17 January 2017

E-mail addresses:
kwong.qijie@mail.com (Qi Jie Kwong),
henry_chf@hotmail.com (Hon Fai Chen),
azlirazak@salam.uitm.edu.my (Azli Abd Razak)
*Corresponding Author

ISSN: 0128-7680

INTRODUCTION

As people are spending most of their time indoors, the need for sustaining indoor thermal comfort becomes important. In typical university settings, lecture halls are generally occupied by students attending lectures or training sessions. Such places also depend solely on air conditioning and mechanical ventilation systems to maintain indoor thermal comfort as they are usually enclosed with fabric-covered sound absorbing wall and heavy building materials which limit the use of natural ventilation.

Displacement ventilation (DV) has been used in many large buildings, such as the auditoria, conference halls, lecture rooms and other spaces with high ceilings (Liu, Min, & Song, 2015). This type of ventilation system is based on the concept of thermal stratification by supplying cool air from the diffusers, and this dense cool air slowly rises after it absorbs heat generated by various heat sources (occupant, office equipment and artificial lighting). The heat is then discharged from the indoor space as it reaches the ceiling exhaust grilles. This method has been found to be very effective in maintaining occupant thermal comfort and reducing cooling costs compared with mixing air systems (Gilani, Montazeri, & Blocken, 2016). An earlier study also found that DV was effective in a maintaining favourable thermal comfort in a passive school's classroom (Wang, Kuckelkorn, Zhao, & Spliethoff, 2013). However, some studies found that this technology runs the risk of creating local thermal discomfort due to cold draughts at the floor level when the supply of air temperature is too low.

Using numerical method or computer software as a research tool to simulate indoor environments is increasingly popular among building experts and researchers worldwide, largely owing to the high accuracy of its results. Among the simulation software, Computational Fluid Dynamics (CFD) technique is one of the most commonly used programmes in the study of indoor thermal environments. This computational technique has the capability to predict the complex flow structure, and with very detailed simulation outcomes at every point of the flow domain. Many indoor environmental studies have employed CFD-based programmes in determining the local temperature and air velocity profiles, and the outcomes were used to estimate the thermal comfort conditions (Abou-deif, Fouad, & Khalil, 2013; Catalina, Virgone, & Kuznik, 2009; Webb, 2013). Previous works have compared the accuracies of CFD simulation with theoretical experiments and actual field measurements, and concluded that this powerful software could serve as an effective tool in estimating the actual thermal conditions of a particular building space (Cheng, Niu, & Gao, 2012; Hajdukiewicz, Geron, & Keane, 2013). Another advantage of applying CFD technique in experiments related to indoor environment is that the simulated outcomes are often used to complement the equipment-generated data (Wang, Zhao, Kuckelkorn, Liu, Liu, & Zhang, 2014; Wong & Mohd Rafique, 2010) as the latter is often restricted to the availability of measuring devices.

This paper reports on the simulation results of thermal environment in a large educational hall located in a tropical country using a commercially available CFD software –FloEFD 12.0, which incorporated Solidworks as the modelling module for the creation of a 3D simulation model. A comparison of the simulated and experimental results based on the recommended comfort ranges for air temperature and velocity is also presented which can provide useful information about the use of DV systems in Malaysian buildings.

MATERIALS AND METHOD

Pilot Survey

A pilot survey was conducted in a lecture hall installed with a DV system. The survey was carried out from November to December 2014. The lecture hall had three sections – middle, left and right seating areas, and an interior environment as shown in Figure 1. There were 215 seats: 12 rows at the centre, and 10 rows at both left and right seating areas. The thermal comfort parameters, such as air temperature, air velocity, relative humidity and room surface temperatures, were measured using calibrated electronic meters as shown in Figure 2 for both comparison and validation purposes. The meters were placed at the centre of the lecture hall and positioned at about 0.6 m above floor level as per the requirement of American Society of Heating, Refrigerating and Air-Conditioning Engineers, Standard 55 (ASHRAE, 2010) for evaluating the thermal environment of seated occupants. Written consent was acquired from the lecturers and the management of the educational institute. Measurements were taken around half an hour after the lecture sessions commenced to ensure that the indoor temperature and air velocity distribution were even.

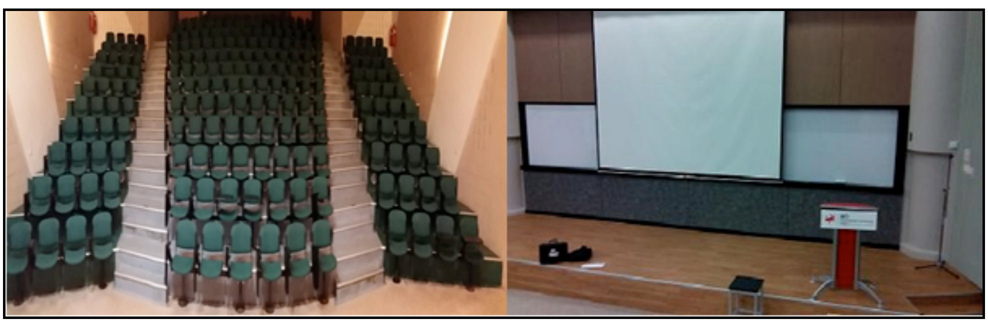


Figure 1. Internal environment of the lecture hall

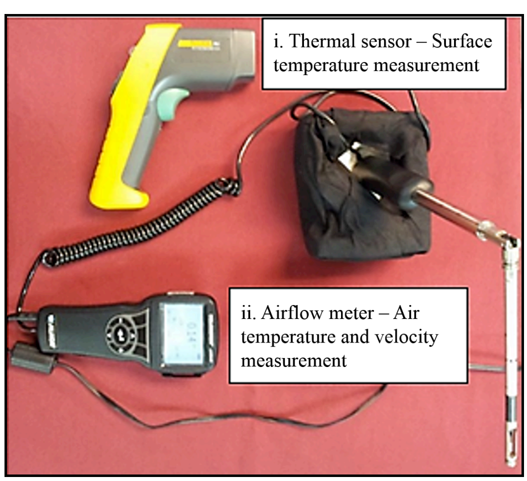


Figure 2. Equipment used for field measurement

Development of the Simulation Model

In order to understand the airflow pattern and temperature distribution in the DV-cooled lecture hall, a simulation model of the latter was developed according to the actual geometrical configuration using the FloEFD software, which is a CAD-integrated CFD tool. The FloEFD is a very effective tool for detailed modelling of indoor air conditions, which includes comfort parameters (Mentor Graphics, 2015). This CFD solver employs a modified k - ε two-equation turbulence model which uses the finite element method. The model was developed based on a partially occupied occupancy level, which is about 50% of the total available seats to replicate the actual situation of the observed lecture sessions. Other occupancy rates, both higher and lower than the prescribed occupancy value, seldom occurred as the lecture hall was normally reserved for combined lecture sessions, which consisted of 100-120 occupants.

The specification of boundary conditions is an integral part of any CFD problem as the results are directly dependent upon the input of boundary conditions. The locations where the simulated "fluid" enter and exit the lecture hall model were the air-conditioning inlet diffusers and exhaust grilles. The hall's surface temperatures, air velocity, temperature of the supply air diffusers and exhaust grilles were set according to field data obtained from the pilot survey, as presented in Table 1. Figure 3 shows the lecture hall with 107 occupants. Existing heat sources within the hall were modelled as heat fluxes to show the amount of heat generated and the heat dissipating rates were obtained from ASHRAE Standard 55 (2010) and Chartered Institution of Building Services Engineers Concise Handbook (2008). Occupants in the lecture hall were modelled using mannequins with heat flux values of 60W/m^2 for students (seated) and 70W/m^2 for the lecturer/trainer (standing), while each of the lighting fittings was represented with a heat dissipation rate of 12W/m^2. The temperature and velocity values obtained from the simulation were then compared with the results obtained from field measurements for validation purpose. Results of the stipulated threshold values and that of earlier works were compared.

Table 1
Input values for boundary conditions

Air-conditioning Equipment	Temperature (°C)	Velocity (m/s)	Pressure (atm)	Boundary type
Ceiling diffusers	18.4	4.98	1	Flow inlet
Stage wall diffusers	15.4	1.15	1	Flow inlet
Door wall diffusers	15.6	0.95	1	Flow inlet
Under seat diffusers	18.5	3.34	1	Flow inlet
Ceiling exhaust grilles	20.0	0.45	1	Flow outlet

Table 2
Input values for heat sources

Heat Source	Heat Dissipation Rate	Heat type
Seated mannequin	60 W/ m^2	Surface source
Artificial lighting	12 W/ m^2	Surface source
Standing mannequin	70 W/ m^2	Surface source

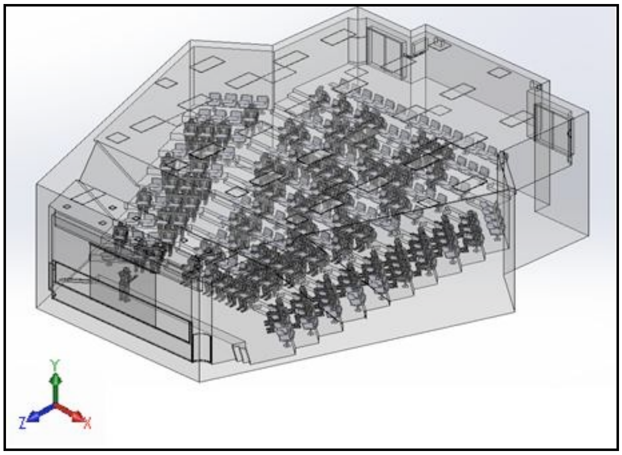


Figure 3. CFD simulation model of the lecture hall

RESULTS AND DISCUSSION

Field Measurements

During the pilot survey, the indoor thermal parameters which included supply air temperature at the diffusers, room surface temperatures (wall, door, floor and ceiling) as well as air velocities at the supply diffusers and exhaust grilles were recorded (as this information was essential for the simulation of indoor environment and for validation purpose). The room surface temperatures, as shown in Table 3, were used to calculate the mean radiant temperature (MRT). The thermal comfort parameters are shown in Table 4. The air temperature ranged from 20.9 to 21.5°C, while the air velocity was found within the range of 0.05 to 0.26 m/s. These measured values are generally not within the recommended comfort ranges stipulated in the Malaysian Standard (MS) 1525 (2014), which is a local energy guideline for non-residential buildings. This suggests that some of the occupants may find the thermal environment unacceptable. An operative temperature of 21.98°C was calculated, and it should be noted that only the thermal environmental parameters at the centre of the lecture hall, which was between the sixth and seventh row along the middle seating area, was measured.

Table 3
Lecture hall's surface temperatures

Location/ Surface Temperature	Min (°C)	Max (°C)	Mean (°C)
Ceiling (Up), T_1	21	22.6	21.97
Floor (Down), T_2	22.1	23.9	23.42
Wall (Left), T_3	22.1	23.7	22.75
Wall (Right), T_4	22.4	23.9	23.46
Whiteboard (Front), T_5	22	23.4	22.47
Door (Back), T_6	22.1	24.2	23.35

Table 4
Thermal environmental parameters

Parameter	Min (°C)	Max (°C)	Mean (°C)
Air Temperature, T_a(°C)	20.9	21.5	21.14
Air Velocity, v(m/s)	0.05	0.26	0.13
Humidity, RH (%)	68.8	72.6	70.78
Mean Radiant Temperature, T_{MRT}(°C)	21.79	23.45	22.81
Operative Temperature, T_{op}(°C)	-	-	21.98

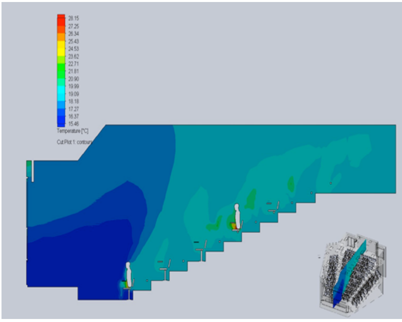
Simulation Model

A model was developed to simulate the airflow pattern and temperature distribution within a partially occupied lecture theatre with about 50% occupancy rate. It was assumed that the left and right seating areas had the same distribution of occupants.

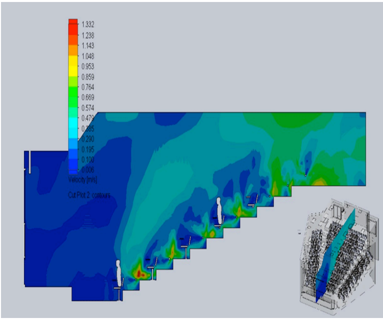
Since the field data collection was conducted at the centre of the lecture theatre, the coordinates of this location in the CFD model were identified at X – 8.05, Y – 4.8, Z – 8.8. Thus, the temperature and velocity profiles of this point were simulated and analysed and presented in Figure 4 to 6. Areas with cooler temperature are depicted by blue and cyan colours, while warmer temperatures are shaded in orange and red. It can be observed that the predicted temperature at the selected point of measurement ranged from 19.59 to 22.04°C, and air velocity varied between 0.05 and 0.22 m/s, which were relatively lower than the findings of Wang et al. (2013) and Liu et al. (2015). This may be due to difference in thermal boundary condition settings as well as the types of simulation selected. Besides, the mean simulated air temperature and velocity were calculated by averaging the point temperature taken along plane Y = 4.8m which corresponded with 0.6 m above the floor level, which is the stipulated measuring location for seated occupants (Table 5).

Table 5
Simulated range of temperature and velocity at measuring point

Temperature (°C)			Velocity (m/s)		
Min	Max	Mean	Min	Max	Mean
19.59	22.04	20.37	0.055	0.216	0.140

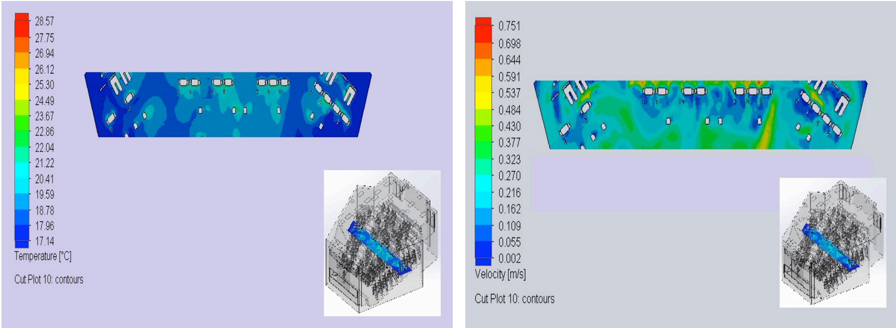


(a)



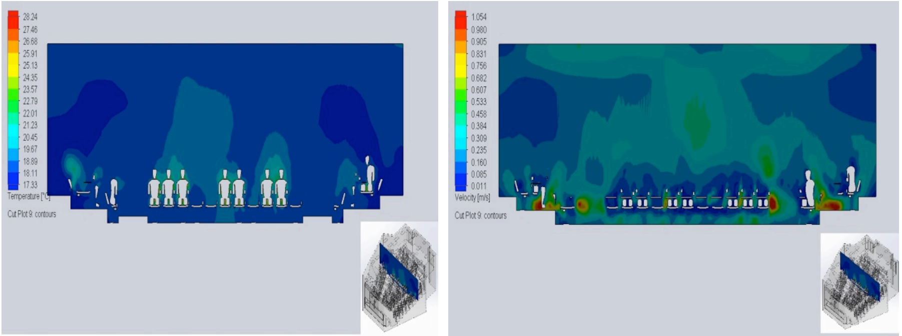
(b)

Figure 4. Simulated temperature and air velocity profile at X = 8.05 m. (a) Cut plot of temperature contour; and (b) Cut plot of air velocity contour



(a) (b)

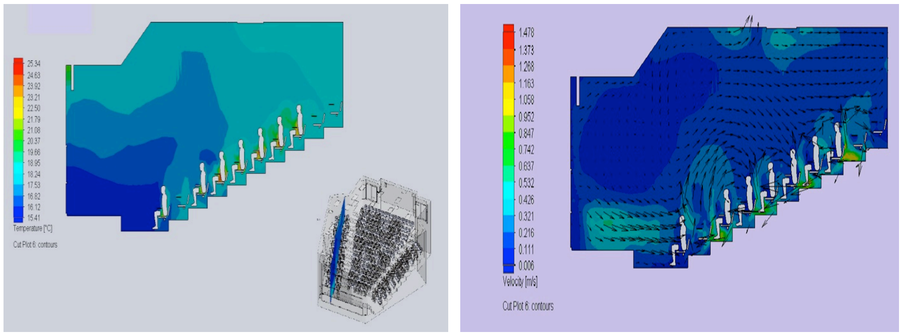
Figure 5. Simulated temperature and air velocity profile at Y = 4.80 m. (a) Cut plot of temperature contour; and (b) Cut plot of air velocity contour



(a) (b)

Figure 6. Simulated temperature and air velocity profile at Z= 8.80 m. (a) Cut plot of temperature contour; and (b) Cut plot f air velocity contour

Cool air was supplied into the indoor space through a combination of wall diffusers, ceiling diffusers and under seat diffusers. The supply and return air conditions for each of the diffuser was modelled according to the actual environment in the lecture hall. The contour plots of the air temperature as shown in Figure 4(a) and Figure7(a) demonstrates that the students seated at the first row would experience cooler air temperatures due to the location being nearer to the wall diffusers on stage, which supplied conditioned air at a temperature of about 15°C. The same condition prevailed at the seating areas near the side doors that had wall diffusers installed beside the doors. Such condition was also reported in an earlier work, where the occupants seated adjacent to the diffusers were more likely to be thermally uncomfortable due to the steep thermal gradients (Wang et al., 2013).



(a) (b)

Figure 7. Simulated temperature and air velocity profiles at the left seating area. (a) Cut plot of temperature contour; and (b) Cut plot of air velocity contour

Other than the selected measuring point, the local air temperature and velocity at the area with the highest occupant density, which was at the left area of the lecture hall, were also analysed. Based on the plotted temperature gradients in Figure 7(a), it can be observed that the use of under seat diffusers may cause local thermal discomfort to some of the occupants in this area. The predicted temperature around the human model varied from the head to foot-leg regions. The air temperature rises as the air moves upwards, which is the heat rejection method of the DV system. Lower temperature regions were found at around the foot-leg region of the mannequins with a mean predicted temperature of 19°C, whereas temperature surrounding the body (chest and back regions) of the human model was about 23°C. It was also found that a higher temperature existed between the thigh and knee area of the mannequins. Such differences in temperature surrounding the mannequins suggest that local discomfort will most likely occur, since the vertical temperature difference was more than the permitted 3°C (ASHRAE, 2010). This suggest that the air temperature should be increased by 1 to 2°C to sustain thermal comfort. An interesting finding in the simulation is that the predicted temperature around the lighting is only around 17 to 18°C. This shows that the heat produced by the light fittings does not have any significant effect on the thermal surroundings, as compact fluorescent lamps (CFL) do not generate as much heat as the incandescent lamps.

Figure 7(b) shows the air velocity profile of the left seating area. The contour plot indicates the presence of high air movements (0.30– 0.95 m/s) around the foot-leg region of mannequins, especially those positioned at the odd number rows because of the underfloor diffusers. If only the perceived air velocity at the core of the mannequins was considered, no sign of draught was found in this seating area as the air velocity was only within the range of 0.11 – 0.32 m/s. This outcome demonstrates that the air flow in the lecture hall was carefully controlled to minimise the possible occurrence of draught, which is among the factors that cause thermal discomfort.

Validation of CFD Model with Empirical Results

The simulation outcomes have to be validated to strengthen its reliability. From Figure 4, the predicted air temperatures at the measuring spot during the field survey were within the range of 19 to 22°C with a mean value of 20.37°C. On the other hand, the mean air temperature measured in the lecture hall was 21.14°C, which indicates a difference of 0.7°C between the measured and simulated outcomes. This shows that the predicted temperature was slightly underestimated. As for the air velocity profile, the predicted outcomes at the location of Y = 4.8m were within the range of 0.05 to 0.22 m/s with an average value of 0.14 m/s, while the measured mean air velocity was 0.13 m/s. Hence a slight difference of 0.01 m/s was found between the simulated and empirical values.

Table 6
Percentage difference between empirical and simulated results

Thermal Parameter	Empirical	Simulated	Differences (%)
Air Temperature (°C)	21.14	20.37	3.5
Air Velocity (m/s)	0.13	0.14	7.7

There are several reasons for this slight variation in results. First, the CFD model was simulated under a steady state condition in this work, which did not fully demonstrate the real-life fluid and heat transfer processes that are mostly transient in nature. Additionally, the fluctuations in the air and surface temperatures with time may have affected the thermal conditions, even though efforts were made to collect field data only after the lecture sessions commenced for half an hour to minimise the potential variations of the thermal parameters. Another factor that led to this discrepancy was the inlet flow patterns, where air flows were set to discharge at an angle of 45° from the diffuser surfaces. The surface temperatures of the wall, ceiling and floor of the lecture hall were fixed at a constant value at which only the selected part of the hall's indoor surface was measured to represent the whole section. These assumptions and simplifications may have influenced the accuracy of the predicted results. Nevertheless, it can be concluded that the prediction showed good correlation with the empirical ones, since the percentage differences between the two results were lower than 10%.

CONCLUSION

Simulations of indoor thermal environments using CFD could provide very convincing and reliable results. A calibrated CFD model of a lecture hall cooled via DV was developed in this work, and the indoor thermal parameters were simulated, analysed and compared with the field survey results. Overall, the simulation outcomes showed that the air temperature and velocity profiles were not within the recommended ranges stipulated in the MS 1525(2014), which was in line with the field survey data. The simulated results depicted good distributions of air temperature and velocity within the seated areas of the partially occupied lecture hall. Albeit minor discrepancies, a good correlation between the simulated and empirical results were obtained in this work, which indicates the applicability of this CFD tool in assessing the thermal environment of DV-cooled spaces. Other than the selected measuring point, the left seating area, which was with the highest occupant density, the simulation results revealed that some occupants seated in that area are expected to experience some level of local thermal discomfort due to the steep temperature gradient between the head and foot-leg regions. Therefore, an adjustment of supply of air temperature and relocating the occupants to other seating areas with lower vertical temperature differences should be considered.

The results showed that the DV system is an effective air conditioning technology to be used in large room areas in the tropics, only if the system is carefully controlled to prevent any form of local thermal discomfort, especially at locations where the air diffusers are installed. Since the occupant thermal adaptation was not evaluated in this work, future studies on evaluating the thermal experience of the occupants, which include psychological adaptation and expectation, are suggested. Other than that, the same model can be used to simulate other occupancy levels, especially the thermal comfort conditions of occupants when the lecture hall is fully occupied.

ACKNOWLEDGEMENTS

This work was funded by the Fundamental Research Grant Scheme (Grant No: 600-RMI/FRGS 5/3(73/2013)). The authors acknowledge the support of the technical staff of the Inti International University, Malaysia in carrying out the CFD simulations and to Mr. Hisham from Universiti Putra Malaysia for providing (at no charge) the indoor air quality meters for field measurements.

REFERENCES

Abou-deif, T. M., Fouad, M. A., & Khalil, E. E. (2013). Numerical investigation of flow patterns and thermal comfort in air-conditioned lecture rooms. *International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering, 7*(5), 917–922.

American Society of Heating, Refrigerating and Air-Conditioning Engineers. (2010). Standard 55: Thermal environmental conditions for human occupancy, ANSI/ASHRAE. ISSN 1041 – 2336.

Catalina, T., Virgone, J., & Kuznik. F.(2009). Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling. *Building and Environment, 44*(8), 1740–1750. doi:10.1016/j.buildenv.2008.11.015.

Chartered Institution of Building Services Engineers (CIBSE). (2008). *CIBSE Concise Handbook*. The Chartered Institution of Building Services Engineers London. ISBN 978 – 1 – 903287 – 94 – 1.

Cheng, Y., Niu, J., & Gao, N. (2012). Stratified air distribution systems in a large lecture theatre : A numerical method to optimize thermal comfort and maximize energy saving. *Energy and Buildings, 55*, 515-525. doi:10.1016/j.enbuild.2012.09.021.

Gilani, S., Montazeri, H., & Blocken, B.(2016). CFD simulation of stratified indoor environment in displacement ventilation: Validation and sensitivity analysis. *Building and Environment, 95*, 299–313. doi:10.1016/j.buildenv.2015.09.010.

Hajdukiewicz, M., Geron, M., & Keane, M. M. (2013). Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room. *Building and Environment, 70*, 73–89. doi: 10.1016/j.buildenv.2013.08.020

Liu, B., Min, K., & Song, J. (2015). Effect of thermal plume on personal thermal comfort in displacement ventilation at one side of the room. *Procedia Engineering, 121*, 1058–1066. doi: 10.1016/j.proeng.2015.09.103

Malaysian Standard (MS) 1525. (2014). Code of practice on energy efficiency and use of renewable energy for non-residential buildings. *Malaysian Standard*.

Mentor Graphics. (2015). FloEFD Products [Online]. Retrieved from http://www.mentor.com/products/mechanical/products/floefd/

Wang, Y., Kuckelkorn, J., Zhao, F. Y., & Spliethoff, H. (2013). Indoor environment of a classroom in a passive school building with displacement ventilation. *Proceedings of BS2013: 13th Conference of International Building Performance Simulation Association, Chambery, France, August 26 - 28, 2013, 1902 - 1909*.

Wang, Y., Zhao, F., Kuckelkorn, J., Liu, D., Liu, J., & Zhang, J.(2014). Classroom energy efficiency and air environment with displacement natural ventilation in a passive public school building. *Energy and Buildings, 70*, 258–270. doi:10.1016/j.enbuild.2013.11.071.

Webb, M. (2013). Building energy and CFD simulation to verify thermal comfort in under floor air distribution (UFAD) design, *Proceedings of BS2013: 13th Conference of International Building Performance Simulation Association, Chambery, France, August 26 - 28, 2013*. 1886– 1893.

Wong, S. J., & Mohd Rafique, S. M. S. (2010). Thermal modelling in product design using FloEFDTMPro: From concept to reality. *WCE 2010 - World Congress on Engineering 2010, 2*, 1263– 1267.