Proceedings of ES2008
Energy Sustainability 2008
August 10-14, 2008, Jacksonville, Florida, USA

ES2008-54232

COMPUTATION OF AIR FLOW IN CMU'S INTELLIGENT WORKPLACE AND ITS EFFECT ON OCCUPANT HEALTH AND COMFORT

Rong Li, Yongjie Zhang, David Archer
Department of Mechanical Engineering, Carnegie Mellon University, Pittsburgh, PA 15213
{rongl, jessicaz, archerdh}@andrew.cmu.edu

ABSTRACT

In this paper, computational fluid dynamics (CFD) is utilized to investigate thermal comfort and energy efficiency of an office in the Intelligent Workplace (IW) of Carnegie Mellon University. First, a comprehensive geometric grid model is constructed to represent the office including the walls, floor, roof, windows, doors, fan coil units, and furnishings. Then the air flow and accompanying heat exchange with the bounding surfaces of the office are calculated based on indoor and outdoor ambient conditions, the operating conditions of the fan coil units and windows, and the occupancy of the space. The computational results of the air flow provide the means to measure whether comfort conditions have been established based on the outside conditions and on the operation of the fan coil units. The operating conditions of the fan coil units, fan speed, and chilled/heated water flow, determine their effectiveness in cooling/heating and their operating costs. This CFD air flow simulation, therefore, when generalized, provides a design support system for architects and engineers. The numerical solutions of office space are calibrated and validated by empirical operational data from the fan coil units. The design support system will enable the evaluation of the annual performance and operating cost of given fan coil units in a given building space.

Keywords: Computational fluid dynamics, Air ventilation, Thermal comfort.

NOMENCLATURE

ρ	Density of the air
u_i	Air velocity along the i^{th} direction
μ	Viscosity of the fluid
μ_t	Turbulent viscosity
σ_k	Turbulent Prandtl numbers for k
σ_ε	Turbulent Prandtl numbers for ε
G_k	Generation of turbulent kinetic energy due to the mean velocity gradients
G_b	Generation of turbulent kinetic energy due to the buoyancy
ε	Dissipation rate of turbulent kinetic energy
$C_{1\varepsilon}$	Model constants
$C_{2\varepsilon}$	Model constants
$C_{3\varepsilon}$	Model constants
v	Component of the flow velocity parallel to the gravitational vector
u	Component of the flow velocity perpendicular to the gravitational vector

1 INTRODUCTION

Development and integration of environmental, energy and ecological issues into building design are becoming more and more important in the US. About 40% of both primary energy consumption and greenhouse gas emissions in the US are attributable to cooling, heating, lighting, and ventilating buildings. Unlike a traditional workplace, the Robert L. Preger Intelligent Workplace, the IW, at Carnegie Mellon University is a collaborative effort among industry, government and the university. Opened in December 1997, the IW demonstrates the economic feasibility to: (1) improve user satisfaction, (2) provide unprecedented levels of organizational flexibility, (3) provide unprecedented levels of technological adaptability, and (4) maximize energy and environmental effectiveness. The mission of the IW is to research, develop, and demonstrate advanced building operating systems and their integration for total building performance.

For a long time, heating, ventilating and air conditioning (HVAC) engineers and researchers have been working to optimize the thermal comfort in the working and living spaces. A large amount of experimental and computational work has been carried out in recent years. Efficient quantitative models which could well predict the indoor air qualities are highly desirable. This can be accomplished via numerical approaches. In order to study the relationship between the response and independent design and operating variables, a large number of experiments are undoubtedly inevitable. This will definitely increase the total cost of study, which is particularly true in the case of physical experiments. Therefore, numerical calculations accomplished by CFD have become increasingly important in previous decades.

In recent years, computational fluid dynamics (CFD) has been widely adopted to study the airflow and indoor environment in the building despite differences between the real world and simulation. It works as a practical design tool for HVAC engineers in today's society. Some CFD researches on air ventilation and thermal comfort have also been conducted. The use of computational fluid dynamics in building design not only improves the comfort and health of the occupants, but also reduces the consumption of non-renewable fossil energy, which leads to the atmospheric pollution and climate changes. Therefore, in order to minimize the energy usage while making sure good thermal comfort condition to be achieved, effective distribution of air flow within a workplace is of practical importance.

This paper presents a numerical study of the ventilation performance of fan coil units using computational fluid dynamics. Three-dimensional CFD models have been created to simulate the working condition of the two offices in the IW. In order to understand the energy consumption and indoor air environment in the IW offices, it is necessary to explore the characteristics of the indoor air flow. In this study, we will simulate air flow and heat transfer inside the IW, analyze their effect on occupant health and comfort, and then utilize the computational results to optimize the design, operation and location of cooling/heating units and operable windows in the IW, considering both personal comfort and energy consumption. A CFD model of an office space was built. Room geometry, boundary conditions, air flow rates, and heat transfer coefficients were provided by the IW engineers to simulate an actual test office used in our project.

2 SUMMARY OF PREVIOUS WORK

In recent years, computational fluid dynamics (CFD) has been used for the prediction of air quality in the buildings. Research on the dispersion of contaminants in an office environment using empirical and modeling techniques was conducted to ensure the health of occupants [1]. Thermal comfort parameters were measured at predetermined grid points within a virtual plane to confirm the airflow pattern of air jet, as well as to determine the occurrence of thermal stratification in the office space [2]. Ooi et al. adopted the CFD method to calculate the distribution of various parameters to determine the best placement location for air conditioner diffuser and also the area most suitable for the occupant [3]. Meanwhile, the best placement of a window-type air conditioner in a residential bedroom was determined by using the CFD software FLOVENT [4]. In addition, a study was carried out to compare the measured and simulated concentration of contaminant in the chamber for both the layouts by using a gas analyzer and a CFD program with the Re-Normalization Group k-ε model, respectively [1]. Their results showed that the simulation results and the measured results were in reasonable correlation with percentage differences ranged between -4.4% and -16.5%. The difference could be introduced by the measurement error and system difference between CFD simulations and experiments, which were not addressed in the paper.

Heating, ventilating and air conditioning (HVAC) have been analyzed by Mathews et al. [5] for the use of human science building (HSB) at the University of Pretoria to achieve a saving of 60% in power consumption. CFD software was utilized to simulate offices with four different heating systems. Results from this study give some new guidelines in the design of heat units, and contribute to the discussion on thermal climate and energy consumption with different heating and ventilation systems [6]. Using a building energy simulation code ACCURACY, Zhang et al. [7] investigated indoor humidity associated with cooling in hot and humid climate. Their results indicate that compared to an air based system, a chilled ceiling saves much fan energy due to reduced air circulation, and saves much chiller energy due to the increases temperature of the cooling surfaces. Similarly, Chen and Kooi's research [8, 9] showed how comfort and energy consumption in buildings could be evaluated by using computer programs for air flow analysis and air-conditioning load calculations.

To study the various equipment design and operating parameters and the performance of an HVAC system, a great number of experiments are needed, which will definitely take large labor-effort and time. Therefore, experimental approaches are not practical as general design tools. Adopting CFD probably will not replace physical experiments completely but it can significantly reduce the experimental workload [10]. Despite the fact that it is not totally free from errors, CFD methods give virtual distribution of airflow, temperature, and air quality in the entire domain which is difficult to get from experiments because of time and cost involved [11]. Consequently, CFD is used to evaluate alternative designs before experimental testing takes place. For example, Haghighat et al. [12] investigated the relationship between the contaminant concentration level in a partitioned room and various positions of door, supply and exhaust. Guohui Gan [13] adopted CFD software to evaluate various room air distribution systems. In his research, he found that some methods of air distribution are effective for the removal of indoor contaminants but may not be so for thermal comfort or energy utilization and vice versa. He also pointed out that an effective air distribution system should be able to achieve thermal comfort and good air quality with minimum energy consumption.

In some cases, the simulation of air flow in buildings can be a formidable task. The degree of difficulty is mainly dependent upon the dimensions, objects, boundary conditions, physical processes of the program. In order to help engineers correctly and effectively perform indoor environmental modeling using CFD, some researchers have set up examples to explain how to verify, validate, and report CFD results. Those efforts help enhance CFD as a reliable tool for the evaluation of indoor comfort. For instance, Chen et al. investigated the necessary steps to improve the CFD analyses [14]. The necessary requirements of several computational aspects that are needed for the accurate CFD simulation were also advanced by Chris et al. [15]. By using CFD, Mak et al. [16] presented a numerical study of the ventilation performance of wing walls and verified the experiments of Givoni [17, 18].

A CFD program [19, 20] has been developed in which the radiation heat exchange was taken into consideration. This program has been used to predict the air movement and thermal comfort in both mechanically and naturally ventilated buildings. Likewise, a new zero-equation model [21] was also utilized to simulate three-dimensional distributions of air velocity, temperature and contaminant concentrations in rooms. This method assumes turbulent velocity to be a function of length scale and local mean velocity. A new model was built up to predict natural convection, forced convection, mixed convection and displacement ventilation in a room. The results agreed well with experimental data and the results obtained by the standard k-ε model. In addition, the new model uses much less computer memory and the computing speed is at least 10 times faster, compared with the k-ε model.

3 COMPUTATIONAL METHODS

In this project, we simulate air flow and heat transfer inside the IW, analyze its effect on occupant health and comfort, and then utilize the computational results to optimize the design and operation of heating units with consideration of both comfort provision and energy consumption. A commercial software package, Airpak [22], has been used to model air flow, heat transfer and thermal comfort in the IW. Airpak provides an interface to construct the geometry of the IW, including all bounding surfaces, windows, doors, fan coil units, and furnishings. Some model components can be adjusted according to different outdoor conditions, for example, the air flow velocity and temperature of fan coil units can be varied.

3.1 Governing Equations

The prediction of air flow is based on a solution of the fundamental flow equations. They consist of the conventional continuity equation, the momentum equation, and the energy equation. The air is assumed to be incompressible and its physical properties are assumed constant. The residuals of continuity and momentum have to reach the magnitude of 10^{-3} for convergence, while the residual of energy has to reach 10^{-6} in magnitude. The properties of the workplace are characterized by the Navier-Stokes equations for the transport of mass, momentum, and energy when one calculates the laminar flow with heat transfer. Additional transport equations are solved when heat transfer is included.

3.2 Turbulence

Six turbulence models are available in Airpak. In this study, we compare several of them and finally select the two-equation (standard k-ε) turbulence model due to its accuracy. It is a semi-empirical model based on transport equations for the turbulent kinetic energy k and its dissipation rate ε. The transport equation for k is derived from the exact equation, while the value of ε is obtained by using physical reasoning. It uses the following equations to calculate the turbulent kinetic energy, k, and its rate of dissipation, ε:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_i}]+G_k+G_b-\rho\varepsilon, \quad (1)$$

$$\begin{aligned}\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = {} & \frac{\partial}{\partial x_i}[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial \varepsilon}{\partial x_i}]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+G_{3\varepsilon}G_b) \\ & - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}. \quad (2)\end{aligned}$$

In Equations (1-2), G_k represents the generation of turbulent kinetic energy due to the mean velocity gradients, and G_b is the generation of turbulent kinetic energy due to the buoyancy. $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. The turbulent viscosity, μ_t, is calculated by the following equation:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon}. \quad (3)$$

The model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ , σ_k and σ_ε have the following default values: $C_{1\varepsilon}$ =1.44, $C_{2\varepsilon}$ =1.92, C_μ =0.09, σ_k =1.0, and σ_ε =1.3. These default values have been found to work fairly well for fundamental turbulent flows.

The degree to which ε is affected by the buoyancy is determined by the constant $C_{3\varepsilon}$. In Airpak, $C_{3\varepsilon}$ is not specified, but is instead calculated according to the following relation:

$$C_{3\varepsilon}=tanh|\frac{v}{u}|, \quad (4)$$

where v is the component of the flow velocity parallel to the gravitational vector and u is the component of the flow velocity perpendicular to the gravitational vector.

3.3 System Description

A rectangular IW office space is shown in Figure 1. The walls of the room are relatively open, allowing air exchange between adjacent spaces. The size of the room is 4.6m(L)×3.9m(W)×5.7m(H). The temperature is controlled by fan speed and chilled/heated water flow in the fan coil units. The room is occupied by two staff members and their computers. They are located in the middle of the left and right walls. Two types of fan coil units (VKD and VKB), provided by a German company (LTG), are installed separately in two systems. The two fan coil units have different structure and generates different air velocity. The air velocity of VKD fan coil unit ranges from 0 to 0.22m/s while the air velocity of VKB fan coil unit ranges from 0 to 4.5m/s.

As shown in Figure 2, the room of Case 1 is heated in winter by warm air entering from three air supply diffusers. The 1200mm×300mm rectangular floor level air diffusers are located just below the windows in the facade. One VKD unit connected with three supply diffusers is installed under the floor. Return air for the VKD fan coil unit enters through a fresh air return with the diameter of 380mm, which is located parallel to the front door of the room. As shown in Figure 3, three VKB fan coil units in Case 2 are connected with three 624mm×110mm outlet diffusers which are located adjacent to the window. Warm air is provided by air diffusers. Return air from the room is taken by the VKB units at the unit itself through the same floor level diffuser immediately adjacent to the facade. The dimensions are 624mm×211.5mm.

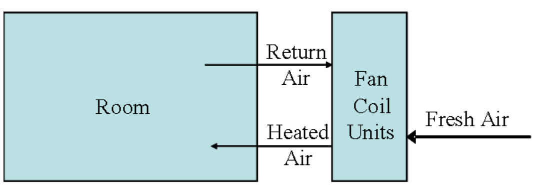


Figure 1. A system with a flat roof which transfers heat between the space inside and outside the room.

3.4 Boundary Conditions

The adiabatic thermal boundary condition is set at both the left and right walls. The heat transfer coefficient is 20.833 W/K-m^2 of the roof, 24.67 W/K-m^2 of the back wall and the windows. The properties of roof, back wall (the facade), and windows are shown in Tables 1, 2 and 3, respectively. The front wall is symmetric and adiabatic during our simulation. The outside temperature of the room is set as 6.8°C, as measured. The supply air diffusers are defined as a mass flow inlet boundary. The mass flow input value is obtained from supply air velocity and temperature measurement during the experiment. The constant heat flux thermal boundary condition is imposed at the surface of each person, who generates 75W sensible heat, and each computer, which generates 100W sensible heat.

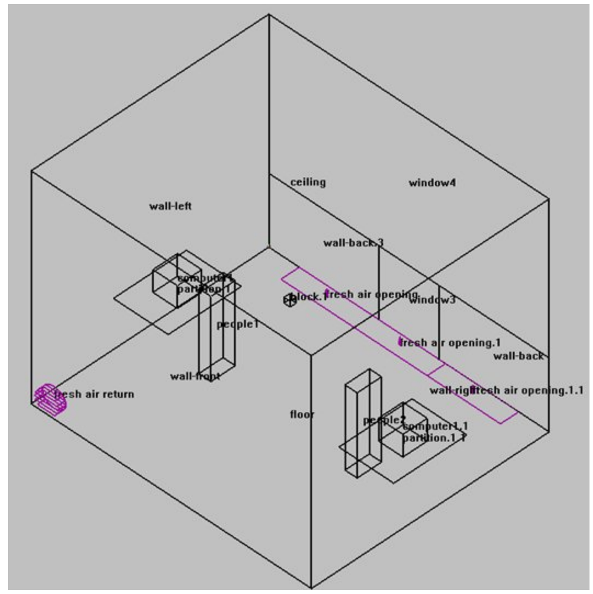


Figure 2. Case 1: the room with a VKD fan coil unit.

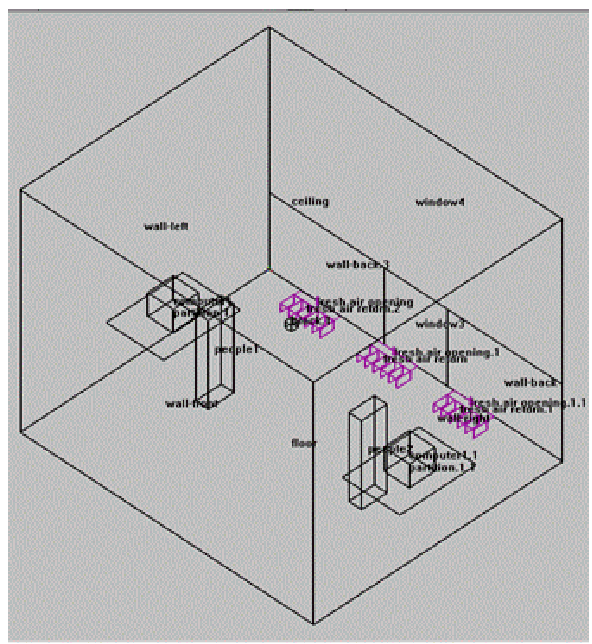


Figure 3. Case 2: the room with VKB fan coil units.

For Case 1, the supply air flows at 35°C from the air diffusers along the facade of the IW and vertically upward, and mixes with the cold air in front of the window. The relative humidity of the air is 12.64RH. The air coming from diffusers flows throughout the room, and the same amount of air is removed from the fresh air return. Similar boundary conditions are set in Case 2. The supply air diffusers discharge heated air at

Table 1. The properties of the ceiling.

Roof (Unit)	External Material	Middle Material	Internal Material
Conductivity (kJ/(hmK))	160.59	0.162	0.588
Capacity (kJ/(kgK))	0.502	0.8368	1.4644
Density (kg/m^3)	7833	8	1121

Table 2. The properties of the back wall.

Back Wall (Unit)	External Material	Middle Material	Internal Material
Conductivity (kJ/(hmK))	25.98	0.162	25.98
Capacity (kJ/(kgK))	0.4191	0.8368	0.4191
Density (kg/m^3)	7869	8	7869

Table 3. The properties of the windows.

Parameter	Value
Roughness (microns)	0.00035
Emissivity	0.94
Solar absorptance	0.08
Solar transmittance	0.1
Hemispherical diffuse absorptance	0.08
Hemispherical diffuse transmittance	0.1

32°C, with the relative humidity of 4.6RH. The upcoming warm air travels along the ceiling, and then returns down the facade to the inlet diffusers. In this study, we need to set the air velocity at diffusers as the boundary condition of the model. An anemometer is used to measure the discharge air velocity. The output air from diffusers is 0.22m/s in Case 1, and 4.5m/s in Case 2.

3.5 Mesh Generation

After building up the geometry of the models, we need to generate the computational mesh. On one hand, if the mesh is too coarse, the numerical solutions may not be precise enough to give enough suggestions to the future design. On the other hand, the computational cost may become prohibitive if the overall mesh is too fine. In summary, the computational expense and accuracy of the results are directly dependent on the size of the mesh. There are three types of meshes available in Airpak: hexahedral, tetrahedral and hexdominant meshes. The hexahedral unstructured mesh is appropriate for most applications. For complicated models, tetrahedral mesh is the best choice. The hex-dominant mesh can be used for automatic meshing of CAD geometries. Based on the computational cost concern, we select the hexahedral mesh for our model.

4 Results and Discussion

4.1 Temperature

Figure 4 and Figure 5 show temperature distributions in Cases 1 and 2, respectively. From these plots, we observe that the temperature values of occupant surroundings fall within the temperature range of 30-31°C in Case 1, and 28-30°C in Case 2. ENV Guidelines [23] recommend the temperature range from 22.5°C to 25.5°C as an acceptable comfort temperature. Based on the standards, we can conclude that the average room temperature for Case 1 and Case 2 is warm for the occupants. In this study, the vertical temperature distribution is also examined. It is found that the large temperature gradient usually occurs near the supply diffusers and ventilation, as shown in Figure 4a and Figure 5a. Figure 4b and Figure 5b also show that the temperature differences in the vertical direction at the occupant locations appear to be less than 3°C on average.

ISO Standard 7730 [24] specifies that the vertical temperature difference for all locations should be within 3°C between the levels at 0.1m and 1.1m. According to the standard, temperature plots show that the vertical temperature difference for all the locations in the two models satisfies the requirement of the thermal comfort. However from Figure 4b and Figure 5b, we can conclude that the vertical temperature in Case 1 is distributed more evenly than that of Case 2.

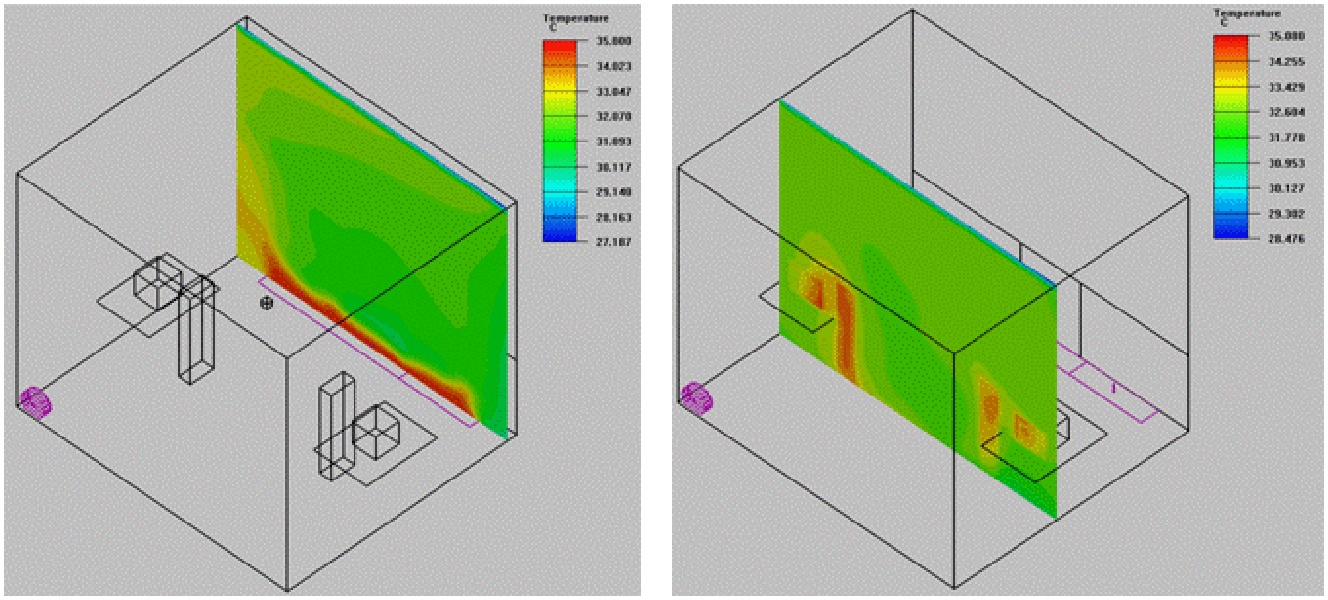
4.2 Air Velocity

Air velocity is an important parameter that defines the thermal comfort of the occupants in the workplace. Strong air flow will cause discomfort to people in the room. In this research, strong flow could be easily awared if an occupant stands directly above the supply diffuser, as shown in Figure 6. Figure 7 demonstrates that the air velocity near the occupants is about 0.15m/s in Case 1, and 0.3m/s in Case 2. The velocity magnitude in Case 2 significantly exceeds the recommended limit for the average air movement of 0.25m/s [23], while the air velocity in Case 1 meets the requirement. It can be deduced that the occupants in Case 2 would complain about the strong air movement. Furthermore, high air flow rates are always accompanied with noise. Since the air velocity coming from supply diffusers in Case 2 is much higher than that in Case 1, the occupants complaint is inevitable, unless the fan coil units are turned off.

During the experiment, we can observe that in both systems, the output air goes vertically upwards. After travelling for a while, it hits the ceiling and travels along it. Then, it begins to drop after a certain distance, as shown in Figure 8. Such an airflow pattern implies that occupants in the Case 2, with the breathing level at 1.2m, are less likely to receive fresh air from the supply diffusers due to the higher air velocity.

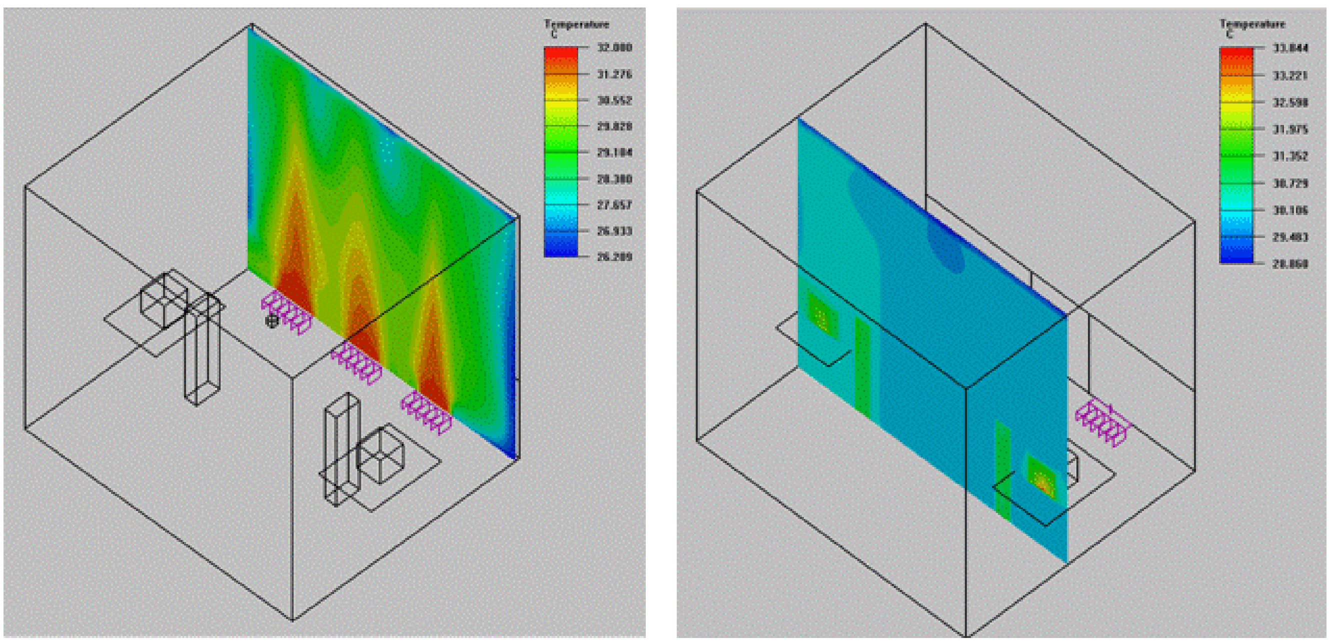
4.3 Thermal Sensation

The measured values of the temperature, the humidity, and the air velocity are used in the computation of predicted mean vote (PMV). PMV is arguably the most widely used thermal comfort index today. Originally developed by Fanger [25], the PMV refers to a thermal scale that runs from Cold (-3) to Hot (+3). The ISO Standard uses limits on PMV as an explicit definition of the comfort zone. In our study, the predicted values of PMV are illustrated in Figures 9 and 10. The PMV values in Case 1 indicate that most calculated values near the location of occupants fall in the range of neutral and slightly warm while



(a) (b)

Figure 4. Vertical temperature distributions at two different locations in Case 1: (a) around fan coil units, (b) around occupants.

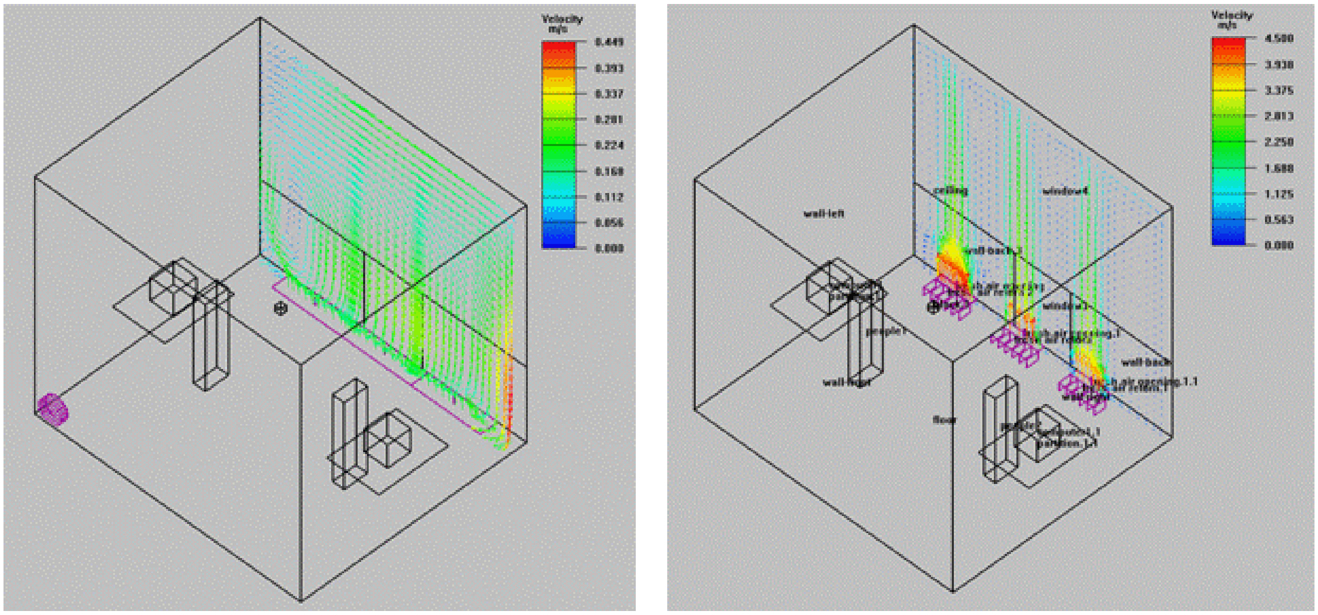


(a) (b)

Figure 5. Vertical temperature distribution at two different locations in Case 2: (a) around fan coil units, (b) around occupants.

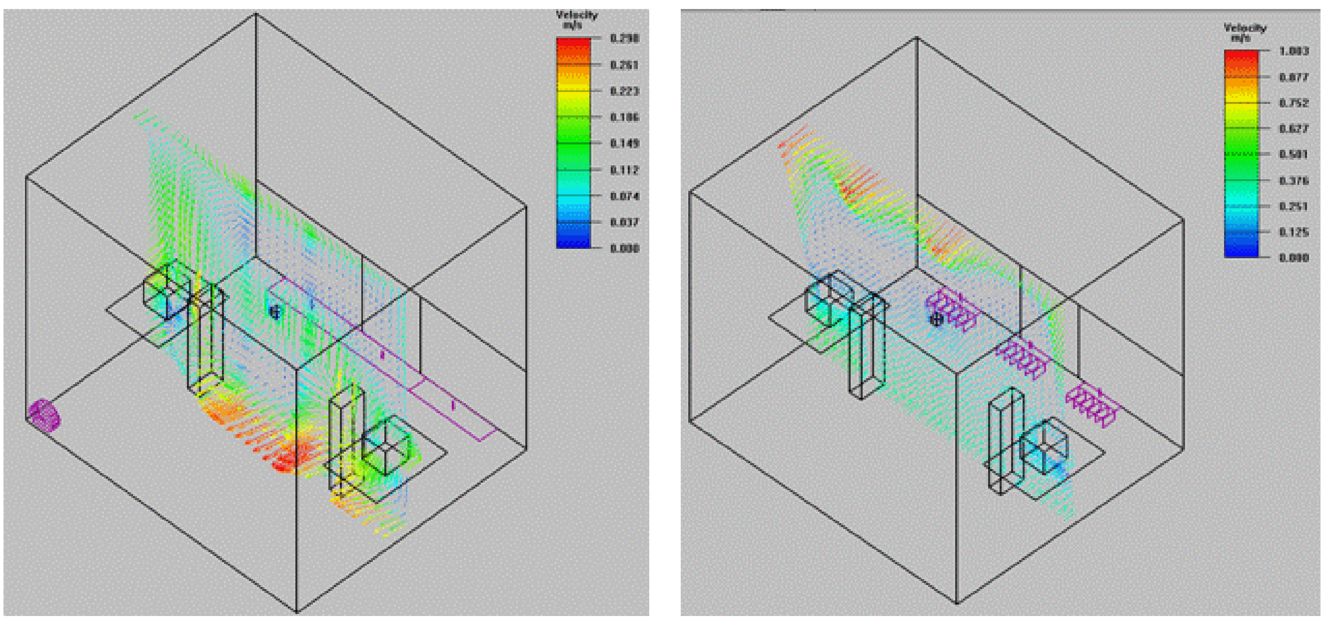
most values at the same position in Case 2 could be classified as neutral. In both cases, most of the PMV values at the points directly above the supply diffusers are observed to vary. In Case 1, the space which is directly above the diffusers is observed to be warm while the rest part of the plane falls can be classified as slightly warm. As for Case 2, the same trend is observed.

Generally, the thermal sensation of occupants is noted to be within the range of neutral to slightly warm [25]. Occupants seldom feel discomfort when they are in a warm area. People would claim discomfort if they stand directly above the diffusers or near the window due to the strong air movement and high temperature. Nevertheless, as they begin to move away from the dif-



(a) (b)

Figure 6. The distribution of air velocity above the air supply diffusers in Case 1and Case 2: (a) Case 1, (b) Case 2.



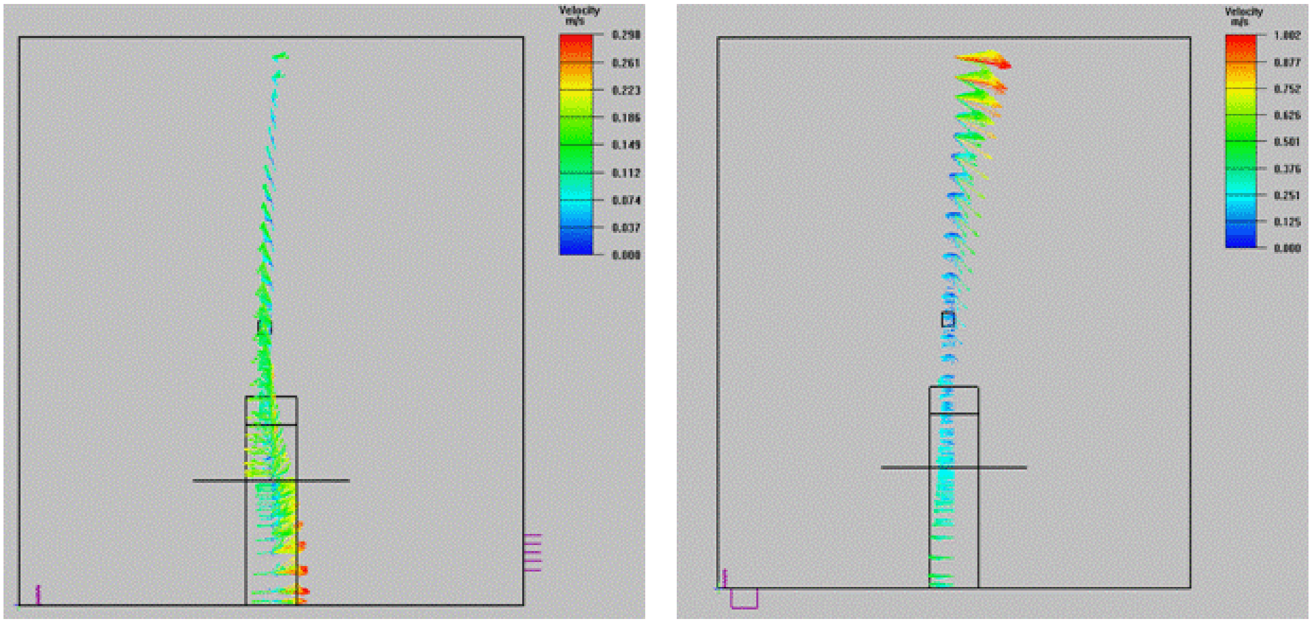
(a) (b)

Figure 7. The distribution of air velocity around occupants in Case 1 and Case 2: (a) Case 1, (b) Case 2.

fusers, they feel more comfortable and make fewer complaints.

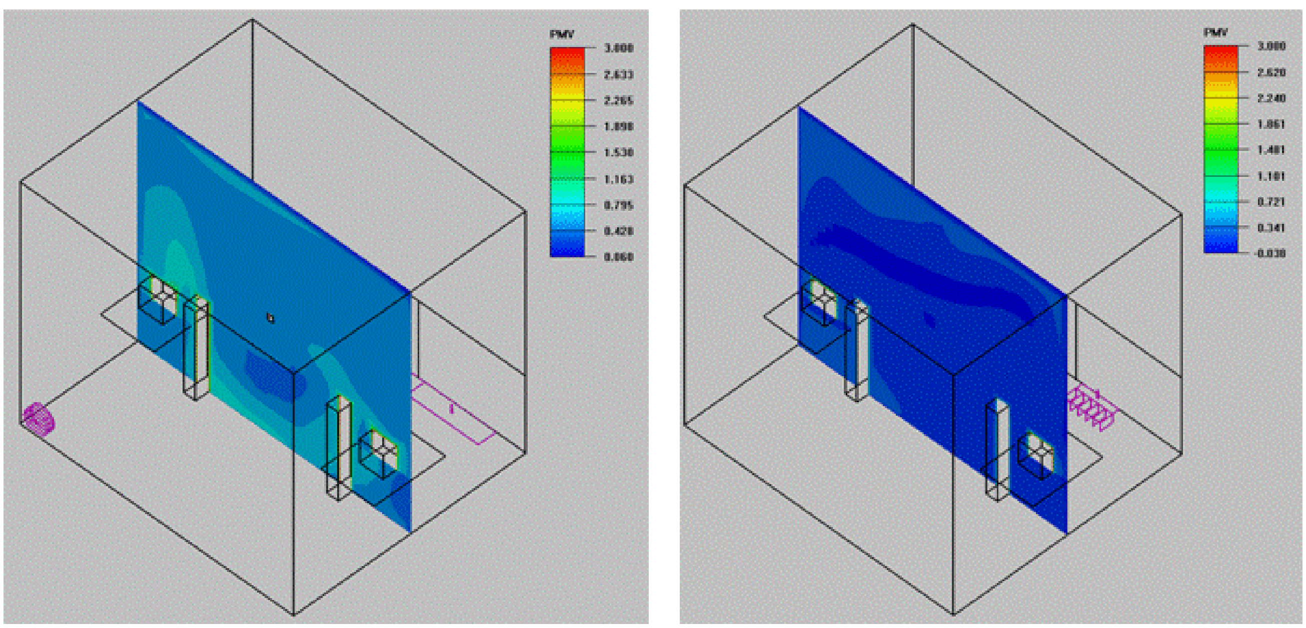
4.4 Validation of Simulation

In order to validate our simulation results, several experiments were conducted for each case and this paper only presents the temperature profiles. For each case, three small sampling points with the size of 10mm$\times$10mm$\times$10mm were set in the middle of the room at the height of 2 feet, 4 feet, and 8 feet. Sensors of temperature are also placed at the same place. During the experiment, we use plastic sheets to cover the whole boundaries of System 1 except the wall with window to make sure the experiment conditions are similar to our simulation model. The plastic sheets make sure that there is no air circulating and heat-



(a) (b)

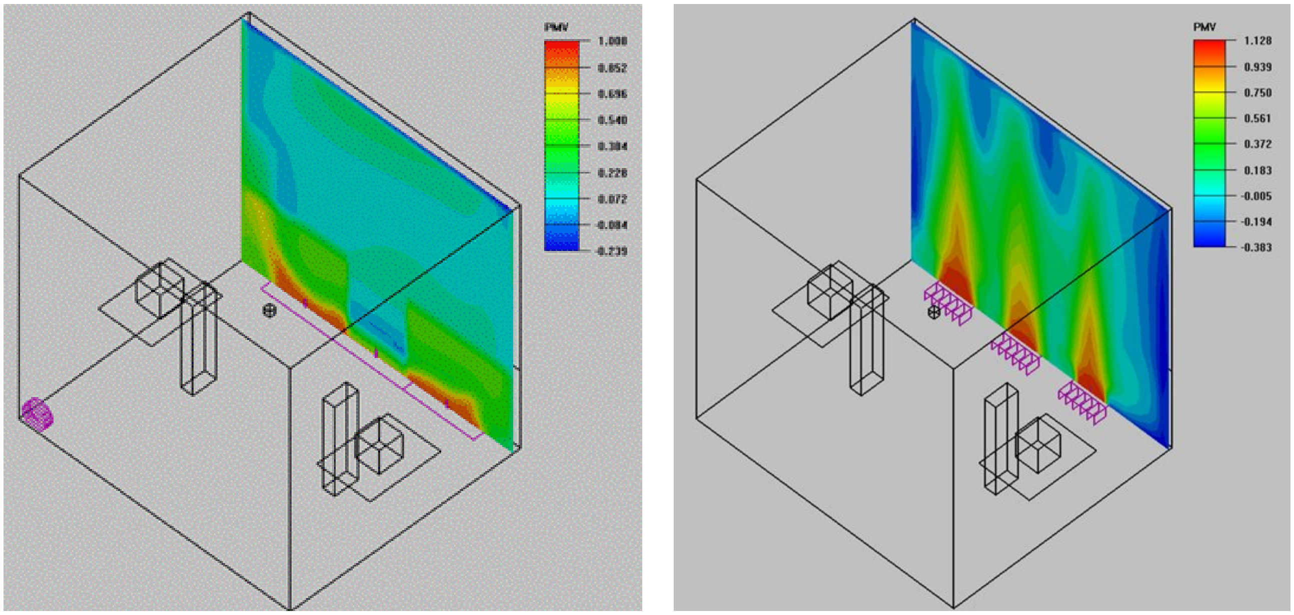
Figure 8. The distribution of air velocity around occupants in Case 1, Case 2 (Viewed from x direction): (a) Case 1, (b) Case 2.



(a) (b)

Figure 9. The PMV distribution around occupants in Case 1 and Case 2: (a) Case 1, (b) Case 2.

ing flux through the wall. Thus, the experimental results are obtained from a similar system and we can compare the simulated results with the experimental results to validate the model. As Figure 11 shows, our calculation results show that the temperature values at the 2 feet, 4 feet, and 8 feet positions are 24.6°C, 24.7°C, and 24.8°C, respectively. As to the experimental data, the temperature values at 2 feet, 4 feet, and 8 feet positions are 24.37°C, 24.45°C, and 24.52°C. The numerical solutions and the experimental measurements match very well. The maximum difference is within 0.28°C.



(a) (b)

Figure 10. The PMV distribution above the air supply diffusers in Case 1 and Case 2: (a) Case 1, (b) Case 2.

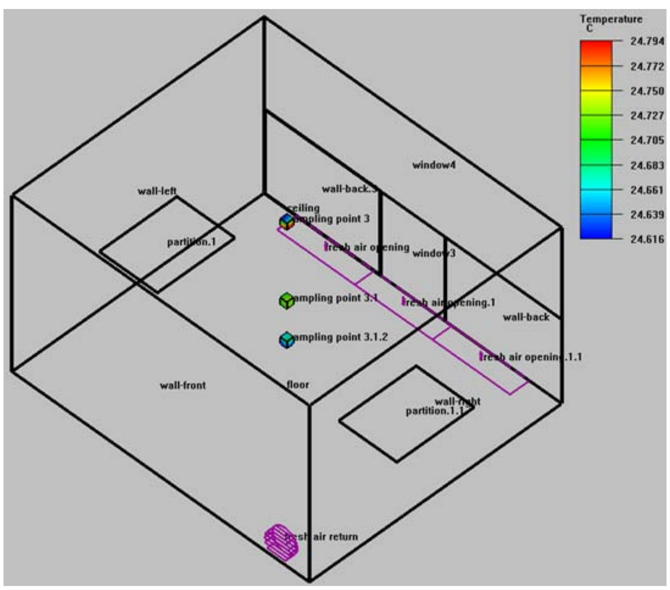


Figure 11. Case 1: the room with VKD fan coils.

5 CONCLUSION

In this study, we simulated airflow inside two rooms with two different types of fan coil units in order to investigate the energy efficiency and the air quality within the two systems. Experiments were conducted to check the validation of the models in the simulation process. Room geometry used in the simulation well represents these office spaces used in the experiments.

Table 4. The comparison of two systems.

Parameter	Case 1	Case 2
Temperature of occupants surroundings		More suitable
Vertical temperature difference	Less	
Air velocity		Sense strong air movement and more noise
Fresh air	More	
PMV	Warm	Neutral

Our computational results are compared with the measurement results. Based on our CFD computations, the temperature, air velocity and PMV values of the two fan coil systems are compared in Table 4.

6 FUTURE WORK

The predicted results are reasonably good. However in order to better understand thermal comfort and energy efficiency in the IW, the computational accuracy needs to be improved, and more simulations should be carried out according to various boundary conditions. The future work are listed as follows.

1. Improved predictions might be obtained by modifying the model to match better with the real room. For example, the roof of the two systems can be represented by a polygonal prism roof instead of a flat one.
2. Additional temperature points could be added to other positions of the space, such as the windows, to check the validation of the models. Moreover, we could measure more parameters during the experiments (for example, the air ve-

locity of the fresh air return) and compare them with the computation results.

3. In different seasons of Pittsburgh, the outside temperature and buoyancy effect vary. Since fan coil units could provide a variety of flow patterns, we can simulate the model at different air velocities. It will help us to determine the most efficient air speed that could provide comfort to the occupants.
4. The difference of the solar radiation at different time will lead to a big difference of temperature distribution. In the future, we could include another important parameter, solar radiation, in our simulation to get complete properties of the systems through out the whole day.

ACKNOWLEDGMENT

We would like to thank Yun Gu, Viraj Srivastava, and Rui Zhang in the IW centre of Carnegie Mellon University for their help on this project.

REFERENCES

[1] Cheong, K. W., Djunaedy, E., Poh, T. K., Tham, K. W., Sekhar, S. C., Wong, N. H., and Ullah, M. B., 2003. "Measurements and computations of contaminant's distribution in an office environment". *Building and Environment,* **38**(1), pp. 135–145.

[2] Sekhar, S. C., and Ching, C. S., 2002. "Indoor air quality and thermal comfort studies of an under-floor air-conditioning system in the tropics". *Energy and Buildings,* **34**(5), pp. 431–444.

[3] Yongson, O., Badruddin, I. A., Zainal, Z. A., and Narayana, P. A., 2007. "Airflow analysis in an air conditioning room". *Building and Environment,* **42**(3), pp. 1531–1537.

[4] Bojic, M., Yik, F., and Lo, T. Y., 2002. "Locating air-conditioners and furniture inside residential flats to obtain good thermal comfort". *Energy and Buildings,* **34**(7), pp. 745–751.

[5] Mathews, E. H., Botha, C. P., Arndt, D. C., and Malan, A., 2001. "Hvac control strategies to enhance comfort and minimise energy usage". *Energy and Buildings,* **33**(8), pp. 853–863.

[6] Myhren, J. A., and Holmberg, S., 2008. "Flow patterns and thermal comfort in a room with panel, floor and wall heating". *Energy and Buildings,* **40**(4), pp. 524–536.

[7] Zhang, L. Z., and Niu, J. L., 2003. "Indoor humidity behaviors associated with decoupled cooling in hot and humid climates". *Building and Environment,* **38**(1), pp. 99–107.

[8] Chen, Q., and Kooi, J. V., 1990. "A methodology for indoor airflow computations and energy analysis for a displacement ventilation system". *Energy and Buildings,* **14**(4), pp. 259–271.

[9] Chen, Q., 1990. "Comfort and energy consumption analysis in buildings with radiant panels". *Energy and Buildings,* **14**(4), pp. 287–297.

[10] Anderson, J., 1995. *Computational fluid dynamics*. McGraw-Hill.

[11] Baker, A. J., Kelso, R. M., Gordon, E. B., Roy, S., and Schaub, E. G., 1997. "Computational fluid dynamics: A two-edged sword". *ASHRAE,* **39**(8), pp. 51–58.

[12] Haghighat, F., Jiang, Z., and Wang, J., 1991. "A CFD analysis of ventilation effectiveness in a partitioned room". *Indoor Air,* **1**(4), pp. 606–615.

[13] Gan, G., 1995. "Evaluation of room air distribution systems using computational fluid dynamics". *Energy and Buildings,* **23**(2), pp. 83–93.

[14] Chen, Q., and Srebric, J., 2001. How to verify, validate, and report indoor environment modeling CFD analyses. Final Report for ASHRAE RP-1133, Welsh School of Architecture, Cardiff University, UK and Department of Architectural Engineering, Pennsylvania State University, PA.

[15] Chris, C. N., and Dang, T., 2005. "Validation of CFD for the flow around a computer simulated person in a mixing ventilated room". In Proceeding of the 10th International Conference, Indoor Air Quality and Climate, International Energy Agency.

[16] Mak, C. M., Niu, J. L., Lee, C. T., and Chan, K. F., 2007. "A numerical simulation of wing walls using computational fluid dynamics". *Energy and Buildings,* **39**(9), pp. 995–1002.

[17] Givoni, B., 1968. Ventilation problems in hot contries. Research report to the ford foundation, Technion, Building Research Station, Israel Institute of Technology, Haifa, Israel.

[18] Givoni, B., 1968. Basic study of ventilation problems in housing in hot countries. Research report to the ford foundation, Technion, Building Research Station, Israel Institute of Technology, Haifa, Israel.

[19] Gan, G., Awbi, H. B., and Croomeg, D. J., 1991. "Airflow and thermal comfort in naturally ventilated classrooms". In 12th AIVC Conference, Air Movement and Ventilation Control in Buildings, International Energy Agency, pp. 24–27.

[20] Mak, C. M., Niu, J. L., Lee, C. T., and Chan, K. F., 1994. "Numerical method for a full assessment of indoor thermal comfort". *Indoor Air,* **4**(3), pp. 154–168.

[21] Chen, Q., and Xu, W., 1998. "A zero-equation turbulence model for indoor air flow simulation". *Energy and Buildings,* **28**(2), pp. 137–144.

[22] http://www.ansys.com/products/airpak/default.asp+.

[23] ENV, 1996. Guidelines for Good Indoor Air Quality in Office Premises. Tech. rep., S. Ministry of the Environment.

[24] ISO, 1984. International Standard 7730, Moderate Thermal Environments-Determination of PMV and PPD Indices and Specification of the Conditions of Thermal Comfort. Tech. rep., International Standards Organisation.

[25] Fanger, P. O., 1995. *Thermal Comfort*. McGraw-Hill.