A NUMERICAL INVESTIGATION OF INDOOR AIR QUALITY WITH CFD

Vai Kuong Sin[1*] and Ho I Sun[1]

Macao, a city with three sides bounded by water, is hot and humid in weather in more than six months of a year. This uncomfortable weather induces the frequency of operating air-conditioners. Choice of location for installation of air-conditioner in a building will affect the performance of cooling effect and thermal comfort on the occupants, which in turn will affect the indoor air quality (IAQ) of the building. In the paper, investigation of distribution on carbon dioxide, room air temperature and velocity, as well as air diffusion performance index (ADPI) of a single bedroom in Macao is studied by using the computational fluid dynamics (CFD) software FLOVENT 3.2. Simulations of locating the air-conditioner at 4 different walls will be done and comparisons and analyses of the results will be performed to decide a proper location for the air-conditioner for obtaining good thermal comfort.

Keywords: Indoor Air Quality (IAQ), Computational Fluid Dynamics (CFD), Turbulence

1. INTRODUCTION

Nowadays, indoor air quality (IAQ) is being an important issue for it has a close relationship with human health. Indoor air pollutants such as carbon monoxide, formaldehyde, ozone, toluene, environmental tobacco smoke and respiratory suspended particles are those that are harmful to human when their concentrations reach a certain level. When ventilation is not enough for indoor environment, these pollutants will be aggregated and reach their harmful level.[1] However, the Macao government has not set any guideline or standard for IAQ, therefore, it is important to investigate the IAQ of the building in Macao. In Macao, most of the residential and commercial buildings need to be operating with air-conditioners for more than half of a year because of the hot and humid weather of Macao. Air temperature and velocity are the first parameters that affect the thermal comfort of the occupants and will straightly affect the IAQ of the building. In order to investigate the IAQ of Macao, analyses of these parameters as well as carbon dioxide distribution of the buildings have to be done. However, different locations of air-conditioner will affect the distribution of carbon dioxide, and these will then affect the IAQ of the building. Moreover, levels of indoor carbon dioxide greater than 1000ppm are generally assumed to be the indicator of inadequate ventilation.

For analysing the air distribution of the building, an important parameter to be investigated is air diffusion performance index (ADPI). ADPI[2] is a parameter that measures the uniformity of the space in terms of the proportion of the volume with velocity lower than 0.35 m/s and draft temperature, T_d, between -1.7°C and +1.1°C from the mean temperature. For example, an ADPI of 0.8 means that 80% of the locations in the simulated area have velocity lower than 0.35 m/s and draft temperature between -1.7°C and +1.1°C from the mean temperature. The most comfortable condition of the room is obtained when the ADPI is 1.0 where 100% of the locations in the simulated area meet the comfort criteria as stated. Generally, an acceptable value of ADPI is 0.8.[3] Another parameter for analysing the IAQ is the distribution of the carbon dioxide emitted by the occupant. Since the carbon dioxide will be diffused due to concentration difference and convected by the flow of air, analyses of distribution of carbon dioxide can give result showing whether concentration of carbon dioxide is maintained at the acceptable level.

Received: July 15, 2004, Accepted: January 24, 2005.

1 Department of Electromechanical Engineering, University of Macau, Macao SAR, China.

The objective of the paper is to investigate the choice of location for air-conditioner installation in order to accomplish the best thermal comfort for occupants as well as acceptable IAQ in a bedroom. Distributions of air temperature, velocity, carbon dioxide and value of ADPI are simulated with the commercial CFD code FLOVENT 3.2. FLOVENT has been successfully used by the authors to investigate the thermal comfort in a double bedroom.[4]

2. MATHEMATICAL MODEL AND SIMULATION OF TURBULENCE - FLOVENT

In FLOVENT, Reynolds-averaged Navier-Stokes equations are solved with finite volume technique and turbulence effects upon the mean flow are modelled through the eddy viscosity concept. Steady state analysis is to be used and the governing equations are

$$\frac{\partial U_i}{\partial x_i}=0 \quad (1)$$

$$\rho U_i\frac{\partial U_j}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\mu\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\rho\overline{u_j u_i}\right)-\frac{\partial p}{\partial x_j}+g_j(\rho_r-\rho) \quad (2)$$

$$\rho C_p U_i\frac{\partial T}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\lambda\frac{\partial T}{\partial x_i}-\rho C_p\overline{u_i T'}\right)+q \quad (3)$$

$$\rho U_i\frac{\partial C}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\gamma\frac{\partial C}{\partial x_i}-\rho\overline{u_i C'}\right)+\rho C \quad (4)$$

where ρ is density of air; ρ_r is reference density of air; μ is viscosity of air; U_i is mean velocity; u_i is fluctuating velocity; T is mean temperature; T' is fluctuating temperature; C_p is specific heat at constant pressure of air; λ is thermal conductivity; q is energy production rate; g_j is gravitational acceleration; C is the mean mass concentration of carbon dioxide (CO_2); C' is fluctuating concentration of carbon dioxide; γ is mass diffusion coefficient of CO_2.

$-\rho\overline{u_j u_i}$ is the Reynolds stress which is defined as:

$$-\rho\overline{u_j u_i}=\mu_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\frac{2}{3}\rho k\delta_{ij} \quad (5)$$

where μ_t is turbulent viscosity; δ_{ij} is Kronecker delta.

Turbulent viscosity, μ_t, defined from dimensional analysis as:

$$\mu_t=C_\mu\rho\frac{k^2}{\varepsilon} \quad (6)$$

where $C_\mu = 0.09$; k is turbulent kinetic energy; ε is dissipation rate of k.

$-\rho C_p\overline{u_i T'}$ is the Reynolds heat flux which is defined as:

$$-\rho C_p\overline{u_i T'}=\lambda_t\frac{\partial T}{\partial x_i} \quad (7)$$

Turbulent conductivity, λ_t, is related to turbulent viscosity by the following equation:

$$\lambda_t=\frac{C_p\mu_t}{\sigma_t} \quad (8)$$

where σ_t is the turbulent Prandtl number (equal to 0.9).

$-\rho\overline{u_i C'}$ is the Reynolds concentration flux which is defined as:

$$-\rho\overline{u_i C'}=\gamma_t\frac{\partial C}{\partial x_i} \quad (9)$$

where γ_t is turbulent mass diffusion coefficient. It is related to turbulent viscosity by

$$\gamma_t=\frac{\mu_t}{Sc_t} \quad (10)$$

where Sc_t is turbulent Schmidt number (equal to 1.0).

Transport equations for k and ε are:

$$\frac{\partial(\rho U_i k)}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right)+P+G-\rho\varepsilon \quad (11)$$

$$\frac{\partial(\rho U_i \varepsilon)}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right)+C_1\frac{\varepsilon}{k}(P+C_3G)-C_2\rho\frac{\varepsilon^2}{k} \quad (12)$$

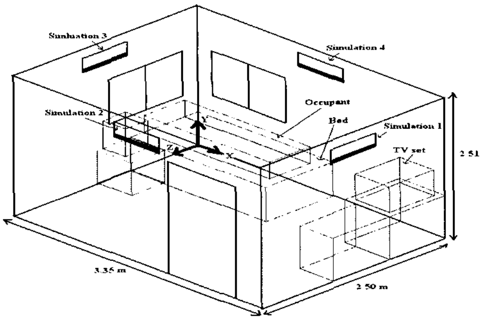

Fig. 1 3-Dimensional layout of the simulated bedroom

where C_1 = 1.44; C_2 = 1.92; C_3 = 1.0; σ_k = 1.0; σ_ε = 1.217; P is shear production defined as:

$$P = \mu_{eff} \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (13)$$

G is production of turbulence kinetic energy due to buoyancy, and is given by:

$$G = \frac{\mu_{eff}}{\sigma_t} \beta g_i \frac{\partial T}{\partial x_i} \quad (14)$$

where $\mu_{eff} = \mu + \mu_t$ is the effective viscosity; β is coefficient of volumetric expansion.

The draft temperature, T_d , which is used to obtain the ADPI, is calculated using the following formula [2]:

$$T_d = T_a - T_m - 7.66 \times (\text{Speed} - 0.15) \quad (15)$$

where T_a is air temperature (i.e. fluid temperature); T_m is mean air temperature in Degree Celsius over all empty regions of all mean flow regions; and Speed is the local air-stream centerline speed which is in m/s. The simulation is terminated when the mass continuity residual (E_P), velocity residual (E_u, E_v, E_w), temperature residual (E_T), and concentration residual (E_C) are reaching the following convergent criteria [2]:

$E_P = 10^{-4}$ kg/s; $E_u = E_v = E_w = 10^{-3}$ N;
$E_T = 10^1$ W; $E_C = 10^{-7}$ kg/s.

3. SIMULATION BEDROOM

The simulated room is a single bedroom in Macao with air-conditioner operated and simple furniture located as shown in Fig. 1. The overall size of the bedroom is 3.35m×2.50m×2.51m with two external walls where two pairs of windows are located, two internal walls, door, floor and ceiling. The windows are made of single-glazed glass with the overall thermal transmission coefficient(U) of 5.9 W/m^2K and that of the external walls is 2.35 W/m^2K.[5] The doors and windows are designed to be closed during the air-conditioner is operated. The internal walls as well as the door, floor and ceiling are assumed to be thermally insulated. Simple furniture that are inside the room include a bed, a bed table, a bed lamp, a dressing table and a TV set. The air-conditioner is a split type one which supplies temperature at 16°C with the supply angle at 90^0 which is normal to the wall in all the four cases. The flow rate of it is 200 m^3/hr (9.5 ACH) where 15% of it is fresh air and the rest is return air from the room. Air change per hour (ACH) is the number of changes of total room volume of air per hour. An occupant who generates metabolic heat of 70 W[5] and breathes out 8.2×10^{-6}kg/s[6] of carbon dioxide is sleeping on the bed. The volume of the occupant is assumed to be $0.28m^3$ so that the metabolic heat divided by this volume will be the value of q in(3). During summer in Macao, theoutdoor air temperature is set as 33°C and the initial temperature of the simulated bedroom is set

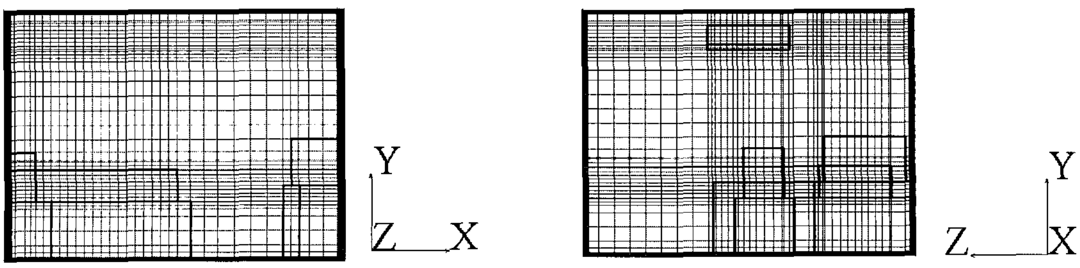

Fig. 2. Non-uniform grids of z-view (left) and x-view (right) of simulation 1

Table. 1 Location of air-conditioner and number of grids used

	Simulation 1	Simulation 2	Simulation 3	Simulation 4
Location of Air-conditioner (x, y, z) in meter	(3.35,2.1,0.95)	(1.37,2.1,2.5)	(0,2.1,0.95)	(1.37, 2.1,0)
Number of grid used	57,120	63,756	51,408	60,984

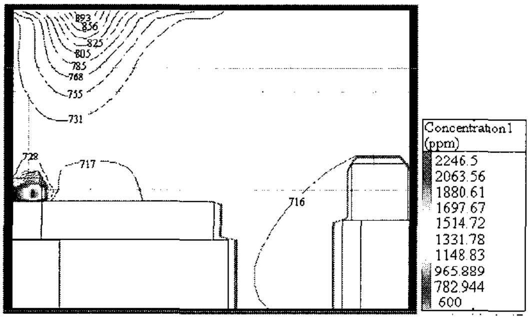

Fig. 3 Concentration of carbon dioxide of simulation 1

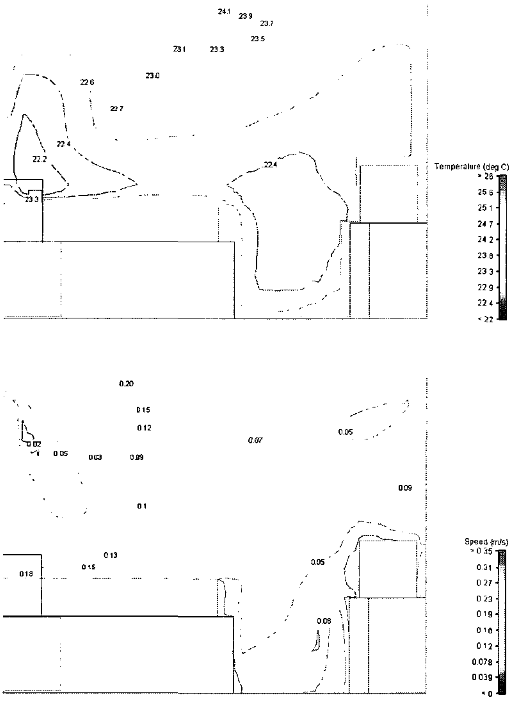

Fig. 4 Temperature (top) and velocity (bottom) contours of simulation 1

as 30°C. Thermal radiation from the outdoor through the windows is not considered in this simulation as Venetian blinds are assumed to be used. For the purpose of modelling simplification, only convective and conductive heat transfer from the occupant and the air-conditioner are being concerned, any heat released from the other furniture and walls are not taken into account.

Four simulations are to be done where the air-conditioner is placed at different wall for each simulation (see Fig. 1). In simulation 1, the air-conditioner is located at x = 3.35m, y = 2.1m, z = 0.95m which is above the TV set and opposite to the occupant. The total number of grids used is 57,120 (see Fig. 2). In simulation 2, the air-conditioner is placed at x = 1.37m, y = 2.1m, z = 2.5m which is on the internal wall where the door is located. The number of grids used is 63,756. In simulation 3, the air-conditioner is located at x = 0m, y = 2.1m, z = 0.95m which is at the external wall where one pair of windows is located. The number of grid used is 51,408. In simulation 4, the air-conditioner is located at x = 1.37m, y = 2.1m, z = 0m which is at another external wall next to the occupant and where another pair of windows is located. The total number of grid used is 60,984. Summary of locations of the air-conditioner and the number of grids being used in each of these four simulations are given in Table 1. The grids are non-uniformly distributed so that it is fine enough for obtaining accurate results. All simulations are run by a Pentium IV 2.4GHz personal computer with 512MB RAM.

4. RESULTS AND DISCUSSIONS

In order to analyze the air velocity, temperature and concentration of carbon dioxide, a vertical plane at z = 0.48m which is in the centerline of the occupant is used to show the contour of these parameters. The analysis will be focused on the area near the head of the occupant because it is the only portion exposing to the air when the occupant sleeps and covers his body with blanket.

In simulation 1, the concentration of carbon

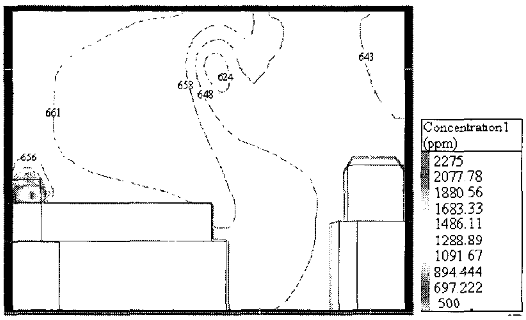

Fig. 5 Concentration of carbon dioxide of simulation 2

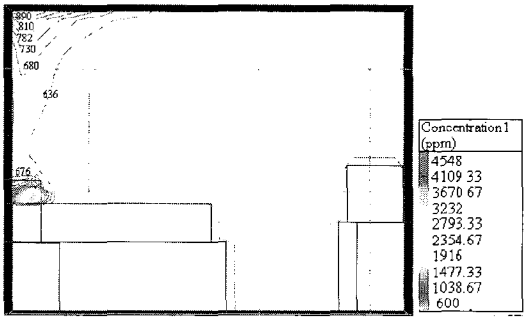

Fig. 7 Concentration of carbon dioxide of simulation 3

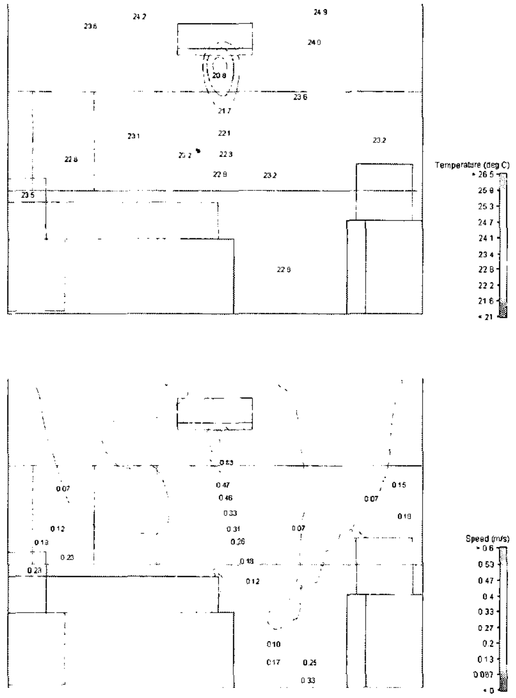

Fig. 6 Temperature (top) and velocity (bottom) contours of simulation 2

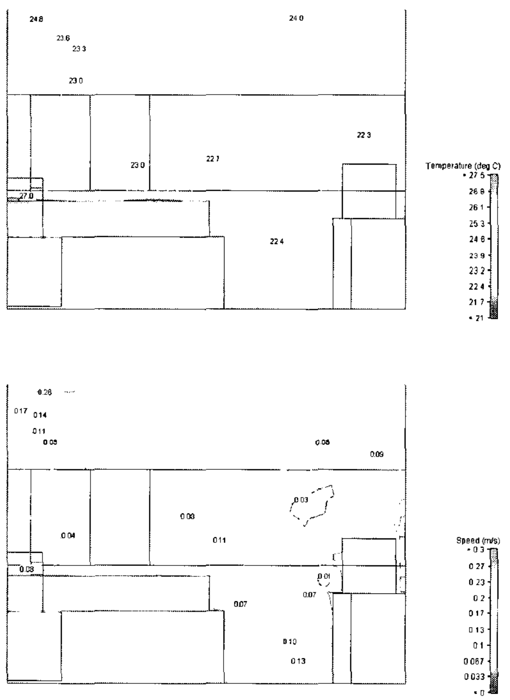

Fig. 8 Temperature (top) and velocity (bottom) contours of simulation 3

dioxide is about 731ppm within the room except at the area near the top left corner of the ceiling which has a relatively high value of about 893ppm (see Fig. 3). At the area near the nose of the occupant, the concentration of carbon dioxide is very high, this is because the occupant is emitting carbon dioxide continuously, and so it is acceptable to have high concentration at this area. The temperature of the room ranges from 14.9°C to 27.5°C and that around the occupant's face is about 23.3°C with the velocity of about 0.18m/s (see Fig. 4). The ADPI of the room is 0.808.

In simulation 2, the carbon dioxide is distributed uniformly within the room with the value of about 660ppm (see Fig. 5). The high concentration of carbon dioxide around the occupant's nose is again due to the fact that this is the source of continuous carbon dioxide emission. The ADPI is 0.814. The temperature of the room ranges from 16.1°C to 30.2°C and that around the occupant's face is about 23.5°C with the velocity of about 0.28m/s (see Fig. 6).

In simulation 3, the concentration of carbon

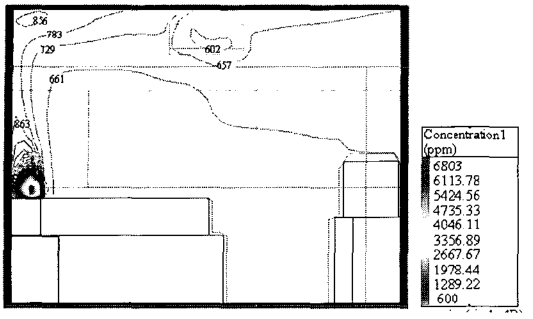

Fig. 9 Concentration of carbon dioxide of simulation 4

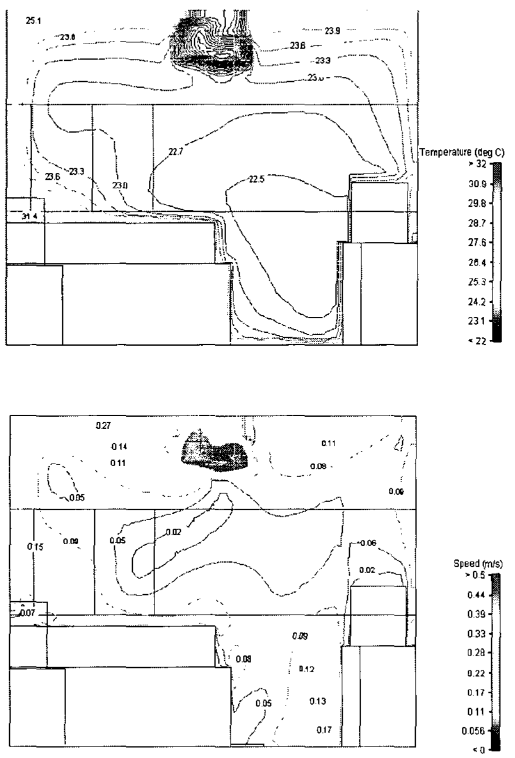

Fig. 10 Temperature (top) and velocity (bottom) contours of simulation 4

dioxide is about 636ppm within the room except at the area near the top left corner which has a relatively high value of about 890ppm (see Fig. 7). High concentration of carbon dioxide is found again around the occupant's nose. The ADPI is 0.783. The temperature of the room ranges from 15.5 °C to 30.8°C and the highest temperature around the occupant's face is about 27°C with the velocity of about 0.08m/s (see Fig. 8).

In simulation 4, the concentration of carbon dioxide is about 661ppm within the room except at the area near the top left corner which has a relatively high value of about 856ppm (see Fig. 9). The high value around the occupant's face is due to the continuous source by the occupant. The ADPI is 0.809. The temperature of the room ranges from 15.3°C to 31.6°C and the highest temperature around the occupant's face is about 31.4°C with the velocity of about 0.07m/s (see Fig. 10). Summary of results of all four simulations is given in Table 2.

Of all four simulations, except simulation 2, there is a high concentration of carbon dioxide found at the top left corner of the ceiling in the plane of $z = 0.48$m. This means that carbon dioxide is trapped in this area. It is also noted that the ADPI of simulation 2 is the best among all simulations. However, this area with high concentration of carbon dioxide will not directly affect the thermal comfort of the occupant for it is far away from the occupant. Except at the area just discussed, the air is distributed quite uniform within the bedroom in all cases. The relatively high value of concentration of carbon dioxide around the occupant's face in simulations 3 and 4 is due to the very low air velocity at this area, so that the dispersion of carbon dioxide is slow here. By comparing the air temperature around the occupant's face, it shows that, again because of low air velocity, temperature is higher than 26°C in simulations 3 and 4, which makes these two simulations not acceptable to be

Table. 2 Summary of results for all four simulations

	Simulation 1	Simulation 2	Simulation 3	Simulation 4
ADPI	0.808	0.814	0.783	0.809
Velocity around occupant's face (m/s)	0.18	0.28	0.08	0.07
Temperature around occupant's face (°C)	23.3	23.5	27	31.4

considered as a good air-conditioned bedroom. The air temperature around the occupant's face is moderate in simulations 1 and 2 but the velocity of simulation 2 is too high that makes the occupant uncomfortable. Therefore, although simulation 2 has the highest ADPI, it is suggested that the environment of simulation 1 is the best for an air-conditioned bedroom. It is noted that installation of air-conditioner in location of simulation 1 may take more work such as using longer piping to connect the indoor unit and outdoor unit of the split type air-conditioner since it is further away from the windows when comparing with the locations in simulations 3 and 4. Due to this reason, most workers will try to install the air-conditioner in the location as in simulations 3 and 4 as it can lower the cost by using shorter piping and is more convenient for them during installation. However, installation in locations as in simulations 3 and 4 will strongly influence the IAQ of the bedroom as discussed before, therefore, in order to obtain better IAQ, the air-conditioner is recommended to install at the location of simulation 1 despite of possible higher installation cost.

5. CONCLUSIONS

Four different installation locations of the air-conditioner in a bedroom of Macao are simulated by FLOVENT 3.2 and results of these four simulations are compared to obtain the best thermal comfort for the occupant. By concluding the analyses of the distribution of carbon dioxide, air temperature, velocity and the value of ADPI of each case, it is suggested that simulation 1 where the air-conditioner is placed opposite to the occupant and above the TV set is the best among the four simulations. Therefore, in order to have a comfortable environment for the simulated bedroom, it is recommended to install the air-conditioner at the location of simulation 1.

REFERENCES

[1] Anonymous, 2001, "ASHRAE Standard 62-2001 Ventilation for acceptable indoor air quality," *American Society of Heating, Refrigerating and Air-conditioning Engineers, Inc.*

[2] Anonymous, 2001, *Flovent reference manual, FLOVENT/OL/MM/0801/1/0*, Flomerics Ltd.

[3] Tibaut P. and Wiesler B., 2002, "Thermal comfort assessments of indoor environment by means of CFD," *8th International Conference Air Distribution in Rooms.*

[4] Sin V.K., Sun H.I., Tam L.M. and Wong K.I., 2003, "Numerical experiment of thermal comfort in a bedroom equipped with air-conditioner," *Proceedings of the Ninth International Conference on Enhancement and Promotion of Computational Methods in Engineering Science (EPMESC IX),* November 25-28, Macau, China.

[5] Stocker W.F. and Jones J.W., 1982, *Refrigeration & air conditioning*, 2nd edition, McGraw Hill Book Company.

[6] Anonymous, 1993, *ASHRAE handbook of fundamentals 1993*, American Society of Heating, Refrigerating and Air-conditioning Engineers, Inc.