AIVC
#13,526

CFD Simulation of Transient Cooling in a Typical Hong Kong Office

Z Lin, T T Chow, J P Liu and K F Fong
Division of Building Science and Technology, City University of Hong Kong

Summary

The transient performance of displacement ventilation has rarely been studied though many researches have been done on the area.. Due to the importance to the analysis and design of an air conditioning system, the unsteady histories of displacement ventilation of a typical office in Hong Kong is simulated numerically with a validated computational fluid dynamics (CFD) model. Solar heat gain is introduced from the glass wall. Turbulent flow with thermal convection is considered. As a preliminary research, the heat capacity of the walls, occupants and machines is ignored. With the reasonable air flux determined by steady state simulated, the transient regime were started with uniform initial temperature and zero velocities profile. The numerical simulation illustrated the evolution of stratified temperature profiles in the office: the cooler fresh air supplied spread horizontally first, then grew along the vertical direction gradually. The simulation also revealed the development of the thermal plum produced by hot walls and objects. Hence the study helps to understand the formation of the initial velocity and temperature fields.

Introduction and Background

In order to provide favourite indoor air quality (IAQ), not only the velocity and temperature distributions, but also their transient profiles are need to be known in a ventilated room. A reasonable ventilating system with good dynamic characteristics may establish favourite temperature profile and take away contaminant released in the room rapidly. Such transient property is also significant for the design and performance of the indoor climate control system. The analysis of the transient behaviour of a building may be carried out through two approaches: experimental investigation and computer simulation. In principle, direct measurements give the most realistic information concerning indoor airflow and pollutant transport, such as the distributions of air velocity, temperature, relative humidity, and contaminant concentrations. Because the measurements must be made at many locations, direct measurements of the distributions are very expensive and time consuming. A complete measurement may require many months of work. Moreover, to obtain conclusive results, the supply airflow and temperature from the Heating Ventilating, and Air Conditioning (HVAC) systems and the temperatures of building enclosure should be maintained unchanged during the experiment. This is especially difficult to achieve because the outdoor conditions change over time and the temperatures of the building enclosure and the airflow and/or temperature from the HVAC systems will also change accordingly.

Alternatively, the heat and pollutant transport can be determined computationally by solving a set of conservation equations describing the flow, energy, and contaminants in the system. Due to the limitations of the experimental approach and the increase in performance and affordability of high-speed computers, the numerical solution of these conservation equations provides a practical option for determining the airflow, heat and pollutant distributions in buildings. The method is the Computational Fluid Dynamics (CFD) technique. Several reports deal with the evaluation of the indoor air temperature progress with numerical method recently. Peng *et al.* (1) proposed a method to predict the dynamic temperature profiles in a flow field determined by simulation of the steady state. By assuming that the velocity field is not changing in time the whole simulation is simplified to have only one transient partial differential so that only the unsteady energy balance equation needed to be solved. Ghiaus and Ghiaus (2) further developed Peng *et al.*'s method to accelerate the calculation. Nannei *et al.* (3) used a further simplified numerical model only included energy and mass balance in the whole indoor domain to predict the transient progress and compare it with experiments. However, the real transient progress is of a three dimensional turbulent flow with heat transfer, for the small air velocity in the occupied zone, the flow induced by natural convection is apparent and significant (4). The energy equation and the momentum equation are coupled and should be solved simultaneously. The ignorance of the velocity variation in the transient procedure will introduce error, especially in the occupied zone. However, no report closely relevant to the topic has been found, which precisely describe simulations of the complicated transient problem with coupled energy and momentum equations.

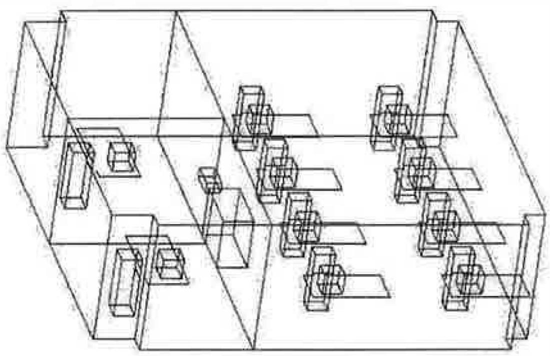

Figure 1. Geometry of the Office

The displacement ventilating system is a popular topic and widely investigated in recent decades. Introducing the supply air from the underside rather than from the upper of the room, the system is generally recognized of better indoor air quality and higher energy efficient than mixing system. Lin *et al.* (5) studied the displacement ventilation in a typical Hong Kong office. As shown in Figure 1, the office is divided into three relatively independent parts by partitions, one larger cubicle office and two-equal individual offices. The block contains ten persons, computers and tables, a photocopier and a fax machine. The solar energy enters from the glass in the south wall. Four diffusers are on the floor and four exhausts on the ceiling. The steady displacement ventilation was investigated. Favorite supply air flowrate and temperature via each diffuser was found. The temperature and velocity distributions for the steady state were determined. In the present paper, the transient starting temperature and velocity profiles are simulated numerically. The CFD technique is a powerful tool to analyze indoor environment problems, such as airflow pattern and the distributions of air velocity, temperature, turbulent intensity, and contaminant concentrations. Due to limited computer power and capacity available at present, turbulence models have to be used in the CFD technique in order to solve flow motion. The use of turbulence models leads to uncertainties in the computed results because the models are not universal. Therefore, it is essential to validate a CFD program by experimental data. If a CFD program is validated by experimental data for displacement ventilation, the program should be able to predict the indoor environments of similar nature. The flow characteristics between displacement ventilation and other mixing ventilation are similar – both have strong pressure and buoyancy driven flows (6).

A validated CFD model based on the RNG k-ε model and wall function (7) was used to generate data for a typical office in Hong Kong (Figure 1). The whole computational domain, the space of the room, needs to be divided into a number of finite volumes by a grid system. The flow variables, such as velocity, temperature and concentration, are solved at the centre of each finite volume. The finer grid is, the more accurate the results will be. However, fine grid will cost more computing time and capacity.

As a preliminary investigation, the purpose of this research was to reveal the general profile of the ventilating system at the pull down stage, this is important for understanding of the mechanism of the complicated physical process. A typical starting process will be simulated: the initial velocity was zero and the initial temperature was uniform at 30°C. At time zero the process started with a given inlet air flow of lower temperature, then the transient momentum and energy transportation occurred in the office. The flow was unsteady three-dimensional turbulent with heat transfer coupled with natural convection caused by density difference. The solid wall, slabs and other objects such as equipment and furniture in the room may exchange large amount of heat with the air when the air temperature is varying, so their heat capacities are important for the transient cooling (8). However, the characteristic time of the heat conduction in the solid objects is much greater than that of the fluid convective heat transfer. To simulate such a problem, the total calculated period is determined by the longest characteristic time but the time step is dominated by the shortest specific time, it might take very bng computation time before any result can be achieved. Another difficulty of such simulation concerns algorithm of CFD. Because of the difference between the solid zone and fluid zone, it will take much longer time to approach convergence at each time step than a single-phase case. However, like the other types of load, the heat stored in solid objects must ultimately be discharged out of the room by the air motion. Therefore, as the first stage, study of the establishment history of the air temperature and velocity in the rooms should logically be the first step towards full understanding of the pull-down regime.

Mathematical Model and Simulation

The governing equations of air flow in ventilated rooms is based on the solution of the general equation according to the symbols in the nomenclature:

$$\frac{\partial}{\partial t}(\rho\,\Phi)+\nabla\cdot\left(\rho\,\vec{V}\Phi\right)=\nabla\cdot\left(\Gamma_\Phi\nabla\Phi\right)+S_\Phi \quad (1)$$

where

ρ = air density,

$\vec{V}$ = velocity vector,

t = time,

Φ = variables to be solved,

Γ_Φ = diffusivities of Φ,

S_Φ = body sources of Φ.

There were seven variables to be considered in the present simulation, the velocities at three directions, pressure, temperature, turbulent kinetic energy and turbulent dissipation. For the RNG $k-\varepsilon$ model of the turbulent flow with Boussinesq assumption of the buoyant term. The detail expressions of Φ , Γ_Φ , and S_Φ are given in (9).

Normal wall function method is adopted and the corresponding boundary conditions of velocities and the temperature are assumed as the following type,

In the wall :

$$\vec{V}=0\ ,\ -k\frac{\partial T}{\partial n}=q_n \quad (2)$$

where

q_n = heat conductivity of the air,

q_n = wall heat flux,

$\frac{\partial}{\partial n}$ = temperature gradient at the normal direction of the wall.

At the inlet plane:

$$\vec{V}=\overrightarrow{V_C}\ ,\ T=T_C \quad (3)$$

Where

$\overrightarrow{V_C}$ = velocity,

T_C = temperature at the inlet plane.

At the outlet plane

$$\frac{\partial\vec{V}}{\partial n}=0\ ,\ -k\frac{\partial T}{\partial n}=0 \quad (4)$$

The initial conditions are:

$$V=0\ ,\ T=T_H \quad (5)$$

Where

T_H = initial temperature.

The initial temperature T_H is 30°C all over the calculated domain. Determined by the heat introduced to the office, the heat flux q_n is zero at the solid wall except the computers, occupants, lamps, glass, north wall and other accessory apparatus.

The above Equations 1 - 6 composed the closed mathematical model of the unsteady ventilating process in the office. A commercial computational fluid dynamics software called CFX4.2 was used to solve the partial differential equations. Based on the finite difference in structured grid system with multi-block composed technology, the code is developed to simulate the complicated fluid flow and transportation phenomena (9). The 84 × 80 × 27 equavelently divided mesh system is adapted to simulate the three-dimensional domain, hybrid scheme is used to discrete the convect-diffusion term and the full implicit differential scheme is applied to deal with the transient terms in the Navier-Stokes equations. Because of the limitation of our computer, the time step used in the simulation is 0.1 s.

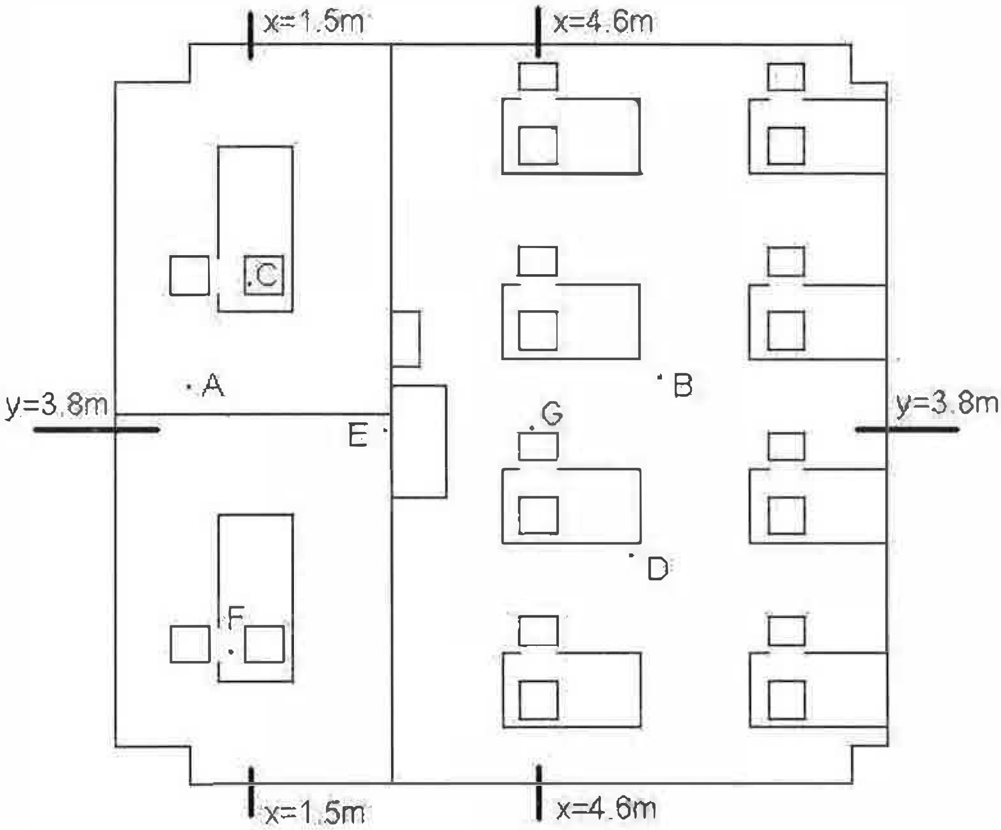

Figure 2. Position of the Plotted Points

Results and Discussions

The simulation starts with zero velocity fields and high uniform temperature profile of 30°C, suddenly the ventilating system begins to work at time zero.

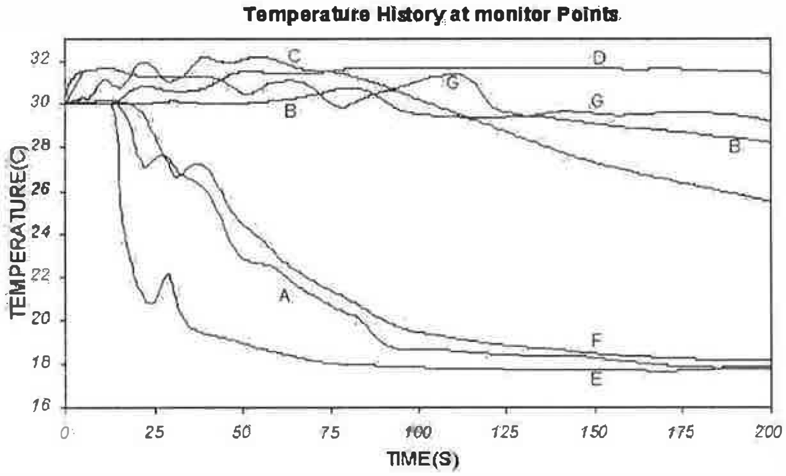

Figure 3. Temperature History at the Points

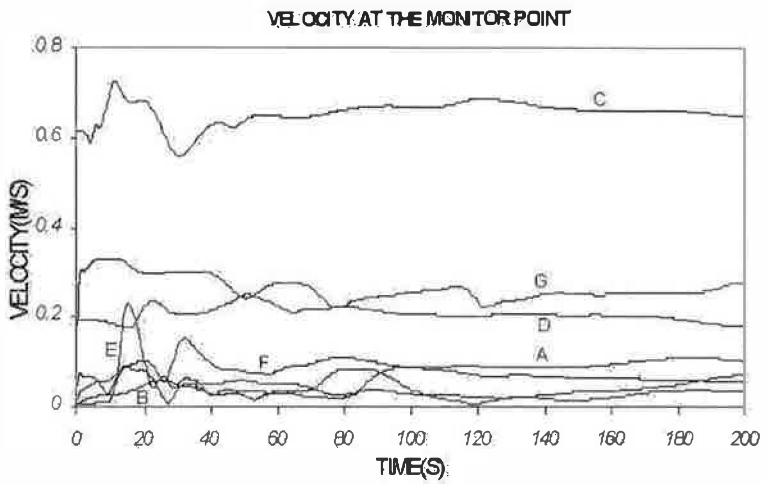

Figure 4. Mean Velocity History at the Points

To illustrate the transient flow properties, velocity, temperature and turbulent kinetic energy at seven points were recorded. As shown in Figure 2, point A and B are 1.0 meters high above the floor, points C and D are 0.15 m far from the centre point of two exhausts that locate in the smalland large parts of the office respectively. Point E locates near the photocopier, points F and G are situated between the occupant and computer. Figure 3-4 show the temperature and mean velocity history from time zero to 200.0 s at the seven points. Figure 2 also gives the location of three planes, x = 1.5 m, x = 4.6 m and y = 3.8 m where the temperature and velocity profile will be given. The plane x = 1.5 m is cross the centres of two diffusers in the small parts of the office, x = 4.6 m cross the centres of four persons and computers in the large part, y = 3.8 m cross the photocopier. Figures 3 - 4 give the temperature and the mean velocity history at the seven points. Figures 5 - 12 give the temperature shaded countors and velocity vector fields at several moments at the three planes prespectively.

1. **Transient property of the ventilation**

Figures 5 - 6 give the temperature and velocity histories at four movements (1.0 s, 5.0 s, 20.0 s and 100.0 s) at the plane of x= 1.5 m. This plane cross both diffusers and exhausts so that may show more information about the air dynamical properties of the ventilating system in the office. It is obviously shown that the cool air introduced to the office through the diffusers in the floor flows and accumulates with time elapsing, then disperses near the floor approaching the the walls. At the 1.0 s, the cool air flow near the diffusers are almost spherical symmetry. *i.e.*, the fresh air flow are evenly distributed at different directions. Such uniformity is not broken after 5.0 s. One reason is that the dispered cool air reaches the wall near the diffusers, another reason could be the bouyant force, which resists the fresh air rising because of its lower initial temperature, become weak as the time elapses and the temperature difference decreases after 1.0 s. The ascending motion of the supply air gradually grows up and final developes to weak jets. The abrupt variations of the temperature and velocity at point E in the Figures 3 - 4 are result from sudden passing through of the cool supply air. Figures 5 - 6 also show the process of the supply air flow climbs along the partition wall. The velocities history illustrated in Figure 4 shows that the velocities near the exhausts (points C and D) jump to specific values at time zero and then are kept fairly constant with slight flunctuations.

2. Transient thermal boundary near heated wall

The major part of the south wall is glazing so that a considerable quantity of solar radiative heat is transmittted. Figures 9 - 12 give the temperature and velocity profiles at the eight moments cross the plane y = 3.8 m, which illustrate the developing process of the thermal plume produced by the heated south wall. At 1.0 s, the thermal boundary layer near the wall is thin and heat conduction controlled (Figure 9 - 10), the flow is a tall vortex near the wall. The layer becames thicker and thicker with the time elapses, the vortex grows and move up and attaches the ceiling (1.0 s and 5.0 s), continues to flow through the ceiling far from the heat sources and reaches the partition wall and descends until meeting the cool fresh air (about 20.0 s). It is very interesting to see the obvious temperature abruptly transvered layer at 20.0 s which becames weaker and weaker after 20.0 s near the partition wall. When the time reaches to 100.0 s, the initial thermal vortex disappears and melts into the whole flow fields. It may be the possible reason for the temperature and velocity flunctuations at points C and D between 10 s and 40 s.

3. Transient thermal plume by hot objects

The thermal plume caused by a hot object in a ventilating room contribute significantly to the air distribution. Figures 7 - 8 show the temperature and velocity profiles at the plane x = 4.6 m cross the centreline of the first row of occupants and computers in the main part of the office, illustrate the process of the origin and the development of the thermal plumes induced by the occupants and computers. Similar with the transient process of the thermal boundary near the heated wall described above, the transient thermal plumes begin with thin layer nestles close to the hot objects, then the thin layger accumulates and move up successively and forms the plume above the hot objects. The initial plume rises, reaches and occupies the space right under ceiling. Whereas, the flow field around the hot objects initiate as local weak vortex and gradually expand to become a typical boundary layer formed by natural convection. Such flow pattern induces the fresh air near the floor to go up, improves the air quality around the occupants, which is evident in the temperature profile of points F and G near the computers (Figure 3). The velocity history of the two points are given in Figure 4, the thermal flow near the hot objects occur almost at the time zero. In this curves the velocity value at point F is much lower than that of G, because G is closer to the hot object with distance of 0.05 m than F with distance of 0.25 m.

4. Difference among the difference office rooms

The office is divided to three independent rooms by partition walls. With difference situations, the transient properties differ largely. The two individual offices near external wall have almost the same ventilating condition and geometry so their performances are almost identical (Figures 5 - 6). With less ventilating air flow per area, the open office is much slower in flow field development.

According to the disscusions at section 2 and 4, the natural convection play a significant role in the transient process and can not be neglected in the research of transient regime. Limited by computational power, only the transient ventilating process up to 200.0 s has been simulated. Generally, a typical pull-down process of the air condition system will last about half an hour to one hour. However, the real ventilated case is more complicated. There are at least several significant facts which are not considered in the present research. *i.e.*, the heat capacity of the wall, equipment and furniture, which is very important for the prediction of the air condition pull-down load (8). The radiation of the solid face may decrease the temperature difference between different walls and be beneficial to the ventilating process. Further research is therefore warrant.

Conclusions

The complex nature of the starting process of a building room cooled down by a displacement ventilated system is revealed by simulation. The cool fresh air supplied from the diffusers , disperse all over the floor, and then displace the air from underside of the office. The thermal plumes produced by hot walls and objects occur and grow all over the simulated time and contribute to the temperature and velocity distributions in the office. The initial velocity and temperature fields are thus established. The finding helps to understand transient behaviour of the conditioned objects such as time lag and time constant, therefore is useful in system design, especially cooling equipment sizing and control strategy design.

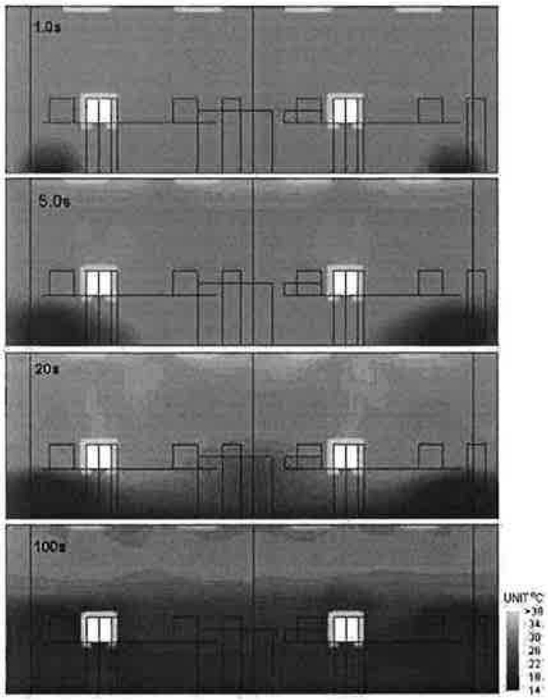

Figure 5. Evaluations of Temperature Profile at x = 1.5 m

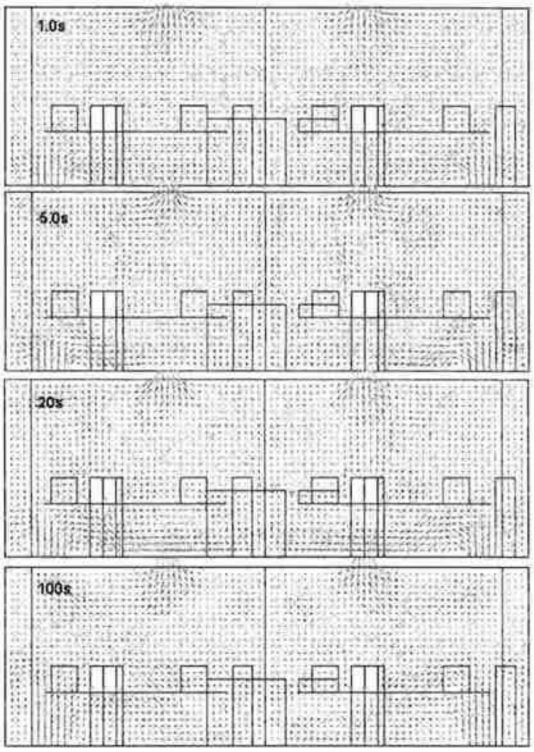

Figure 6. Evaluations of Velocity Field at x = 1.5 m

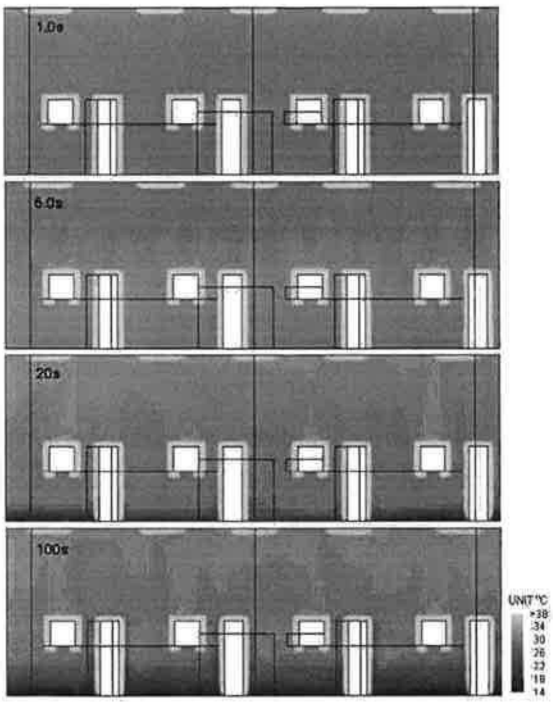

Figure 7. Evaluations of Temperature Profile at x = 4.6 m

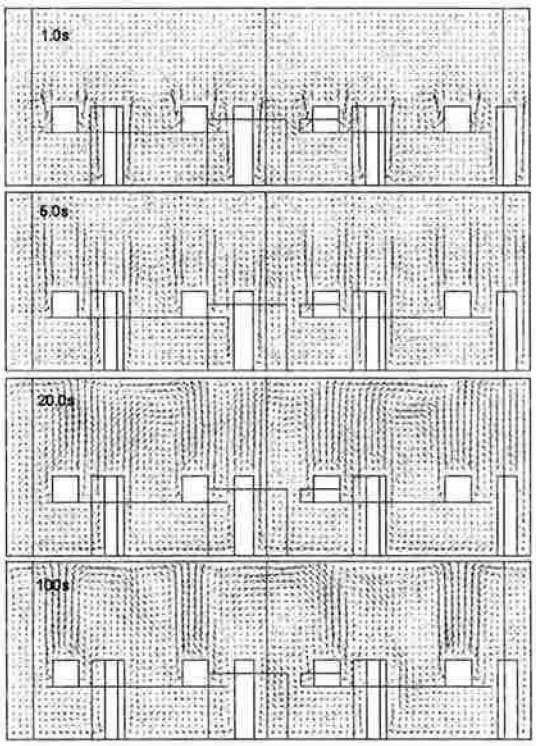

Figure 8. Evaluations of Velocity Field at x = 4.6 m

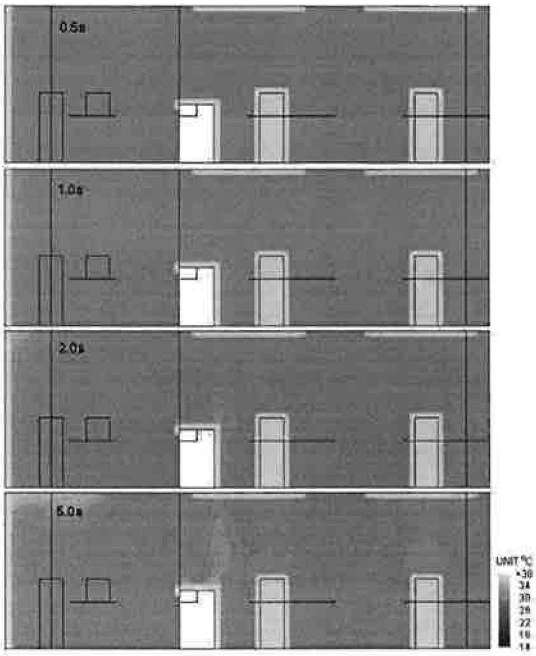

Figure 9. Evaluations of Temperature Profile at y = 3.8 m

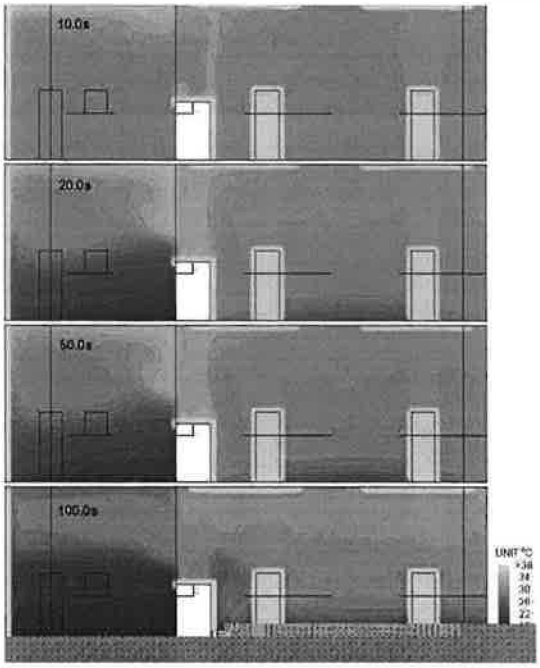

Figure 10. Evaluations of Temperature Profile at y = 3.8 m (continued)

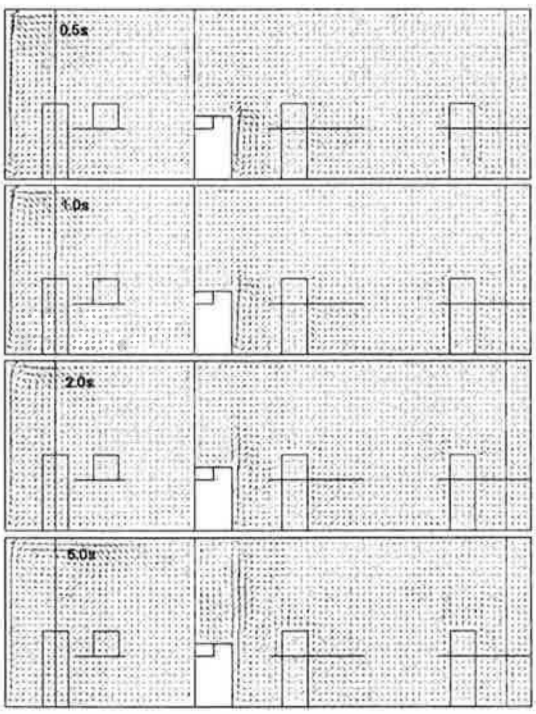

Figure 11. Evaluations of Velocity Field at y = 3.8 m

Figure 12. Evaluations of Velocity Field at y = 3.8 m (continued)

References

(1) Peng X., van Passen A.H.C. and Chen Q. A. "A type of calculation of indoor dynamic temperature distribution" Proceedings of System Simulation of Building Conference. Liegue. 1994

(2) Ghiaus, C. M. and Ghiaus, A. G. "Evaluation of the indoor temperature field using a given air velocity distribution" Building and Enviroment, Vol. 34, pp. 671-679. 1999

(3) Nannei, N. and Schenone, C. "Thermal transient in buildings: development and validation of a numerical model" Energy and Building. Vol. 29, pp. 209-215. 1998

(4) Lin, Z., Chow, T.T., Fong, K. F., Wang, Q.W. and Chen, Q. "CFD simulation of concentration of gaseous impurities in a typical Hong Kong office" The Proceedings of the 20th Congress of IIR. Sydney, pp. 19-24. 1999-1

(5) Lin, Z., Chow, T.T., Fong, K. F. and Chen, Q. "Review on application of displacement ventilation to Hong Kong buildings" Proceedings of the Fifth International Conference on Tall Buildings. Vol. I, pp. 246-251. 1998

(6) Yuan X., Chen Q. and Glicksman L. R. "Critical review of displacement ventilation" ASHRAE Transaction. Vol. 104, Part 1A. 1998

(7) Lin, Z., Chow T.T., Fong K.F., Wang Q.W. and Chen Q. "Validation of CFD model for research into application of displacement ventilation to Hong Kong buildings" The Proceedings of the 3rd International Symposium on HVAC. Shenzhen, Vol. 2, pp. 602-613. 1999-2

(8) Lin Z and Lau C M. "Prediction of pull-down load/time of buildings" Proceedings of Symposium on Tall Building Design and Construction Technology. Beijing, Vol. 2, pp. 146-154. 1999

(9) AEA Technology. "CFX-4.2 Enviroment User Manual" Vol. 1 to 4. 1997