EVALUATION OF SMOKE EXHAUSTION PERFORMANCE OF A VESTIBULE WITHIN TALL BUILDINGS IN TAIWAN

K.C. Chung
Department of Mechanical Engineering, National Yunlin University of Science and Technology
Touliu, Yunlin 640, Taiwan

ABSTRACT

Smoke is recognized as a major hazard to human life in building fires, especially in tall buildings. Stairshafts and vestibules frequently become smokelogged in case of fire, thus preventing evacuation and inhibiting rescue and firefighting. In order to improve the efficiency of smoke exhaustion in vestibules within tall buildings, the smoke distribution of a full-scale smoke exhaust chamber have been studied numerically and experimentally. A three-dimensional finite volume numerical model is used to simulate turbulent airflow and smoke distribution in the smoke exhaust chamber. The model described in this paper might be considered as a part of the performance-based design tools for smoke control of vestibules in tall buildings in Taiwan. Also, field measurements were conducted in this paper to validate the predicted results. The measurements concern internal airflow pattern, air velocity fields, and smoke distribution inside the smoke exhaust chamber. Under the same physical configurations, computed results agreed reasonably well with the measured data. However, there was a discrepancy between the computed and measured distribution of the smoke concentration. Possible reasons for the discrepancy are discussed.

1. INTRODUCTION

The objective of designing the smoke exhaust system for stairwell/elevator vestibule is to ensure that the life safety performance requirements can be met. There are some fundamental design rules of the smoke exhaust system for stairwell/elevator vestibule [1]. First, the smoke exhaust system is to be designed to maintain tenable conditions on escape routes (stairwell/elevator) throughout the period they are likely to be in use by occupants of the building. Then, the system is to be designed to maintain visibility for, and mitigate heat exposure to, trained fire fighters, thus allowing quicker and less hazardous attack on the fire. Finally, the system is to be designed to enable smoke to be cleared from a building after the fire has been brought under control [2].

Smoke is a hot buoyant gas. Basically, hot air plus contaminants. As such, it obeys to the fundamental laws of fluid mechanics. Several basic principles should be understood by designers of smoke control and extract systems. Recently, the successful development of numerical techniques to solve the basic equations of fluid mechanics has led to their extensive use in design and analysis. The use of mathematical models based on these techniques to predict the behavior of fire and smoke movement is not an exception, and a number of applications have been reported in the last few years [3].

In Taiwan area, the vestibule of stair/elevator is commonly designed as a smoke exhaust lobby with supply and exhaust systems in typical tall buildings as shown in Fig. 1. The smoke accumulated in the lobby can be exhausted through the exhaust vent by a mechanical ventilated system. On the other hand, the fresh air is supplied from the opposite side through the supply vent. Fig. 2 illustrates a section of smoke exhaust lobby in a tall building. In the event of a fire, stairshafts and vestibules are the primary escape routes in a tall building. Unfortunately, the lobby might become smoke-logged in a short time and obstruct occupants from evacuating safely. Therefore, some provisions are required to keep the smoke exhaust lobby tenable so that occupants are not trapped to smoke. To prevent smoke from entering the vestibule through extraneous wall leakage openings, the lobby can usually be pressurized with outdoor air using an air supply fan connected to a vertical air distribution shaft with a supply air outlet in each lobby.

Stairshaft pressurization systems under fire conditions have been studied by many investigators [4-7]. Two hypothetical buildings with different leakage characteristics were used to study the effect of the opening of stairwell doors on a top-injection stairwell pressurization system by Yuill and Haddad [8]. The published leakage data can be used to form two leakage combinations for the same building that lead to different design decisions concerning the stairwell pressurization system. Tamura [9] reported the performance of three vestibule pressurization systems (a) a stair vestibule pressurization (VP) system alone, (b) a stair vestibule pressurization system in combination with stairshaft pressurization (VPSP), and (c) a stair vestibule pressurization system in combination with mechanical exhaust of the fire floor (VPME). In order to evaluate the performance of such pressurized vestibule system, tests were also conducted in the experimental fire tower under fire conditions with open stair and vestibule doors. When all stair and vestibule doors were closed, the above three smoke control systems prevented smoke contamination of the stairshaft. Except for VPME with mechanical exhaust of the fire floor, vertical shafts other than the stairshaft and nonfire floor spaces were contaminated with smoke. However, the current building technique code of Taiwan does not require a pressurized system in a vestibule of a tall building. Contrarily, a forcible exhaust air at the rate 4 m^3/s was demanded for vestibule by the code. It appears that this specification is not good enough, as the geometry of the vestibule space is not considered. Therefore, it is necessary to investigate the performance of supply/exhaust mechanical ventilation system of vestibule.

This research is carried out by experimental and numerical methods to study the temperature distribution inside smoke exhaust duct as well as the smoke removal effectiveness in a smoke exhaust lobby. A full-scale ventilation chamber was adopted for field measurements. The measured data are compared with the predicted results for the same physical conditions. A three-dimensional k-ε turbulent model was used as the simulation tool. Our aim, in this paper, is to study airflow, velocity field and smoke distribution in those escape routes in order to find reliable method to control the smoke concentration under safety requirement and build up technical procedure to evaluate performance-based designs. Also, this research is to assess the agreement between the measurements and the computations using an airflow numerical model.

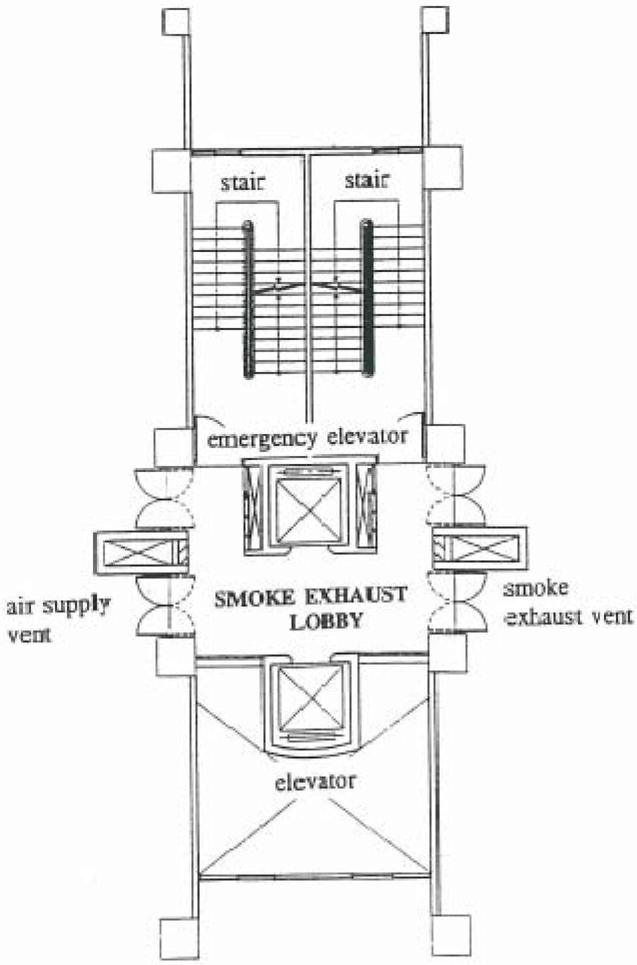

Fig. 1: A typical smoke exhaust lobby design of tall buildings in Taiwan area

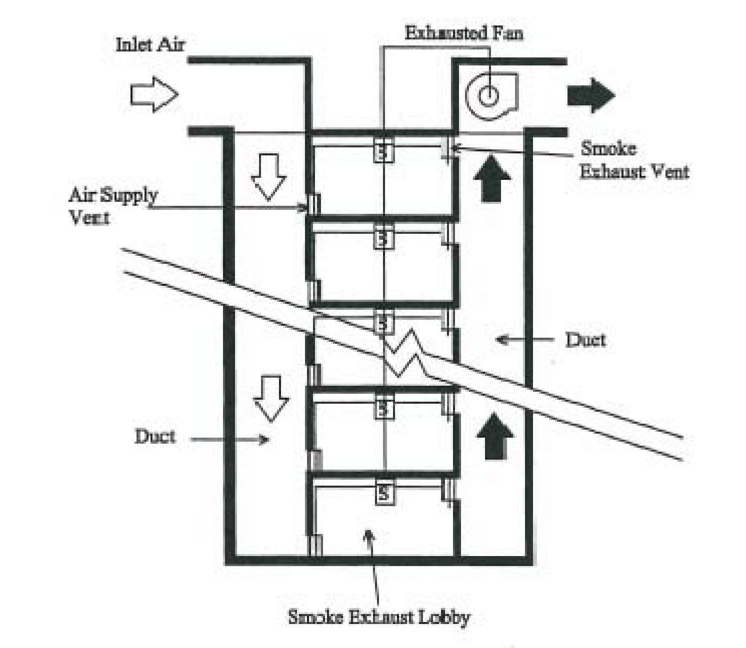

Fig. 2: Description of the mechanical ventilated smoke control system in a tall building with air supply and smoke exhaust vents

2. COMPUTATIONAL MODEL

The successful development of numerical techniques to solve the basic equations of fluid mechanics has led to their extensive use for design and analysis. The use of mathematical models based on these techniques to predict the behavior of fire and smoke movement is not an exception, and a number of applications have been reported in the last few years [10-12]. The flow of smoke is considered to be a turbulent flow, which is simulated by a three-dimensional k-ε two-equation turbulence model. With the Boussinesq approximation, the conservation equations of mass, momentum, energy, and the turbulent properties can be written in the following form:

Continuity equation:

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (1)$$

Momentum equation:

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial(p + 2\rho k/3)}{\partial x_j} + \frac{\partial}{\partial x_j}\left[(\nu + \nu_t)(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j})\right] - \beta g_i \theta \quad (2)$$

Energy Equation:

$$\frac{\partial \theta}{\partial t} + \frac{\partial \theta u_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(k + \frac{\nu_t}{\sigma_\theta})\frac{\partial \theta}{\partial x_j}\right] + h(x,t) \quad (3)$$

Equation for k:

$$\frac{Dk}{Dt} = \frac{\partial}{\partial x_j}\left[(\nu + \frac{\nu_t}{\sigma_k})\frac{\partial k}{\partial x_j}\right] + \nu_t(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j})\frac{\partial u_i}{\partial x_j} + \beta g_i \frac{\nu_t \partial \theta}{\sigma_\theta \partial x_j} - \varepsilon \quad (4)$$

Equation for ε:

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j}\left[(\nu + \frac{\nu_t}{\sigma_k})\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left[C_1 \nu_t(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j})\frac{\partial u_i}{\partial x_j} - C_2 \varepsilon + C_3 \beta g_i \frac{\nu_t \partial \theta}{\sigma_\varepsilon \partial x_j}\right] \quad (5)$$

where $\nu_t = C_D k^2 / \varepsilon$. The empirical constants for the above equations are $C_D = 0.09$, $C_1 = 1.44$, $C_2 = 1.92$, $C_3 = 1.0$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, and $\sigma_\theta = 0.9$.

Smoke concentration equation:

$$\frac{\partial C}{\partial t} + \frac{\partial C u_j}{\partial x_j} + \frac{\partial C W_p}{\partial x_p} = \frac{\partial}{\partial x_j}\left[\frac{\nu_t}{\sigma_p}\frac{\partial C}{\partial x_j}\right] + C_s \quad (6)$$

The SIMPLE algorithm [13] is employed to solve the set of differential equations. The false time step and under-relaxation are used to avoid the divergence. Overall continuity corrections for velocity and pressure at the section of opening are added to assure the mass balance across the opening converse. The flow properties at the grid nodes near solid surface are computed through the wall function method [14]. The k-ε model used in this paper is relatively computationally efficient and stable compared with the more complicated Reynolds stress models, yet it is also reasonably accurate for a wide range of turbulent flows [15]. The transport equation for concentration C is used to simulate the smoke concentration distribution.

The configurations and specifications of the model smoke exhaust lobby used for simulation were depicted in Fig. 3. The three-dimensional airflow properties were analyzed on the basis of the k-ε model. The constant boundary temperatures are assumed in all the calculations. The complete temperature distributions of smoke exhaust dust was studied first with a mesh system of 40 (X) x 30 (Y) x 25 (Z) in a smoke exhaust chamber. Then, the analyses of the airflow and diffusion fields were performed. All the physical sizes and dimensions used in simulation are the same as those used in experiments. Total of 33046 (41 x 31 x 26) grid points are calculated at every 0.1 second.

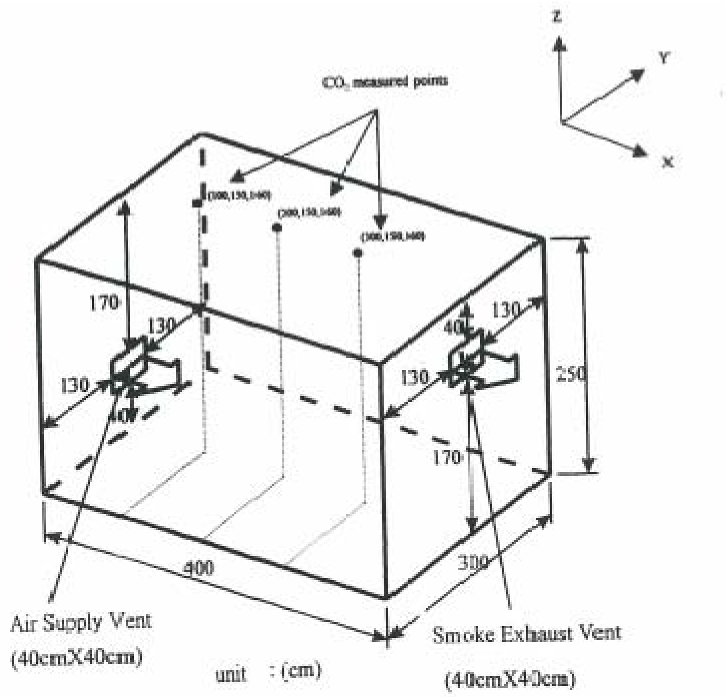

Fig. 3: Model smoke exhaust chamber used for full-scale experiments and simulations

3. FULL-SCALE MODEL EXPERIMENTS

In order to validate the computer simulation results of the smoke exhaust lobby analysis, a full-scale smoke exhaust chamber with mechanical ventilation system was studied experimentally. A test was carried out in a model facility at a university laboratory. The space is 4 m long, 3 m wide and 2.5 m high with one exhausted fan on the right-hand side wall as shown in Fig. 3. Both inlet and outlet diffusers that are 40 cm high and 40 cm wide were located on the left-hand as well as right-hand side wall. Four different air change rates (40, 60, 80, and 100 ACH) were used to evaluate the smoke exhaustion efficiency of the smoke exhausted chamber. The points of air velocities and CO_2 concentrations measurement are located evenly inside the test chamber. This test smoke exhaust chamber was constructed under an outer room for controlling the ambient environment conditions. The test chamber measures 4 m long by 3 m wide by 2.4 m high with a mechanical exhausted fan on the right-hand side wall.

The concentration decay method with CO_2 tracer gases was employed for simulating the smoke exhausted from the chamber. Three measuring points of tracer gas concentrations were located at (x = 1m, y = 1.5m, z = 1.6m), (x = 2m, y = 1.5m, z = 1.6m), and (x= 3m, y = 1.5m, z = 1.6m). The concentration of the CO_2 were measured at every

20 seconds by a CO_2 monitoring systems (VAISALA CO_2 monitor), with uncertainty within ± 3 %. During the experimental series the conditions were kept approximately the same for every run. First, the CO_2 tracer gas was injected into the chamber. The diffusers were sealed during the injection process. After the concentration of tracer gas reached an appropriate level (approximately 2000 ppm), the injection was stopped. Within about 10 minutes, indoor air was mixed with the tracer gas by the electric fan. After "perfect" mixing of indoor air was achieved, the seals were stripped off the openings and the measurement of concentration decay was started. The duration of concentration sampling was five minutes.

4. RESULTS AND DISCUSSION

In Taiwan, the fire safety engineers develop design solutions from the specifications which are given in "prescriptive" code (regulations). The traditional codes are referred as a prescriptive code because they set up specific prescriptive requirements for building design. For example, the code required a minimum air volume (4 m^3/sec) in the lobby when using mechanical exhaustion systems. It does not consider about the volume factor of lobby. The bigger lobby, a higher smoke exhaustion rate shall be provided. Those requirements result from the past experience and have been proved to be effective to fire prevention of buildings. However, as buildings are different from one to another, prescriptive requirements tend to be lack of flexibility. Performance-based design is the new design approach that is established on engineering principles and methodologies. It applies the principles of fire science and engineering to fire safety design and develops design solutions through quantitative engineering analyses and calculations. Performance-based design provides the designers with a great extent of design flexibility and enable the design to develop fire safety solutions which best meet the requirements of individual buildings. Performance-based code is developed from the principle and methodology of performance-based design. Therefore, it is essential to promote research in fire science and fire safety engineering, to build up fundamental database, to develop engineering analysis models and methods including reliable computer code. The smoke exhaustion computational model described in this paper is one of the possible methods to evaluate the smoke exhaustion performance of vestibules within most buildings in Taiwan area.

Computation of the temperature gradient of the smoke exhaust duct and the smoke distribution of the exhaust lobby was conducted on a UNIX computer, which took about 9.6 and 12.4 CPU hours, respectively. The maximum residual in the mass conservation was less than 1 x 10^{-3}.

In order to evaluate the predictability of the numerical model, the simulation process used exactly the same initial and boundary physical data as in the experiments. The temperature distribution of the smoke exhaust lobby including the 30 m high and 2 m wide smoke exhaust duct was simulated to examine the effect of hot smoke flow transported inside the duct. The worst situation of fire case, that smoke was produced from the ground floor, was considered in this simulation. For the quantitative analysis, the temperatures of the smoke exhaust duct and the lobby are assumed to be remained 300 K and 600 K, respectively. The effect of fresh air induced by smoke is ignored. The exhaust velocity of hot smoke flow leaving lobby is 4 m/sec through the exhausted vent which is located at 1.2 m under the ceiling. Fig. 4 presents the temperature gradient of smoke exhaust duct in transient situations at 10, 30, 60 and 90 sec. The time interval of calculation is 0.1 second and records the data every 10 seconds. These temperature contours imply the noticeable buoyancy force and the rapid rise above the fire source. For evaluating the effect of building height on temperature and transport of smoke inside the exhaust duct, three different height levels: ground, fourth, and eighth floor, are chosen as the location of the fire source to calculate the temperature distributions. A detail comparable data are shown in Fig. 5. It is obviously found that smoke produced from higher level is more quickly exhausted by the fan. The smoke produced from the eighth floor was exhausted to outside about 12 seconds. On the other hand, the fire source located at the ground floor, the smoke needed more than 90 seconds to exhaust through the duct.

Although the smoke exhaust time is able to assist to evaluate the buoyancy force, it is not an essential considerate parameter for designing a smoke exhaust system. Whether the exhaust system can offer adequate air volume to replace the smoke within the lobby shall be considerate first priority. Even though, the smoke can be exhausted through the duct very quickly, the high toxic smoke concentration (mainly carbon monoxide) in the lobby still can cause death for the majority of fire victims. Therefore, the air change per hour (ACH) shall be studied to understand the smoke removal effectiveness. A x-y plane of velocity fields were plotted at z = 1.5 m. Fig. 6 demonstrates that the simulation results of the smoke exhaust chamber is a complex three-dimensional flow field. Some small recirculated flows were found near the diffuser as shown in Fig. 6.

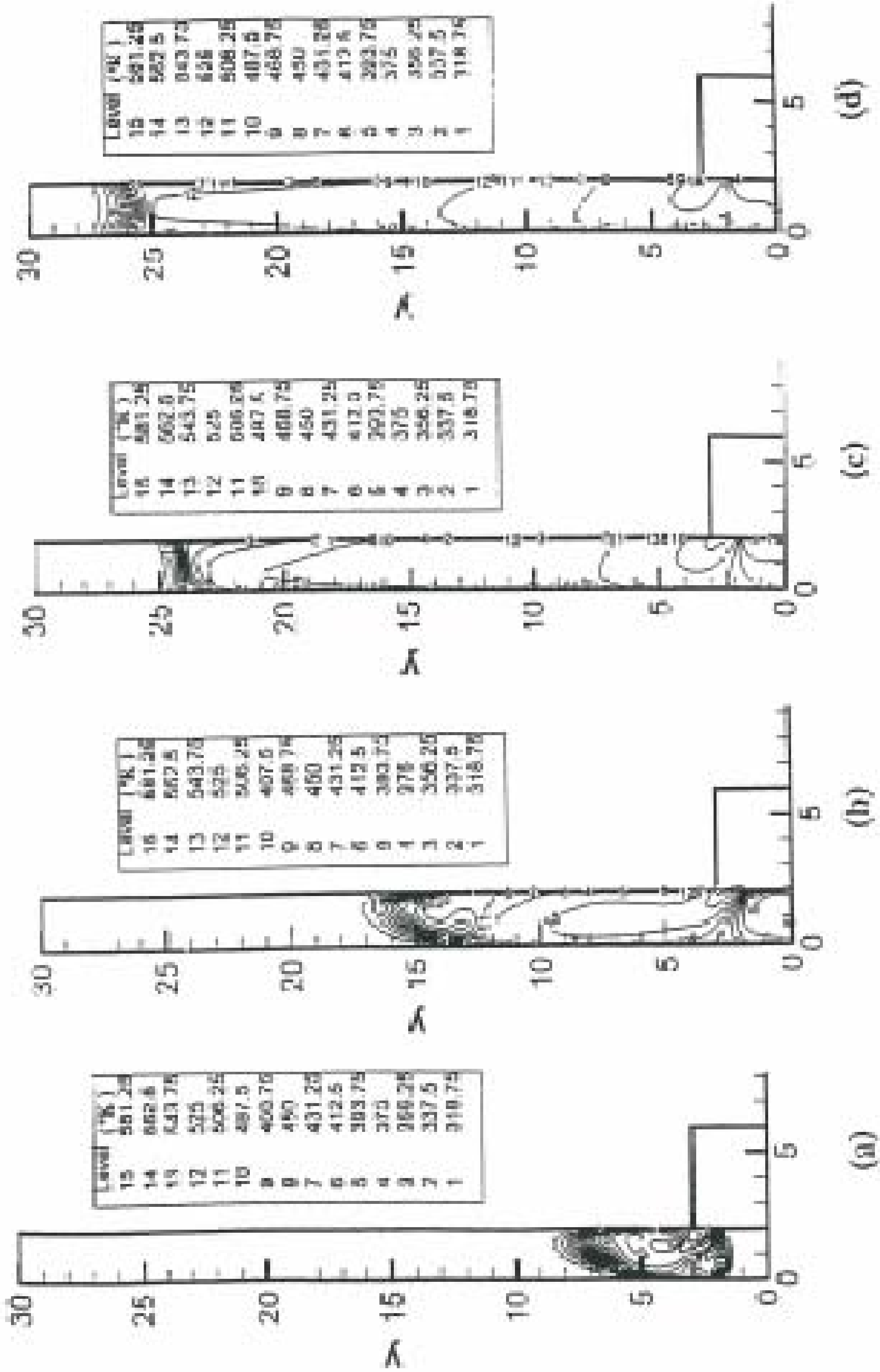

Fig. 4: Predicted temperature gradient in smoke exhaust duct at different time when the smoke occurred at the ground floor

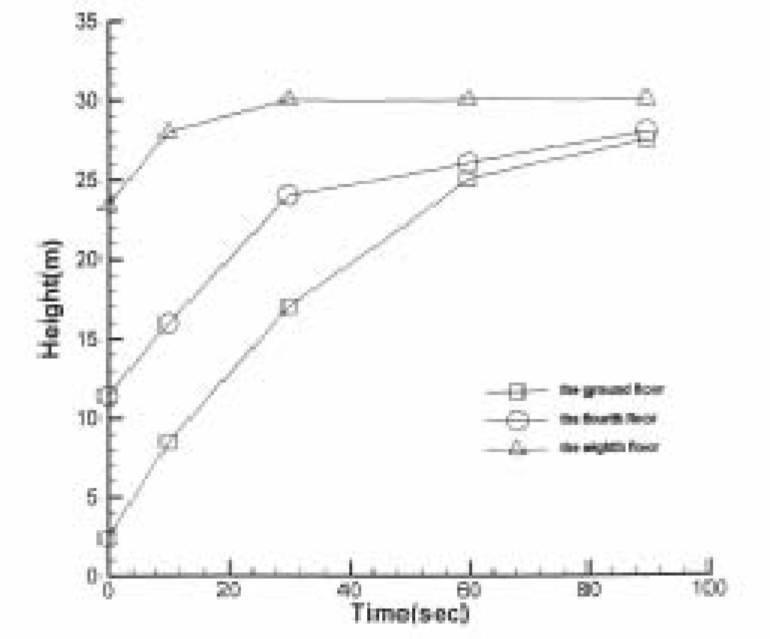

Fig. 5: The effect of floor level on smoke temperature transportation inside a smoke exhaust duct

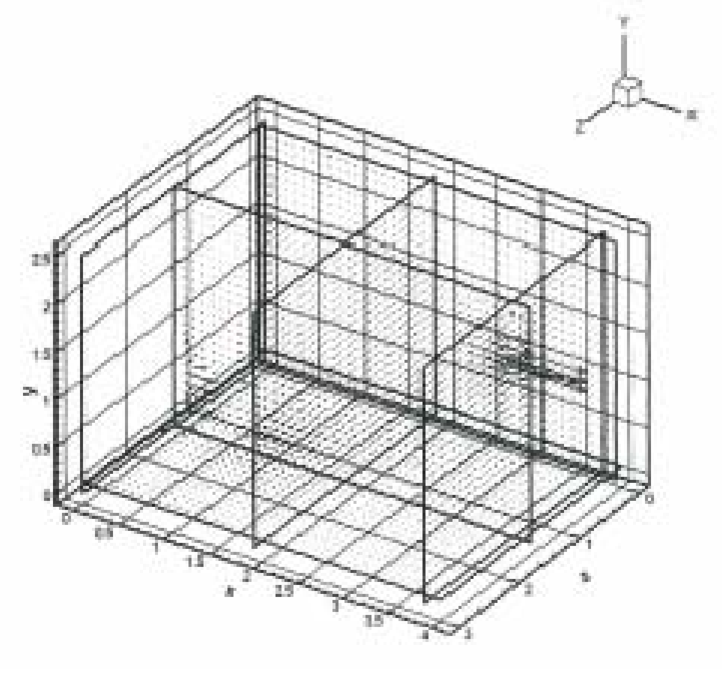

Fig. 6: Three-dimensional computed airflow patterns of smoke exhaust chamber at z = 1.5m for x-y plane

In Fig. 7, the relative smoke concentrations of the smoke exhaust chamber were demonstrated at z = 1.5 m for x-y plane. The relative concentrations are the calculated concentration values of every time step divided by the initial concentration. It can be found that the smoke was exhausted very quickly by the fan at 60 ACH. During 5 minutes (300 seconds) the smoke relative concentration can decrease approximately from 0.5 to 0.1. One of the advantages of a field model simulation is that information is obtained at every computed points in the flow domain. It enables to assess whether smoke will block the evacuation and inhibit rescue in smoke exhausted lobby or not. Therefore it is easy to apply the simulation results to designing the escape route. For example, at the upper corner of left-hand side in Fig. 7, some accumulated smoke layer can be discovered. Hence, the EXIT signs shall not be installed in such locations.

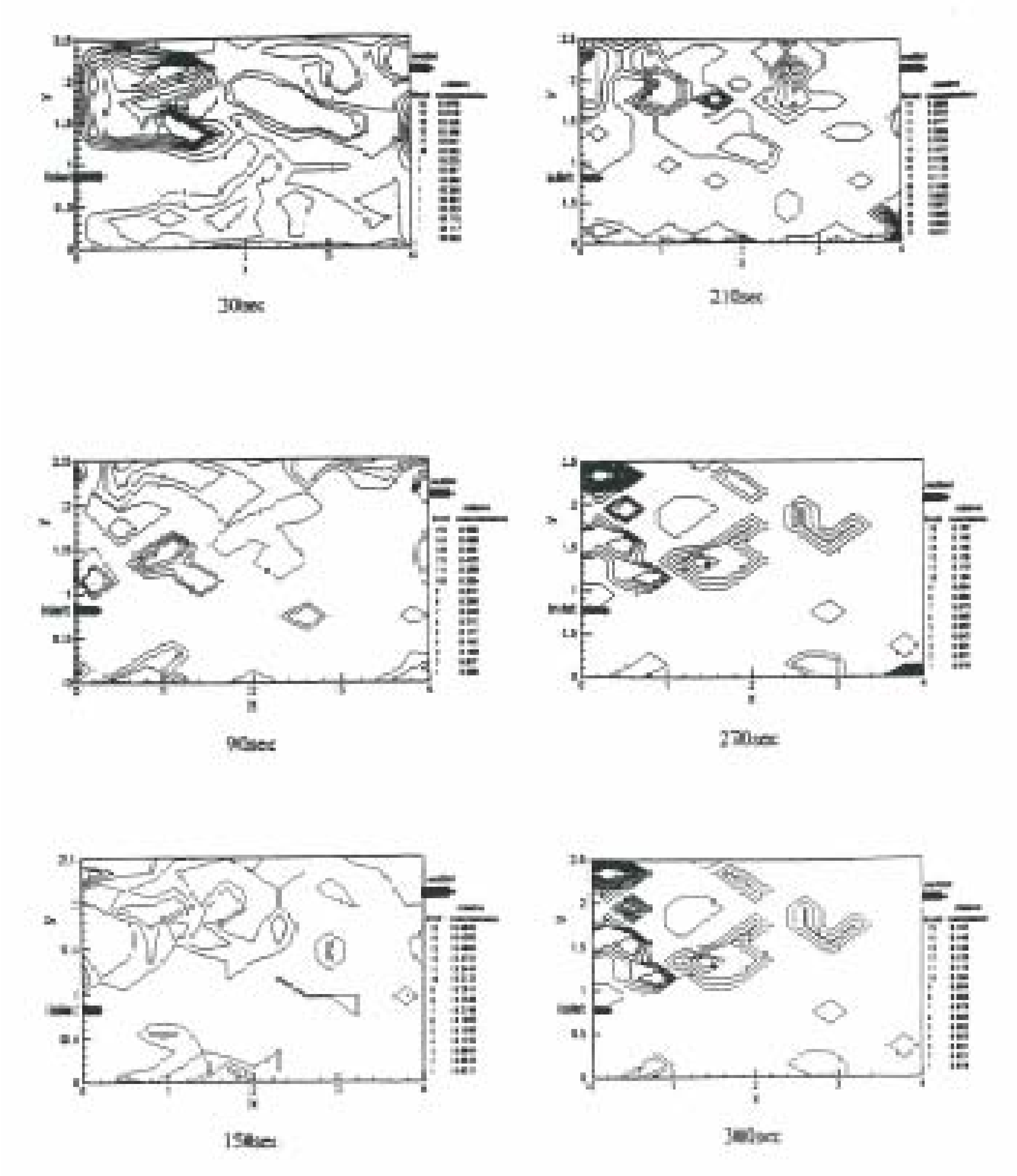

Fig. 7: Computed smoke concentration contours of smoke exhaust chamber at z = 1.5 m for x-y plane with different time

Fig. 8 shows a comparison between the predicted and experimental results of CO_2 relative concentration in the smoke exhaust lobby. The agreement between the computed and measured temperature distributions is reasonably good. However, some discrepancy can be found between the predicted and measured temperature values in Fig. 8. The average deviation is within ± 17% and the maximum deviation is approximately ± 23% for all the sampling points. Quantitatively, the discrepancies between computed and measured CO_2 relative concentration are thought to be due to that during the experimental process the CO_2 concentration could not be made homogeneous while the initial values of them were assumed to be constant in the computations. In addition, a slight difference in the measuring location can cause a significant error.

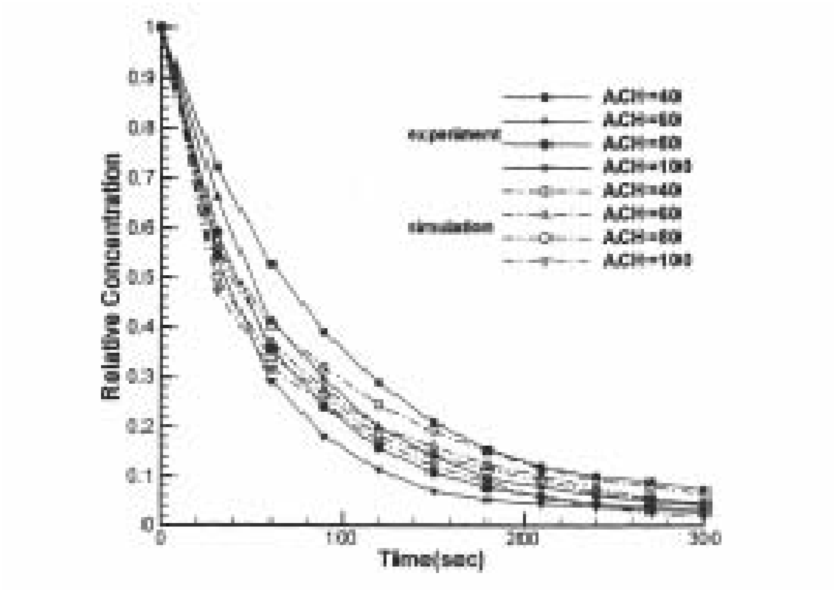

Fig. 8: Comparison of measured and predicted relative concentration of smoke exhaust chamber at 60 ACH

5. CONCLUSIONS

There is no doubt that exhaust ventilation system is one of the most effective tools available to the ventilation engineers for controlling the smoke movement in buildings. In order to understand the effect of air change rate on smoke exhaust system of smoke exhaust lobby, the smoke ventilation system of smoke exhaust chamber was investigated in this paper.

From the simulation results, eighty to one hundred ACH smoke exhaustion rate might be enough for a lobby with 30 m^3 (4m x 3m x 2.5m) space volume. The exhaust volume rate was about 0.67 to 0.83 m^3/sec which is much smaller than the value of existing code required (4 m^3 /sec). When fire engineers design a smoke control system for vestibules, the complete computational model and results in this paper can offer a basic evaluation procedure for smoke exhaustion performance of vestibule in a building. As the performance-based design is the mainstream of future development in fire safety design for buildings, the techniques used in this paper might be considered as a part of performance-based design tools for smoke control systems in Taiwan.

A field measurement was conducted at a full-scale environmental control laboratory to obtain validated data for simulation results. The same physical parameters for the smoke exhaust chamber were used as the initial and boundary conditions of a three-dimensional k-ε turbulent simulation model. The airflows and field conditions of real smoke exhaust lobby studied in this paper were usually not as stable as those under well-controlled laboratory conditions. However, the computed results of smoke concentrations and distributions agreed reasonably well with the measured data for full-scale environmental control laboratory. The average deviation of relative concentration of CO_2 was within 17 % approximately. The discrepancies are believed to be due to that the fluctuation of the CO_2 concentration was not exactly homogenously distributed inside the test smoke exhaust chamber. The results presented in this paper indicated that computation fluid dynamics techniques could be powerful tools for evaluating the required air change rate of smoke exhausted lobby to secure the public safety. Also, with the use of this predicting model, it is possible for a ventilation engineer to predict the influence of different design parameters on the smoke exhaust efficiency at a very early stage in the design of a proposed high-rise building.

NOMENCLATURE

Symbols

C	smoke concentration
C_1	empirical constant
C_2	empirical constant
C_3	empirical constant
C_D	empirical constant
C_S	source term of smoke concentration
g_I	gravitational acceleration in x_i direction
h	mean volume heat source generation rate
p	mean static pressure difference
u_I	mean velocity component in x_i direction
t	time
x_I	cartesian coordinates
W_p	settling velocity of smoke particle
β	volumetric coefficient of expansion
ρ	density of air
ε	dissipation rate of turbulence energy
k	turbulent kinetic energy
θ	mean temperature difference
σ	turbulent Prandtl number
v	kinematic viscosity of air
v_t	eddy viscosity

Subscript

p	smoke particle

REFERENCES

1. J.H. Klote and J.A. Milke, "Design of smoke management systems", ASHRAE (1992).
2. F.B. Clarke, "Physiological effects of smoke: Managing escape", ASHRAE Journal, March, pp. 47-56 (1997).
3. N. Rhodes, "Prediction of smoke movement:An overview of field models", ASHRAE Transactions, Vol. 95 (CH-89-12-4), pp. 868-877 (1989).
4. C.Y. Shaw, "Methods for conducting small-scale pressurization tests and air leakage data of multi-story apartment buildings", ASHRAE Transactions, Vol. 96, No. 1, pp. 241-250 (1990).
5. J.H. Klote, "Stairwell pressurization", ASHRAE Transactions, Vol. 86, No. 1, pp. 604-623 (1980).
6. G.T. Tamura and C.Y. Shaw, "Experimental studies of mechanical venting for smoke control in tall buildings", ASHRAE Transactions, Vol. 84, No. 1, pp. 54-71 (1978).
7. G.T. Tamura, "Assessment of stair pressurization systems for smoke control", ASHRAE Transactions, Vol. 98, No. 1, pp. 66-72 (1992).
8. G.K. Yuill and K.H. Haddad, "Effect of opening stairwell doors on the performance of a stairshaft pressurization system", ASHRAE Transactions, Vol. 100, (NO-94-12-2), pp. 990-999 (1994).
9. G.T. Tamura, "Fire tower tests on vestibule pressurization for protection of stairshafts", ASHRAE Transactions, Vol. 100 (NO-94-12-1), pp. 981-989 (1994).
10. W.K. Chow, "Prediction of fire environment in apartment using zone models", Journal of Fire Sciences, Vol. 14, No. 4, pp. 263-312 (1996).
11. S. Nam and R.G. Bill, "Numerical simulation of thermal plumes", Fire Safety Journal, Vol. 21, pp. 231-256 (1993).
12. W.K. Chow, "Studies on the stability of thermal stratified layer in a forced-ventilation fire using Computational Fluid Dynamics", Journal of Applied Fire Science, Vol. 6, No. 1, pp. 15-25 (1997).
13. S. Patankar, Numerical heat transfer and fluid flow, Hemisphere Publishing Corp., New York (1980).
14. K.D. Knappmiller and A.T. Kirkpatrick, "Computational determination of the behavior of a cold air ceiling jet in a room with a plume", ASHRAE Transactions, Vol. 100, pp. 677-684 (1994).
15. M.A. Yaghoubi, K.D. Knappmiller and A.T. Kirkpatrick, "Three-dimensional numerical simulation of air contamination dispersal in a room", ASHRAE Transactions, Vol. 101, pp. 1031-1040 (1995).