A CFD STUDY OF WIND-INDUCED FLOW OF REFUGE FLOOR DESIGNS USING CENTRED SERVICE CORE SHAPES

C.C.K. Cheng
Department of Building and Construction, City University of Hong Kong, Hong Kong, China

B.J. Williamson
School of Design, Faculty of Built Environment and Engineering
Queensland University of Technology, Australia

D. Chen
College of Water Resources and Hydropower Engineering, Wuhan University, P.R. China

ABSTRACT

Wind-induced natural ventilation is essential for the refuge floor design of high rise buildings in Hong Kong. Cross ventilation is achieved by two sides of the refuge floor being open to the outside. The wind flow on the refuge floor will be influenced by the presence of a large service core. The Hong Kong open refuge floor concept occupying at least 50% of a floor is unique and is therefore vastly different from the limited refuge area proposed in the US during the 1980's. This paper presents a computational fluid dynamic study of the refuge floor design in relation to wind-induced ventilation flow. The study also considers the effect of different wind angles when using square and circular shaped service cores on the refuge floor.

KEYWORDS: Passive building construction, High-rise building design, Refuge floor, Safety codes

INTRODUCTION

In Hong Kong a refuge floor is an essential passive building element in high-rise buildings taller than 25-storeys. The design of this refuge floor has to comply with the Building Codes of Hong Kong[1] that based its refuge floor requirements on the safety zone concept proposed by Conway[2] in the 1970's. During the 1980's a number of refuge design proposals[3] were advocated for high rise buildings. In the United States of America the refuge floor was not accepted and instead a limited refuge area in the protected escape route[4] was adopted by American high-rise buildings. This refuge area (not total floor) approach was also accepted in the Building Codes in a number of countries such as Canada[5] and Singapore[6] in the 1980's. Although refuge floors occupying an entire floor were accepted in the People's Republic of China[7] during the 1990's, the refuge floor design considered in this study is the fully separated floor area adopted in Hong Kong since 1996.

The Building Codes of Hong Kong require natural wind ventilation of the refuge floor area by having at least two sides of this floor open to the outside. The resulting natural cross ventilation is intended to remove any smoke entering the refuge spaces and thereby avoid any smoke logging hazard on that floor. Therefore, wind-induced ventilation is an essential requirement for the Hong Kong refuge floor area. Although not required in other countries, the refuge floor in Hong Kong is required to be totally separated from the remaining spaces of the building and it cannot be an integrated part of a mechanical floor. Under this situation, fully separated service cores have become a common feature of high rise Hong Kong buildings. These cores with enclosed lifts and distributing service ducts for the buildings are generally located at the centre of the floor plan of the buildings. This feature is allowed to occupy 50% of the refuge floor area and this obviously has a significant influence on the wind ventilation on the floor due to the location of the blunt body of the core. Unfortunately insufficient studies were undertaken in the past to be certain that the existing refuge floor design will provide at all times the required fire safety for occupants using these refuge floors in high rise buildings.

EARLY INVESTIGATIONS

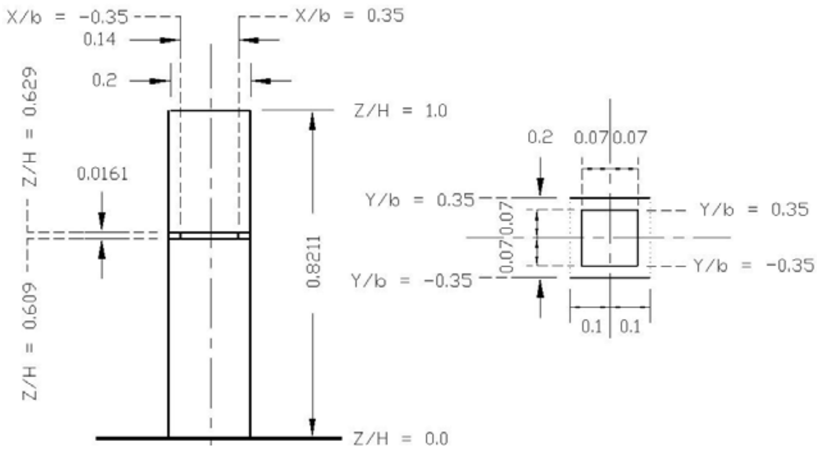
Due to the inherent difficulties associated with physical experiments and tests, refuge floor design was instead studied using CFD methods in the 1990's by Yuen *et al.*[8] and Lo *et al.*[9]. They studied the ventilation at normal incident to a section of a building in which two opposite sides of a mid-level refuge floor were opened. In the simulation results, it could be observed that smoke dispersed from a fire at the level immediately below the front wall opening of the refuge floor followed the wind flow stream entering the refuge floor and persistently logged around the main services core when the flow speed was slow. Similar results were also observed in Lo *et al.*'s CFD simulations[9] in which the ventilation was a boundary layer type of flow produced at the flow domain inlet according to the power equation and blowing at normal incident to the building with a refuge floor located at its mid-height. Other CFD wind studies were performed by Lu *et al.*[10-12] and these simulations showed that if there was a small core at the middle of the refuge floor covering 11% of the floor area, the prevailing wind would remove the smoke by the openings in opposite sides of the building. However, it was found that if the refuge floor had four or five small core blocks (in total covering 23% of the floor area), then smoke would become trapped on the refuge floor and thereby would not comply with the refuge floor requirements of Hong Kong. However, these early investigations did not examine the effect of one large service core in the middle of the floor which is the common design feature of high rise buildings in Hong Kong.

Although an improved CFD technique to produce wind effect around the high-rise building was demonstrated in these early studies, there were extra difficulties in performing parametric studies due to the extensive demand in computing effort involved and this produced problems for further investigations. In addition, these early studies did not have access to fundamental scientific data to validate the predictions obtained. In view of these problems, a CFD technique was developed by Cheng *et al.*[13-14] to establish the fundamental wind patterns around the building and inside the refuge floor and these results were validated by the wind tunnel data obtained during the development of the technique[15]. The models tested in the CFD experiment were based on the physical models tested in the wind tunnel experiment.

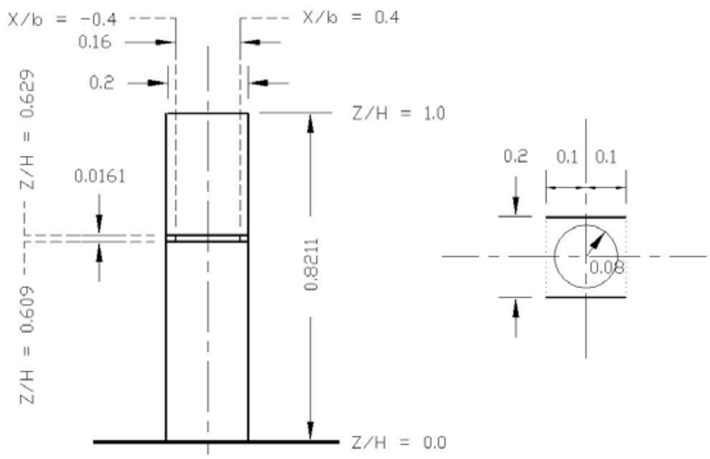
MATERIALS AND METHODS

Tests of two building models (Fig. 1) in the closed-circuit type re-circulation boundary layer wind tunnel are simulated in this study. The CFD wind tunnel model is based on the physical wind tunnel located at the Civil Engineering Department of the University of Hong Kong. The time-mean wind field data around the high-rise building and inside the refuge floor was computed in the simulations. The computational flow domain was digitized by unstructured tetrahedron elements. The width and height of the flow domain were set equal to the corresponding physical dimensions of the wind tunnel test section of 3 m and 1.7 m, respectively. The basic length of computational tunnel contains one times the building height (H) in front and 6 H behind the building model.

The buildings tested in the wind tunnel were generic buildings in square-planned form representing 40-storeys high-rise buildings scaled down by 150:1. The aspect ratio of the building according to its H to its base length (b) was 4:1. A semi-permeable refuge floor with two opposite sides opened was located at the building mid-height. The clear ceiling height of the refuge floor was equal to 0.02 H. In the middle of the refuge floor, there was a large service core occupying 50% of the floor area. The two models had a different shape for the service core; one being square in plan (Fig. 1(a)) and other circular in plan (Fig. 1(b)).



(a)



(b)

FIGURE 1. High-rise building models: (a) A large square-planned main services core at the centre of the refuge floor, and (b) A large circular based main service core at the centre of the refuge floor

Computed velocities data was used to depict the patterns of wind on the horizontal symmetry plane of the refuge floor (Z/H = 0.615) with the wind blowing at three selected directions to the building; namely: stream-wise (normal, 0°), diagonally (45°), and perpendicular (90°). The patterns showed in the presented figures are formed by the distribution of velocity vectors which were generated by normalized velocity components U and V according to the reference velocity U_0 computed at the building height on the domain inlet plane (U/U_0, and V/U_0), and the distribution of wind pressure is reflected by contour lines of pressure coefficient, C_p, ($\frac{P - P_0}{\frac{1}{2}\,\rho\,(U_0)^2}$). An intensive analysis on the simulation results was performed to determine the behaviour of the wind induced flow on the refuge floor according to the different shapes of the services core. This information provided an important basis for an evaluation of the implications on the movement of smoke entering the refuge floor and the safety design aspects of the refuge floor.

Boundary Conditions

In the simulations, calibrated empirical equations of velocity and intensity profiles are applied as the boundary conditions of domain inlet and outlet.

At the inlet

i. Cartesian normal velocity component U

$$U = U_h \left(\frac{Z}{H_h}\right)^{0.19} \quad where\ U_h = 10.478\, m/s \quad and \quad H_h = 0.85\, m \qquad [1]$$

ii. Turbulent intensity

$$I = I_h \left(\frac{Z}{H_h}\right)^{-0.47} \quad where\ I_h = 0.088 \quad and \quad H_h = 0.85\, m \qquad [2]$$

iii. Cartesian transverse velocity components V and vertical velocity components W are taken to be 0.

At the outlet

i. Relative static pressure is taken to be 0, and

ii. Turbulent intensity follows the profile described by Equation 2.

Governing Equations

Turbulence of wind flow in the simulations is modelled by two equations k-ε model[17] which offer a good compromise between numerical effort and computational accuracy. The model use the gradient diffusion hypothesis to relate the Reynolds stresses to the mean velocity gradients and the turbulent viscosity. The turbulent viscosity is modelled as the product of a turbulent velocity and turbulent length scale. While the turbulence velocity scale is calculated from solving the transportation equation for turbulent kinetic energy, the turbulent length scale is estimated from solving the transportation equations for the turbulent kinetic energy and its dissipation rate. The governing equations[18] can be written as listed in the followings:

Continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \boldsymbol{U}) = 0 \qquad [3]$$

Momentum transportation equation

$$\frac{\partial \rho \boldsymbol{U}}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} \otimes \boldsymbol{U}) - \nabla \bullet (\mu_{eff} \nabla \boldsymbol{U}) = \nabla p' + \nabla \bullet (\mu_{eff} \nabla \boldsymbol{U})^T + B$$

$$where\ p' = p + \frac{2}{3} \rho k, \ \mu_{eff} = \mu + \mu_t, \text{ and } \mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \qquad [4]$$

Turbulent kinetic energy, *k*, transportation equation

$$\frac{\partial (\rho k)}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} k) = \nabla \bullet \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \rho \varepsilon \qquad [5]$$

Turbulence dissipation rate, *ε*, transportation equation

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} \varepsilon) = \nabla \bullet \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon) \qquad [6]$$

Remarks: Constants used in the model are $C_{\varepsilon 1}$ = 1.44, $C_{\varepsilon 2}$ = 1.92, C_μ = 0.09, σ_k = 1.0, and σ_ε = 1.3.

WIND FLOW AROUND THE SQUARE-PLANNED SERVICES CORE ON THE REFUGE FLOOR

Fig. 2(a) shows the pattern of wind on the horizontal symmetry plane of the refuge floor when the wind was blowing at normal incident to the building. By observation, the wind flows follow the wind pressure gradient distribution on the flow plane. On the upstream side of the refuge floor, while the separation of wind flow was observed at the building side wall corners, the wind was also observed to be dragged by the high gradient of wind pressure formed between the middle upstream region of the main service core where $C_p = 0.8$, and at the entrance of the internal corridors where $C_p = -0.6$. The entering wind after encountering the front face of the main service core then curved toward the building side walls. The flow finally escaped the refuge floor at the rear wall opening. The speed of the wind accelerated at the entrance of the internal corridors and forming a channel-like flow to exit the refuge floor opening adjacent to the building side walls. Meanwhile, re-attachment wind to the building flowed in an opposite direction against the direction of the prevailing wind direction at the downstream side of the building. A secondary wind structure was observed beyond the separation flow region adjacent to the building side walls. However, the wind behind the building rear face in the middle region of the refuge floor was found to re-enter the floor. The rear re-entering wind flow after encountering the rear face of the main services core, then curved towards to the refuge solid side walls. Although the re-entering wind joined the escaping wind flow, the exit speed of the rear re-entering wind flow was found to be relatively slow. This meant that the removal of captured smoke at the rear of the service core was questionable especially when the prevailing building wind speed was slow.

Fig. 2(b) shows the wind pattern for wind entering and escaping the refuge floor when the wind direction was 45^o to the building. The entering wind was divided by the symmetry axis of the flow stream and curved to the side walls of the building. Channel-like flows were observed in the internal corridors. Acceleration of the wind speed at the corridor entrances followed the magnitude of the pressure gradient. By observation, the acceleration of the wind after passing the side wall corner of the main service core became relatively slower so that the C_p at the region in front of the entrance of the connecting corridor was about 0.5 and C_p at the entrance to the first corridor was about -0.3. On the other hand, the gradient of C_p is about 0.3 to -0.6 around the entrance of the other corridor on the other side of the building. In the corridor on this other side, a small low speed reversely flowing stream was observed in the vicinity of the side wall of the service core. The building side wall cutting the axis of prevailing wind stream facilitated the wind to flow straight to the downstream side of the building. The flow stream formed a barrier to the re-attachment of wind to form the secondary flow at the side wall of this side of the building. Re-attaching wind was still re-entering the refuge floor at the building wake flow region but the wind speed was relatively stronger than in Fig 2 (a) so that there was a larger amount wind flow escaping the refuge floor on this wind side of the floor.

In Fig. 2(c), the wind is perpendicular (90^o) to the solid side walls of the building with the side wall closest to the wind direction being a barrier to the wind by preventing the wind to enter the refuge floor. Meanwhile, the other side wall of the building formed another barrier that prevented the re-attachment wind entering the refuge floor. A reverse flowing secondary flow structure was formed in the vicinity of the opened sides of the refuge floor. As a result, re-attachment wind of the building flowed from the building downstream side to the upwind corners of the building. The flow then joined the separated wind to flow around the building. However, channel-like flows were not formed in the corridors and instead the flow in this region was similar to a stagnant flow.

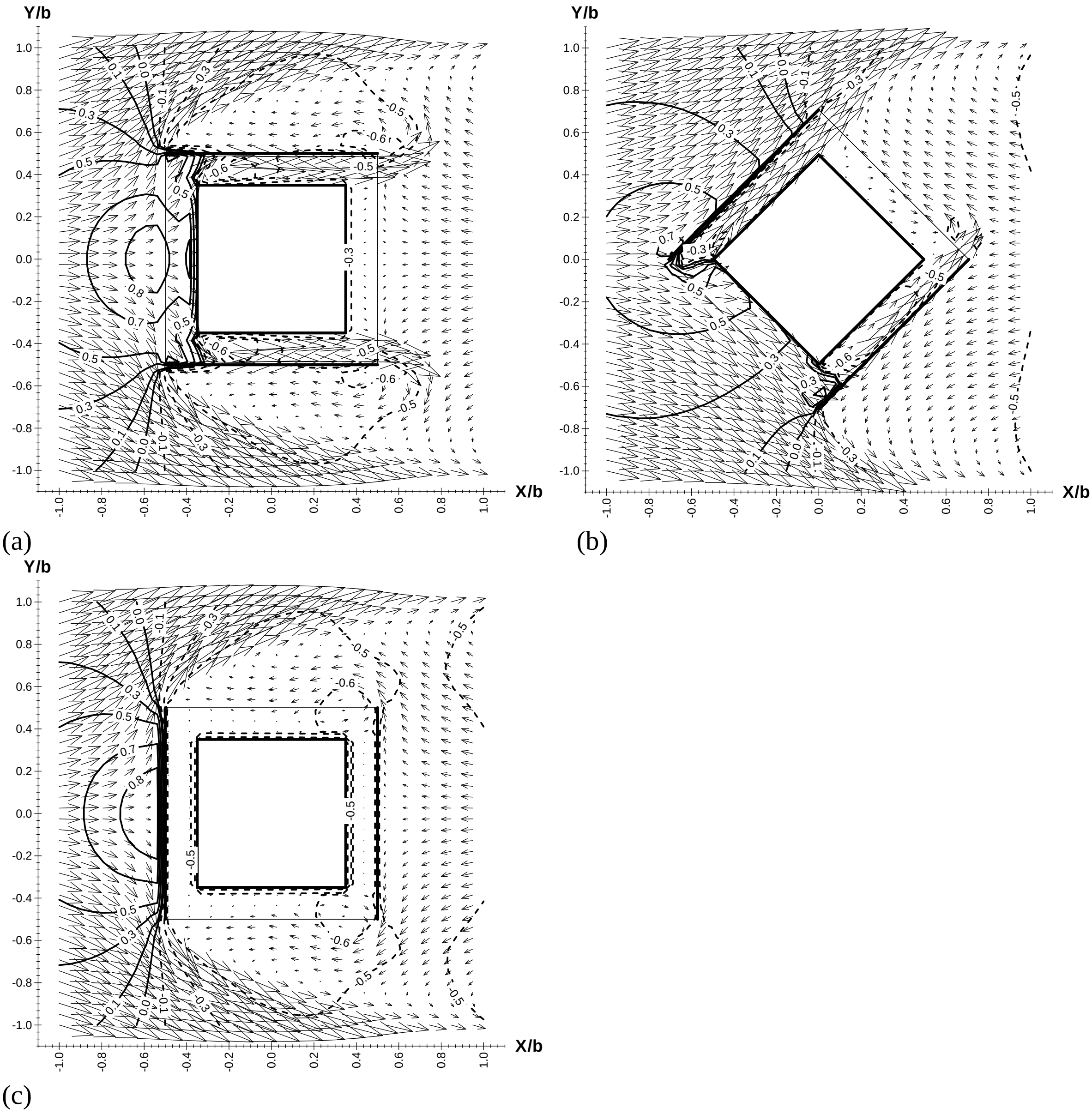


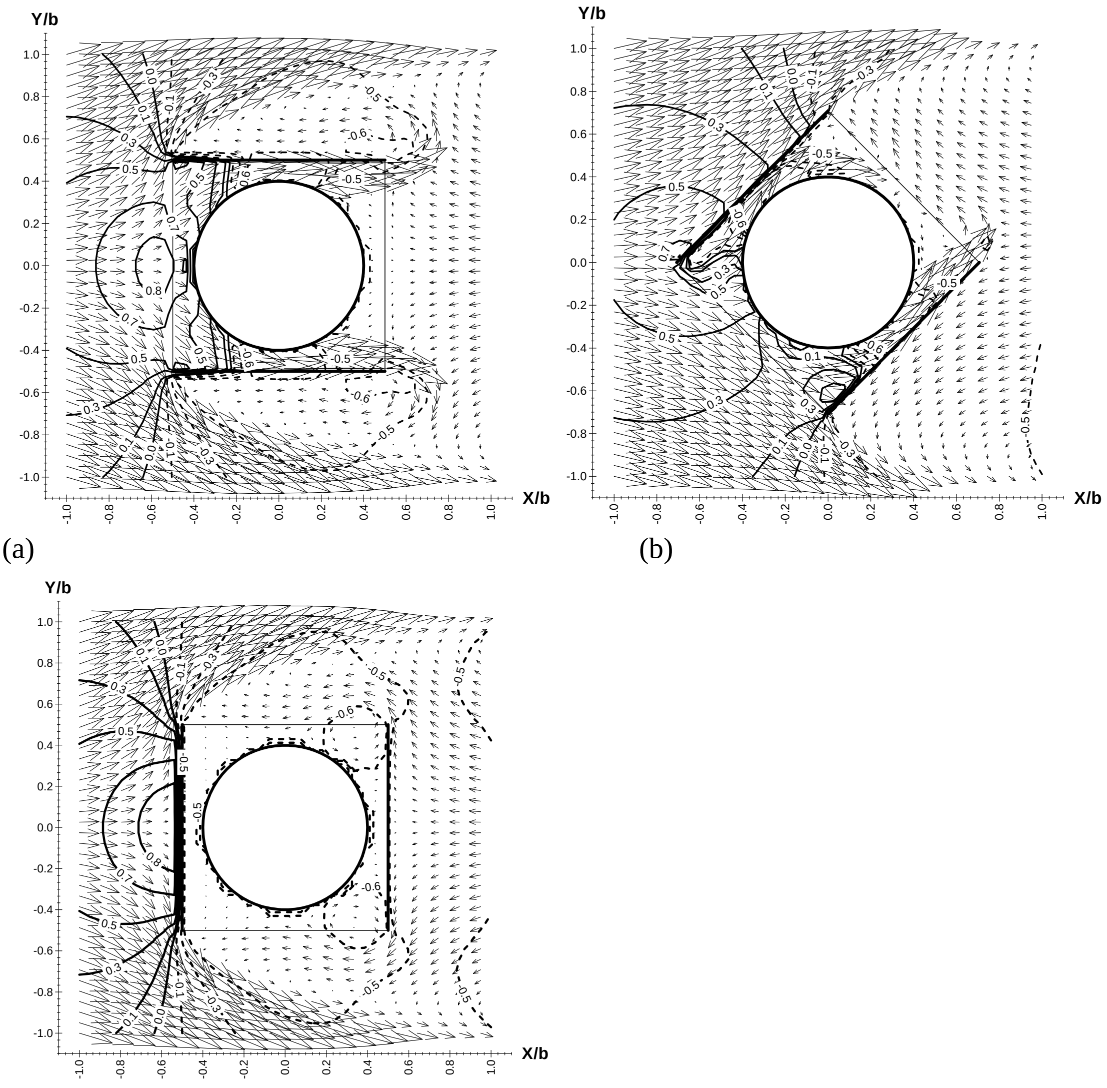
FIGURE 2. Normalised velocity vector distribution and contour lines of pressure coefficient over the symmetry horizontal flow plane of the refuge floor which is located at the building mid-height with a large square-planned main services core located at its centre: (a) Wind blowing at normal incident to the building (stream-wise), (b) Wind blowing at an angle diagonally to the building, and (c) Wind blowing at an angle perpendicular to the solid side walls of the refuge floor

WIND FLOW AROUND THE CIRCULAR SERVICES CORE ON THE REFUGE FLOOR

Fig. 3(a) shows the wind pattern on the horizontal plane at the same height of the refuge floor presented in the last analysis but with a circular service core at the middle of the refuge floor with the wind blowing at normal incident to the building. By comparing Fig. 3(a) with Fig. 2(a), it can be observed that the wind patterns generated by the two different service core shapes are similar. The pressure coefficient contours in the two figures are almost the same but for the circular service core the highest pressure gradient occurred some distance further into the refuge floor. However, the size of wake flow region was relatively smaller with a circular service core. In this case, the escaping wind was still facilitated by the channel-like flow formed beside the side walls of the main service core. Meanwhile, the low speed re-entering wind at the middle of the wake flow region of the service core was redirected by the rear face of the service core to flow to the building side walls and to join the escaping wind stream into the building wake flow region.

By comparing Fig. 3(b) with Fig. 2(b) with the wind diagonally to the building at 45°, similar patterns were also observed. However, the wind flow tended to adhere on the wall surface of the circular service core in the wake flow region of the service core. A small almost stagnated flow region was formed at the corner region on the refuge floor on the wind side of the building. Also the wind speed in the wake flow region of the circular service core was relatively slower than the speed recorded with the square-planned service core on the refuge floor.

By comparing Fig. 2(c) with Fig. 3(c) with the wind perpendicular to the solid walls, again the patterns were similar. It can be observed that the circular service core did not alter significantly the wind pattern observed with a square-planned service core.



(a) (b)

(c)

FIGURE 3. Normalised velocity vector distribution and contour line of pressure coefficient over the symmetry horizontal flow plane of the refuge floor which is located at the building mid-height with a large circular-based main services core located at its centre: (a) Wind blowing at normal incident to the building (stream-wise), (b) Wind blowing at an angle diagonally to the building, and (c) Wind blowing at an angle perpendicular to the solid side walls of the refuge floor

IMPLICATIONS ON THE MOVEMENT OF ENTERING SMOKE WITH SELECTED SHAPE OF MAIN SERVICE CORE IN THE MIDDLE OF THE REFUGE FLOOR

From the findings mentioned above, it can observed for a refuge floor at building mid-height that depending on the prevailing wind angle, there are different patterns for the wind entering and leaving the refuge floor. Regarding the wind effect due to the main centre service core being either square or circular, the patterns of wind behaviour on the horizontal symmetry flow plane of the refuge floor were similar at most wind angles. Areas of almost stagnant wind regions (including the refuge floor corridors when the wind is perpendicular to the side walls) and areas of low speed re-entering wind flow were identified. These are areas where the authors believe outside smoke could be trapped on the refuge floor. Fortunately, when the prevailing wind is normal or diagonal to the refuge floor opening, the channel-like flow created in the enclosed corridors minimises the induced flow from being logged on the floor and therefore removes any smoke entering the corridors of the refuge floor. Under these circumstances, it is recommended that the connecting staircase in the centre cores should be opened to face the refuge solid side walls so as to avoid the doors opening into areas of possible stagnant wind flow.

With the wind at 45^o to the building, the induced flow behaviour in the downstream side of the service core will be different according to the plan shape of the core. In the case of the circular service core, one side of the exiting wind from the corridor adheres to the surface of the core in the downstream region of refuge floor but does not escape the floor. Also an almost stagnated flow region was observed on the same side adjacent to the building rear wall corner. This observation implies an additional smoke logging hazard in this case. This indicates that there may not be any additional benefit to the safety design of the refuge floor by requiring the side walls of main service core to be curved. In fact, based on studies undertaken, it appears that stagnant like wind flows could be promoted by curving the side walls of the core and this could result in additional logging potential of entering smoke.

CONCLUDING REMARKS

A CFD wind tunnel experimental study is presented in this paper. The patterns of wind induced flow on the symmetry horizontal plane of the refuge floor where a large services core is located at its middle are depicted by the computed velocities data. The investigation was undertaken to determine the safety design of wind induced ventilation of a refuge floor with a square and circular planned service core. From the presented study (square refuge floor plan with a large service core and two solid walls on the opposite sides of the refuge floor and the other two sides open) the following conclusions can be drawn:

- Wind induced natural ventilation flow on the refuge floor is achievable at most wind angles with different sized pockets of low wind speed that may trap smoke in the wake area behind the large service core on the refuge floor.
- The channel-like wind flow created between the large service core and the solid side wall of the floor has an important role that facilitates the ventilation of the refuge floor and the removal of smoke.
- Compared with a square core, a large circular service core does not provide improved natural cross ventilation flow but in some wind angles (for example 45^o) may produce undesirable wind flow patterns which could produce a safety problem.
- When wind was blowing at an angle perpendicular to the solid wall on the refuge floor, almost stagnant wind flow was induced in the corridors formed by the side walls of the refuge floor and the large service core. Entering smoke in this region may easily be trapped.

As a general recommendation the connecting staircase doors in the large service core should be located so that they open to face the solid side walls of the refuge floor.

REFERENCES

1. Building Codes of Hong Kong, Code of Practice of the provision of means of escape in case of fire, pp. 29-33, 1996.
2. Conway, A.S., "Fire and Safety in Tall Buildings", Regional Conf. Tall Buildings, Hong Kong, 298-309, 1976.
3. Council on Tall Building and Urban Habitat, "Fire Safety in Tall Buildings" in Tall-Building Fire Performance, McGraw Hill, pp. 49-60, 1992.
4. Fire Code of the United States of America; NFPA 101: 1991, Code for Safety to Life from Fire in Building and Structure, National Fire Protection Association, 1991.
5. Fire Codes of Canada; NBCC: 1992, Codes for Safety to Life from Fire in Building and Structure, Section 3.2.6.3, 1992.
6. Fire Code of Singapore, Fire Safety and Shelter Bureau, Singapore Civil Defence Force, Chapter 6, Section 1.2.4(a), 2001. (Online version)
7. Fire Code of the People's Republic of China; GBJ 16-87: 1995, Fire Safety Code for Building Construction Design, Ministry of Public Security China, 1995. (In Chinese)
8. Yuen, K.K., Lo, S.M. and Yeoh, G.H., "Numerical Simulation of Wind-smoke Effect on the Designated Refuge Floor in High-rise Buildings", Interflam 99', Edinburgh, 1273-1280, 1999.
9. Lo, S.M., Chen, D., Yuen, K.K. and Lu, W.Z., "A Numerical Study of External Smoke Spread in Designated Refuge Floor", Building and Environment, 37, 257-268, 2002.
10. Lu, W., Lo, S.M., Fang, Z. and Yuen, K.K., "A Preliminary Investigation of Airflow Field in Designated Refuge Floor", Building and Environment, 36, 219-230, 2001.
11. Lu, W., Lo, S.M., Yuen, K.K. and Fang, Z., "An Investigation of Impact of Floor Setting on Airflow and Smoke Extraction in Designated Refuge Floor", International Journal of Computational Fluid Dynamics, 14, 327-337, 2001.
12. Lu, W., Lo, S.M., Fang, Z. and Yuen, K.K., "A CFD Study of Air Movement in Designated Refuge Floor", International Journal of Computational Fluid Dynamics, 15, 169-176, 2001.
13. Cheng, CK., and Yuen, K.K., "A Review of Designated Refuge Floor Studies", Intl. Conf. Building Fire Safety, Brisbane, 170-177, 2003.
14. Cheng, C.K., Yuen, K.K., Lam, K.M., and Lo, S.M., "CFD Wind Tunnel Test: Field Velocity Patterns of Wind on a Building with a Refuge Floor", International Journal of computational fluid dynamics, 19(7), 531-544, 2005.
15. Cheng, C.K., Lam, K.M., Yuen, K.K. and Lo, S.M., "Wind Tunnel and CFD Studies on Wind Flow around a High-rise Building with a Refuge Floor", 6th Intl. Conf. Tall Buildings, Hong Kong, 435-441, 2005.
16. Cheng, C.K., Lam, K.M., Yuen, K.K., Lo, S.M., and Liang, J., "A Study of Natural Ventilation in a Refuge Floor", Building and Environment, 2007 (In Press).
17. Launder, B.E. and Spalding, D.B., "The Numerical Computation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, 3, 269-289, 1974.
18. CFX-5 Reference Manual, 63-65, 2001.