The Seventh Asia-Pacific Conference on Wind Engineering, November 8-12, 2009, Taipei, Taiwan

# WIND FLOW IN THE RECESSED CAVITIES OF A TALL BUILDING

Charles C.K. Cheng[1], K.M. Lam[2] and Andrew Y.T. Leung[3]
[1] *Instructor, Department of Building and Construction, City University of Hong Kong, Hong Kong, bcccheng@cityu.edu.hk*
[2] *Associate Professor, Department of Civil Engineering, University of Hong Kong, Hong Kong, kmlam@hku.hk*
[3] *Chair Professor, Department of Building and Construction, City University of Hong Kong, Hong Kong, bcaleung@cityu.edu.hk*

**ABSTRACT**

In a congested city like Hong Kong, residential tall buildings are often built with an irregular plan form and with a number of apartments arranged as wing sections extending from a central core. To provide views and sufficient ventilation to the apartments, deeply recessed cavities are placed between adjacent building wings. This paper reports a CFD study of the wind-induced flow inside a recessed cavity of a tall building with an objective to assess the adequacy of ventilation inside the cavity. The dimensions of the cavity are varied systematically to investigate the flow exchange between the cavity and the outside at different heights. It is found that the flow inside the cavity is not a simple cross flow or a stagnation flow. Flow exchange takes place in different directions along the building heights.

***KEYWORDS: CFD, TALL BUILDING***

## Introduction

Wind effects play an important role in the serviceability of residential tall buildings. In order to optimize views and service requirements, most residential high-rise buildings adopt an irregular cross-sectional shape with apartments arranged as wing sections extending from a central core. Between adjacent building wings are deeply-recessed re-entrant bays (recessed cavity) or "light-wells" towards which kitchen and bathrooms windows open. Wind-induced flow pattern inside the re-entrant bays and air flow exchange with the external wind flow is important in many serviceability issues of the residential apartments such as natural ventilation, dispersion of kitchen and bathroom exhaust, and propagation of fire and smoke. The recessed re-entrants and the resulting irregular building cross-sections may also modify how wind flows around the building.

Wind loads on buildings with recessed corners or re-entrant cavities have been investigated in Building Research Establishment, UK (Cook 1985). For narrow recessed bays, it is speculated that flow tends to skip past the bays and leave almost stagnant flow inside. Pressure inside the bay is uniform and equals the average pressure on the opening face of the bay. However, there is no information on the flow pattern inside the bay and possible flow exchange between the recessed bay and the outside. Chow et al. (2002) carried out computational fluid dynamics (CFD) simulations to show that a flow would come down the re-entrant bay from the roof if there was no heat discharged into the bay but the study did not include an external wind flow. The major reason not including external wind flow in that study would be the excessively large demand of computational resources. Recently, the present authors developed an efficient CFD technique to include external wind flow in similar studies such as wind-induced natural ventilation of a refuge floor installed at the mid-height of a tall building (Cheng et al. 2005, 2007, 2008, Cheng 2009). This paper reports CFD investigation of wind-induced flow inside the recessed cavities of a tall building and the effect of the cavity size.

## Building Configurations and CFD technique

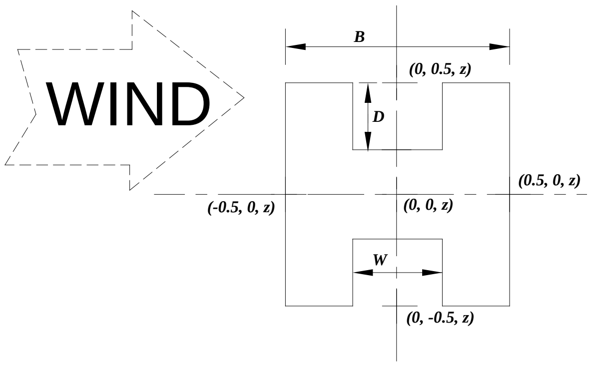
The objective of this study is wind-induced ventilation flow inside a recessed cavity of varying dimension. A tall building with an "H" planform is chosen as the target building configuration in which wind-induced flow can be studied inside the two recessed cavities on opposite walls of the building. Nine different H-shaped buildings are studied and their configurations were listed in Table 1. All buildings have the same square enveloping "H" planform with breadth $B$. In these H-shapes, the horizontal dimensions of the recessed cavity vary in a systematic manner covering three different widths, $W/B = \{0.25, 0.5, 0.75\}$ and three different depths, $D/B = \{0.125, 0.25, 0.375\}$. As a control, a square building with no recessed cavity is also studied. All buildings have the same height at $H/B = 6$.

Flow around the building and inside the recessed cavity is obtained numerically with the CFD technique (Cheng, et al. 2005, Lam and To 2005). A large computational domain of sizes ($10H \times 5H \times 3H$) in the ($x$, $y$, $z$) directions covers an entire building model and the surrounding space of sufficient fetch in all directions. Flow equations are solved by the finite volume code Ansys CFX with turbulence closure by the standard $k$-$\varepsilon$ model. Mean flow velocities and pressure are obtained from the steady flow solutions. In the inlet to the computational domain, natural wind flow over an open land terrain is prescribed as the boundary conditions on the profiles of mean wind speed and turbulence intensity as follows:

$$\frac{U(z)}{U_{\mathrm{H}}} = \left(\frac{z}{H}\right)^{0.19}, \quad I_u(z) = I_u(H) \times \left(\frac{z}{H}\right)^{-0.47}, \quad z < H \qquad (1)$$

where $U_{\mathrm{H}}$ is the mean wind speed at building roof height the turbulence intensity there has the value $I_u(H) = 0.104$.

Table 1: Building configurations: dimensions and aspect ratio of recessed cavities




| Case No. | Width ($W/B$) | Depth ($D/B$) | Aspect ratio ($W/D$) |
|---|---|---|---|
| 1 | 0.25 | 0.125 | 2:1 |
| 2 | 0.25 | 0.250 | 1:1 |
| 3 | 0.25 | 0.375 | 2:3 |
| 4 | 0.50 | 0.125 | 4:1 |
| 5 | 0.50 | 0.250 | 2:1 |
| 6 | 0.50 | 0.375 | 4:3 |
| 7 | 0.75 | 0.125 | 6:1 |
| 8 | 0.75 | 0.250 | 3:1 |
| 9 | 0.75 | 0.375 | 2:1 |

## Results and Discussion

CFD computation is carried out at different wind incidence angles to the building. This paper mainly reports the case in which the building faces without a cavity is normal to the wind (Table 1). Under this wind incidence, flow inside the cavity is induced by the external wind flow and there is no active ventilation of the cavity. If wind hits the building face with the cavity, the cavity on the windward face will be actively ventilated.

For the normal wind incidence reported in this paper, flow inside the cavities has slow velocities. Figure 1 shows the mean flow vectors around the nine H-shaped buildings and inside the recessed cavities at mid-height of the building ($z/H = 0.5$). The results show that the wind flow separates at the upwind corners of the building side faces and the side and leeward faces of the building are immersed in the separated wake. The flow in the vicinity of the building envelope is in the reverse direction at most locations. At the opening of the cavity on the building side face, upwind flow is found and near the rear portion of the opening there is flow of noticeable strength entering into the cavity. In the time-averaged sense, this result in a low-speed circulation vortex located at the rear corner of the cavity. In a cavity where the width is not too much larger than the depth, the vortex is well developed and covers a large portion of the cavity space. Thus, as the depth of the cavity becomes larger, the centre of the vortex structure shifts from a location near the rear wall of the cavity towards the center of the cavity.

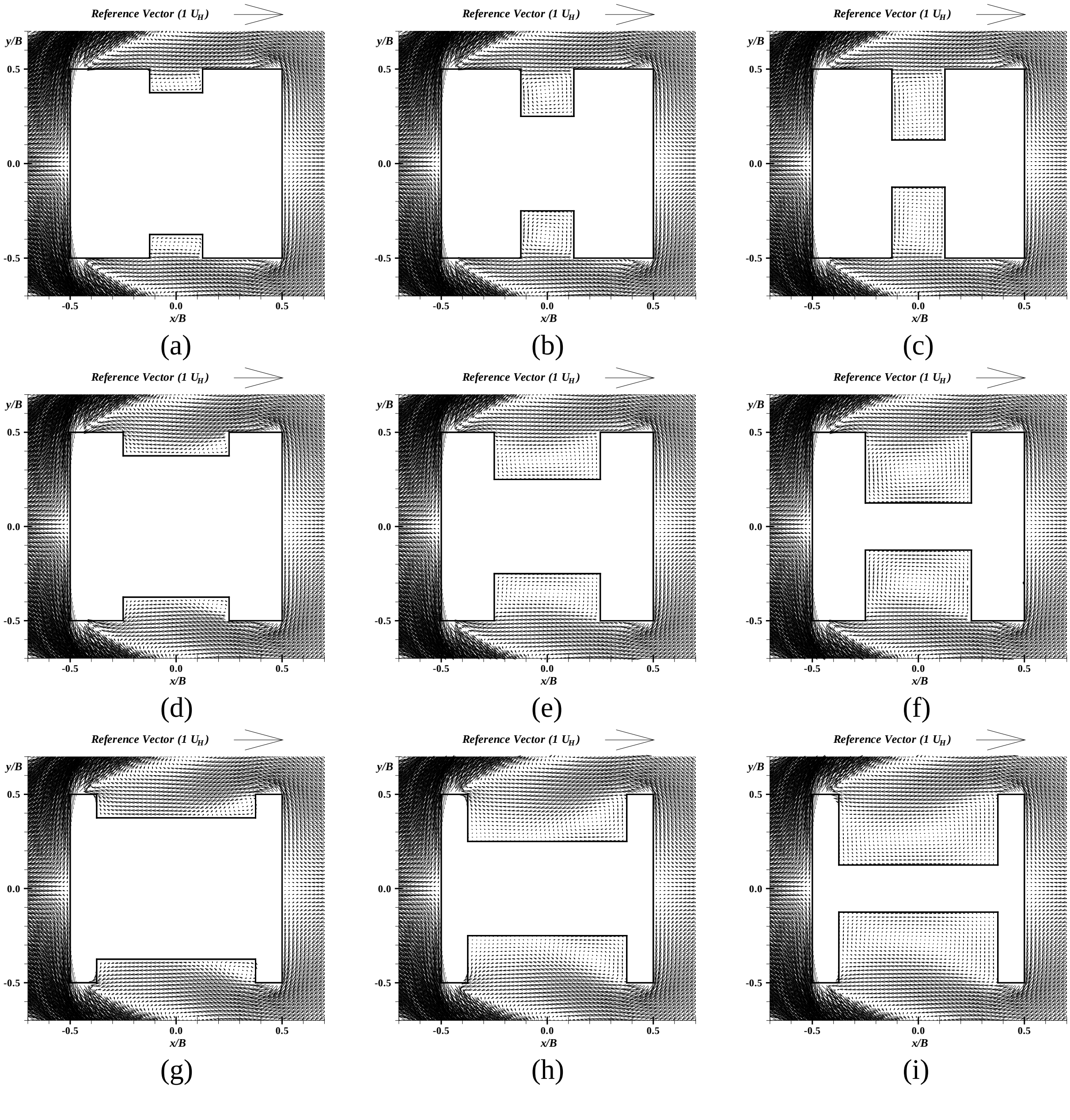



Figure 1: Examples of horizontal flow patterns at $z = H/2$: (a) Case 1; (b) Case 2; (c) Case 3; (d) Case 4; (e) Case 5; (f) Case 6; (g) Case 7; (h) Case 8; (i) Case 9.

Only the horizontal flow components are shown in Figure 1. At other heights, there can be net flow out of the cavity due to the horizontal flow. For example, Figure 2 shows that horizontal wind patterns at $z/H$ = 0.25 and = 0.75 for Case 2. It is clear that there is a net flow leaving the cavity at $z/H$ = 0.25. The flow comes from a vertical mass transport.

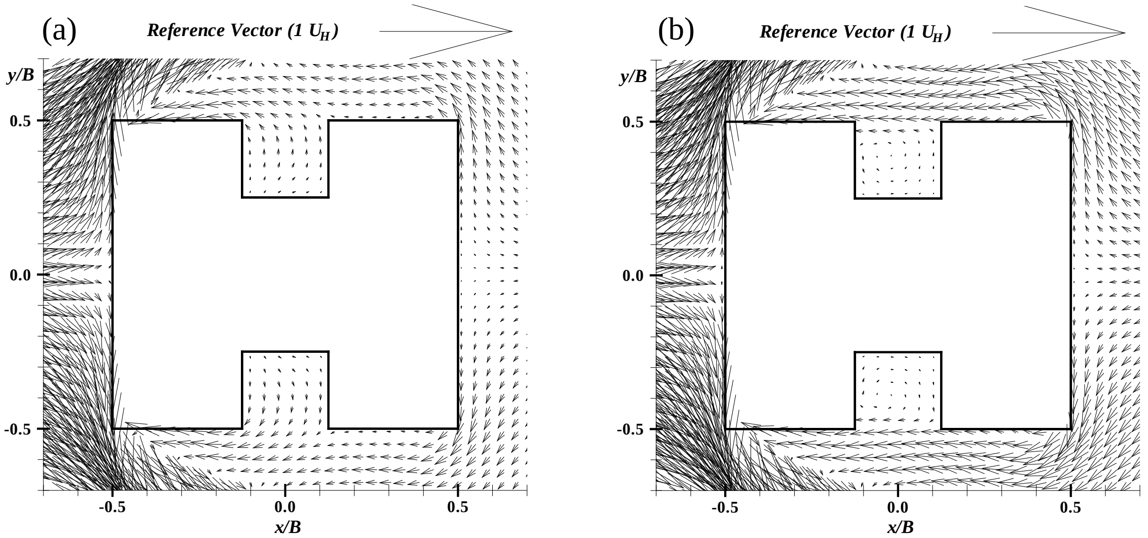



Figure 2: Examples of horizontal flow patterns of Case 2 at (a) $z = 0.25H$; (b) $z = 0.75H$

The net flow going into or leaving a recessed cavity is caused by the lateral velocity components at the opening face of the cavity. These components can have significant magnitudes. To quantify the net flow exchange between the cavity and the outside, a lateral velocity coefficient is defined as:

$$C_v(z) = \frac{1}{U_H \cdot W\Delta z} \int_z^{z+\Delta z} \int_W v \cdot dydz \qquad (2)$$

where the integration of the lateral velocity component $v$ is carried out along the opening of the cavity. The coefficient is actually the mean lateral velocity averaged along the opening and normalized by the mean wind speed at building roof height. The vertical distributions of this coefficient are shown in Figure 3 for the nine buildings, in three groups of different cavity depths. In every cavity of any size, there is a net flow into the cavity, that is $C_v < 0$, at the middle heights. The amount of this net inflow clearly increases with the width of the recessed cavity. For the narrowest cavities ($W/B$ = 0.25), the net inflow at the middle heights is the smallest. Nearer to the ground, there is a net flow out of the cavity over a short height and then the flow near the base of the building is always into the cavity with large magnitudes. Near the top of the building, flow is into the cavity and slightly lower than this zone, there is some low flow going out of the cavity except for the widest cavities. Actually, it is evident from all three parts of Figure 3 (for the three groups of cavities of different widths) that the widest cavities always have the largest amount of flow into them.

The vertical flow inside the cavity is important for supplying the net inflow or outflow of the cavity and the associated vertical mass and pollutant transport are related to of the cavity. The average vertical flow is represented by the vertical velocity coefficient defined as:

$$C_w(z) = \frac{1}{U_H \cdot DW} \int_D \int_W w \cdot dxdy \qquad (3)$$

The integration is carried out over a horizontal section of the cavity to compute the average vertical velocity component through that section. Figure 4 shows the distribution of this coefficient over the building height. Over the lower one quarter of the building height, there is net upward flow inside the cavity. Higher up, the flow is downward ($C_w$ < 0) with the maximum negative value of $C_w$ occurring between $z/H$ = 0.3 and 0.5 for cavities of different sizes. Inside the deepest cavities ($D/B$ = 0.375), Figure 4(c) shows that there is downward flow over the entire upper part of the cavities (above $z/H$ > 0.3) except very near to the roof. This downward flow is stronger and covers more upper floors when the cavities are not too wide. Figure 4(b) shows that downward flows of similar strengths are also observed inside the narrower middle-depth cavities ($D/B$ = 0.25 and $W/B$ < 0.75). Inside the shallowest cavities ($D/B$ = 0.125), the flow changes to upward on the upper half height of the building.

By continuity, all the net flow into the cavity through the cavity side opening (in Figure 3) must leave the cavity through the top face on the building roof. This "chimney flow" is observed for the wider cavities but for the narrowest cavities at $W/B$ = 0.25, flow is found to enter the cavity from the roof.

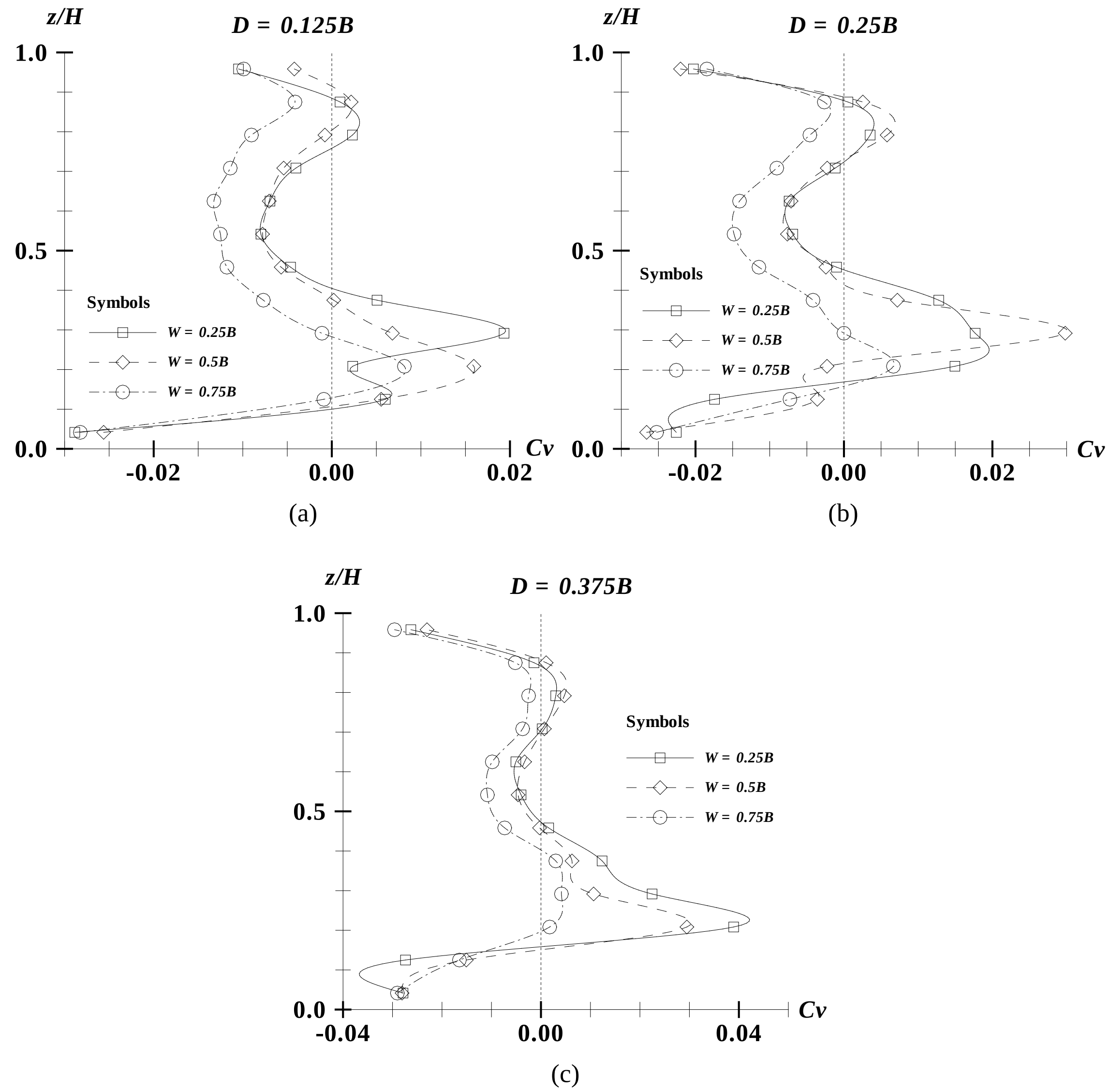



Figure 3: Vertical distribution of lateral velocity coefficient $C_v$.
$D/B$: (a) 0.125; (b) 0.25; (c) 0.375.

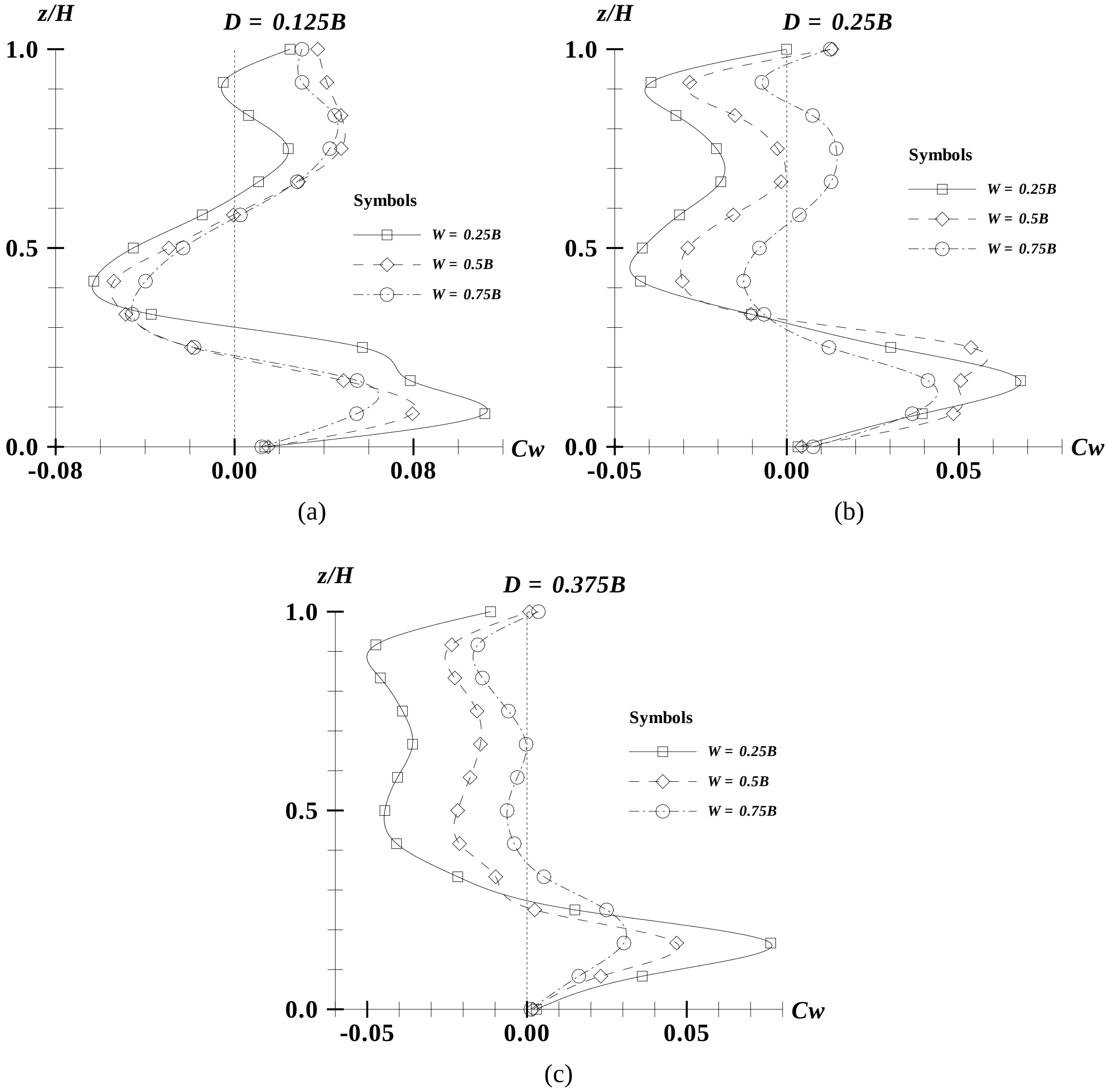



Figure 4: Vertical velocity coefficient $C_w$. $D/B$: (a) 0.125; (b) 0.25; (c) 0.375.

## Natural Ventilation of a Recessed Cavity

Some global features relative to ventilation of the recessed cavities can be deduced from Figures 3 and 4. Except the narrowest cavities, the chimney flow effect is in action. This means that there is a net inflow through the opening of a cavity and this flow leaves on the roof. However, the side flow is not all inward on all building floors. Flow enters the cavities mainly near the building base and at the middle floors. Only for the widest cavities that inward flow occurs for all upper part of the building. No matter the size of a cavity, there is always some outflow in the lower floors (between $z/H$ = 0.2 and 0.4). Inside the cavities, the vertical flow is also not always upward. Instead, downward flow is found on most upper building floors inside five deeper and narrower of the nine recessed.

The observed features suggest that there exist two counter acting circulation vortices inside the cavity. The lower vortex brings flow into the cavity near the ground and ejects flow out of the cavity at a height around $z/H \approx 0.3$. The upper vortex draws in flow at the upper floors, drives a downward flow inside the cavity and ejects the flow out of the cavity together with the lower vortex. In most cases, the centres of these vortices are located nearer to the cavity open face. The vortices are responsible for the global ventilation behavior in Figures 3 and 4. As an illustration, Figure 5 shows some flow streamline patterns for Cases 6 and 9. These two buildings have the deepest cavities at $D/B$ = 0.375. The streamlines shown are derived from the velocities ($v$, $w$) on the vertical ($y$, $z$) plane at the mid-width of the recessed cavity. They show how flow is induced to flow into the cavity from the middle levels of the cavity open face and leaves at the roof opening. The two vortices inside the cavity can be observed in the complicated flow patterns.

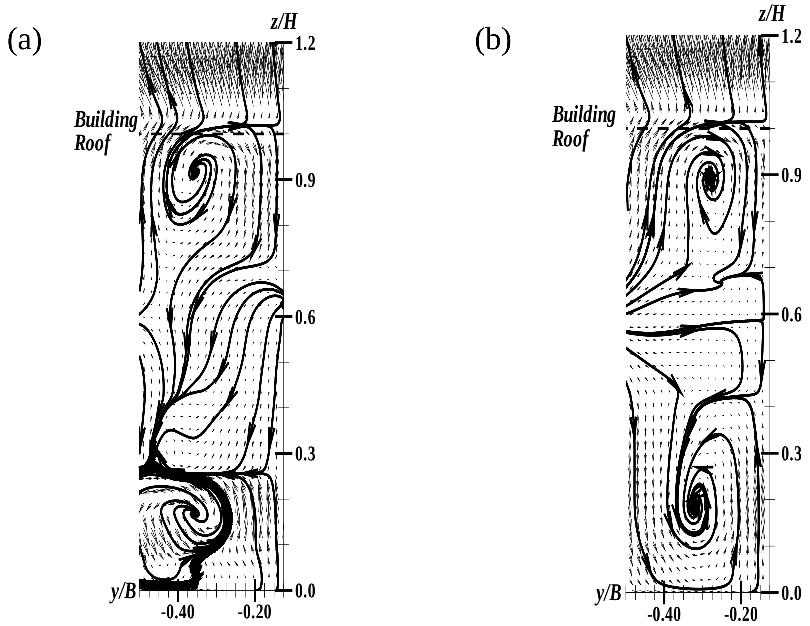



Figure 5: Some wind flow streamlines: (a) Case 6; (b) Case 9.

For the narrowest cavities at $W/B$ = 0.25, the side flow into or out of the cavity has much lower strength and so is the flow escaping through the roof face. There is little air exchange between the outside wind flow and the cavities. The poor ventilation of these recessed cavities will hinder the dispersion of smoke or virus from the cavities. In this aspect, wider open recessed cavities are preferred.

## Conclusions

This paper reports a CFD study of the wind-induced flow inside a recessed cavity of a tall building. The effect of the dimensions of the cavity is studied systematically to investigate the flow exchange between the cavity and the outside at different heights. It is found that the flow inside the cavity is not a simple cross flow or a stagnation flow. Through the open face of a cavity, there can be flow going into or out of the cavity depending on different heights. In the vertical direction, the flow at different heights can also be upward or downward inside the cavities. The flow patterns inside the cavities are very complex and they seem to be caused by two circulation vortices inside the cavity. Wind-induced ventilation is important in taking up exhaust gases, heat or even smoke from the recessed cavities. The results suggest that, wider recessed cavities are better ventilated.

## Acknowledgements

This investigation is supported by a grant awarded by the Research Grants Council of Hong Kong (HKU/713507). The CFD resource is partially supported by the centre of Applied Computing and Interactive Media, City University of Hong Kong.

## References

Cheng, C.K., Yuen, K.K., Lam, K.M. and Lo, S.M. (2005), "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.

Cheng, C.K., Lam, K.M., Yuen, K.K., Lo, S.M. and Liang, J. (2007), "A study of natural ventilation in a refuge floor", Building and Environment, 42, 3322-3332.

Cheng, C.K., Lam, K.M. and Demirbilek, F.N. (2008), "Effects of building wall arrangements on wind-induced ventilation through the refuge floor of a tall building", Journal of Wind Engineering and Industrial Aerodynamics. 96(5), 656-664.

Cheng, C.K. (2009), "A fire safety of Hong Kong refuge floor building wall layout design", Journal of Fire Safety, 44, 545-558.

Chow, T.T., Lin, Z. and Liu, J.P. (2002), "Effect of condensing unit operation on kitchen exhaust at residential tower", Journal of Architectural Science Review, 45, 3-12.

Cook, N.J. (1985), The designer's guide to wind loading of building structure, Part II: Static structures, Building Research Establishment, Watford, 1985.

Lam, K.M. and To, A.P. (2006), "Reliability of numerical computation of pedestrian level wind environment around a row of tall buildings", Wind and Structures, 9(6), 473-492.