

Wind environment for a high-rise building adjacent to a vertical wall

W.K. CHOW

Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, China

Abstract: Adequate provision of natural ventilation in tropical areas is regarded as one of the design criteria for assessing 'green' or 'sustainable' building. Driving forces for natural ventilation in those areas are basically wind-induced action, not stack effect nor buoyancy of warm air columns. Wind-induced air movement is a transient phenomenon depends not only on the wind speeds measured in some designated sites, but strongly affected by the surrounding environment. This is particularly obvious in a dense urban area where buildings are closely built together. For a building located adjacent to a taller building, turbulent effects due to incident wind fields would be important. Note that windows or even refuge floors might be opened at some levels, giving flow paths to change the entire patterns of air flow and pressure distribution.

In this paper, the wind-induced indoor airflow in a building adjacent to a vertical wall is studied by Computational Fluid Dynamics. A CFD software based on large-eddy simulations is selected. Different scenarios on building height, openings and distances away from the vertical wall will be simulated. Incident airflows of a parabolic boundary layer and a top-hat flow pattern are considered. The separate flow at the sharp corner would be different for these two incident airflow profiles. Results are useful not only for providing better ventilation design, but also for working out fire safety management in case of accidents. Note that smoke, or even flame, will spread by following the wind-induced air motion.

Keywords: wind environment; CFD; vertical wall

1. Introduction

Introducing more natural ventilation is one of the targets for new green or sustainable buildings[1,2] in both advanced[3] and developing countries. Those buildings are typical for new projects in 'cities groups' in the Far East[4,5]. This would provide more fresh air and reduce the amount of cooling load in areas with hot and humid climate. Better thermal comfort might be achieved in comparing to those buildings relying on mechanical air-conditioning systems. In fact, this is one of the criteria to satisfy the requirements of 'green or sustainable' buildings[1,2].

Driving forces for natural ventilation[6] are stack effect, buoyancy [7] due to warm air columns and wind-induced action. Stack effect is not significant for places with low temperature differentials between indoor and outdoor, as demonstrated before in Hong Kong[8]. Stack effect might be obvious in lift shaft or staircase in high-rise buildings as their geometrical ratio of length or width to height is very small. There are no significant changes in surface temperature distribution on the ground level in many places, and so warm air columns are not found. How can gliders fly without 'thermals'? Therefore, wind-induced action is the main driving force for natural ventilation. However, wind is a transient phenomenon. Orientation of the

building and ventilation openings have to be designed with respect to the geological features and past record on wind environment to give adequate natural indoor air flow, i.e. 'wind capture'. But the design should be considered carefully with reference to neighbors. This is very obvious in dense urban areas with many high-rise buildings like big cities[5] in the Far East.

When wind is moving towards a block with sharp edge as in Fig. 1c, there will be separate flow (as the pressure beneath is not balanced) at the top and the sides of the block. The flow might be reattached, depending on the flow conditions and length of the block. There will also be recirculation found in the front of the cube as pointed out by Paterson and Apelt [9] and others [10-12] in studying incident air flow to a cube in a wind tunnel. If another smaller cube is placed in there, the resultant air flow would be very complicated. Depending on the distance between the two cubes, the incident horizontal air flow would be directed downward by the vertical wall, moved along the barrier, and then diverted horizontally in the opposite direction as in Figure 1c. Lower part of the leeward side might have higher air speeds than the same level at the windward side, but flowing in a direction opposite to the incident wind [13].

It is very difficult to carry out field measurement on

natural ventilation in building sites as the phenomenon is transient and chaotic. Scale model studies with a wind tunnel are useful but scaling parameters have to be identified. Numerical simulations of natural ventilation using CFD appeared in the literature[14-16] would be another possible way of studying the air flow. In-depth physical justification for simulating building aerodynamics is required[12].

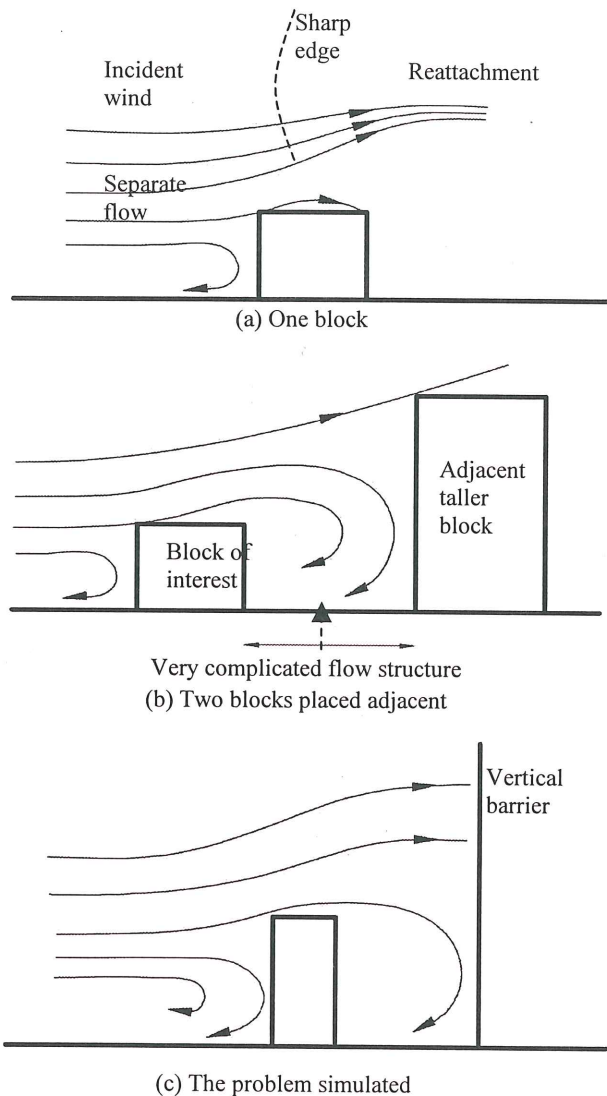


Fig. 1 Wind effect of adjacent buildings

As demonstrated by an earlier study[13] with Computational Fluid Dynamics (CFD), the indoor air speed inside a unit with windows on the leeward side opened might be even stronger than the indoor air speed when the windward side window is opened. That depends on the height of the unit above the floor level. The study was based on a CFD model with Reynolds Averaging Navier-Stokes (RANS) equation [17] method. There are debates on using RANS to study the wind-induced airflow. Software based on large-eddy simulation (LES)[11,18] appears to be

better for solving some problems such as thermal-induced air flow. That was pointed out years ago but the computer resources required are too demanding. With the rapid development of hardware, it is now possible to carry out LES on studying some flow problems even in a personal computer, though still impossible for others[19].

2. Computational Fluid Dynamics

A Computational Fluid Dynamics (CFD) package, Fire Dynamic Simulator (FDS) version 3.01 was released by the Building and Fire Research Laboratory, National Institute of Standards and Technology, USA. Compressible fire-induced air flow can be simulated with hot smoke taken as thermally expandable gas[18,20,21]. A set of governing equations^[22] suitable for simulating fluid flow induced by buoyancy with low Mach number was derived. The Boussinesq approximation is no more necessary and constraints on inviscid fluid are removed. The FDS model allows larger variations of both density and temperature. An important assumption is that the pressure variation is small, so that the ideal gas law can be approximated by decomposing the pressure p into a "background" component p_0 , a hydrostatic component $\rho_\infty g z$, and a flow-induced perturbation ϕ .

$$p = p_0 - \rho_\infty g z + \phi \quad (1)$$

This assumption is reasonable for flows with low Mach number because the temperature is inversely proportional to density. The pressure p in the state and energy equation is replaced by the background pressure p_0 to filter out sound waves of speeds much faster than typical flow speeds expected in a compartment fire. The momentum equations were rearranged based on the assumption that vorticity generation due to buoyancy prevails over its generation due to baroclinic effect.

Short turbulence length scale will be simulated by large eddy simulations (LES). A flow variable $\phi(x, t)$ would be decomposed by filtering into a resolved component (large-scale component) $\bar{\phi}(x, t)$ and a subgrid-scale component $\phi'(x, t)$ as:

$$\phi(x, t) = \bar{\phi}(x, t) + \phi'(x, t) \quad (2)$$

The resolvable-scale component $\bar{\phi}(x, t)$ is obtained by integrating a filter kernel $G(x - x', \Delta)$, over the domain of interest Ω and the filter width Δ :

$$\bar{\phi}(x,t) = \int_{\Omega} G(x-x', \Delta) \phi(x', t) dx' \quad (3)$$

Small-scale turbulence is simulated by LES such as Smagorinsky sub-grid models.

3. Numerical Experiments

For simulating indoor air flow and environment surrounding a building, an incident flow pattern of wind moving towards the building is assigned. Effect of partition in a single-vent house with cross ventilation was studied[23]. Similar studies in apartment with large openings were reported[24,25]. Distribution of wind pressure coefficients was predicted[16], values were lying between -0.8 to 0.8 as reported[26]. Results would be different if the incident wind fields are changed.

In this paper, wind is taken as blowing towards the building with one of the two flow profiles. The first one is a 'top-hat' model with constant speed of 10 ms^{-1} . The second one is a parabolic wind profile similar to an atmospheric boundary layer with horizontal wind speed u (in ms^{-1}) at height z (in m) above the ground level given by:

$$u = 10 \left(\frac{z}{100} \right)^2 \quad (4)$$

A 20 m cube is taken as an example with six cases simulated. The computing domain is 100 m long, 50 m wide and 50 m high as in Fig. 2a. A three dimensional Cartesian coordinate system is used with z along the vertical direction. Incident wind was flowing towards the left hand side with the two profiles as described above. The right hand side is either a free boundary or a solid wall. The other three sides (two vertical planes at $y = 0 \text{ m}$ and 50 m and the horizontal plane at $z = 50 \text{ m}$) are taken as free boundaries. The six cases as shown in Figures 2b to g are:

Case W1: A cube of 20 m, all free boundaries except the wind flow profiles of 'top-hat' model or parabolic profiles at the left hand side. The computing domain is 100 m long, 50 m wide and 50 m high being divided into 128 by 64 by 64 parts along the x -, y - and z -directions.

Case W2: Same as W1 but with a solid wall at the end, 30 m from the cube.

Case W3: Same as W2 but another 20 m cube placed 3 m above the first cube. This is to simulate air flow through a refuge floor [27,28] as

required in than 25 m.

Case W4: Same as W3, but a 'wall' is put in the 'gap' at windward side.

Case W5: Same as W4, but the whole set-up moved closer to the wall, at 10 m away.

Case W6: A tall object 43 m high with area 20 m by 20 m placed 30 m from the wall. Some fire codes for buildings taller

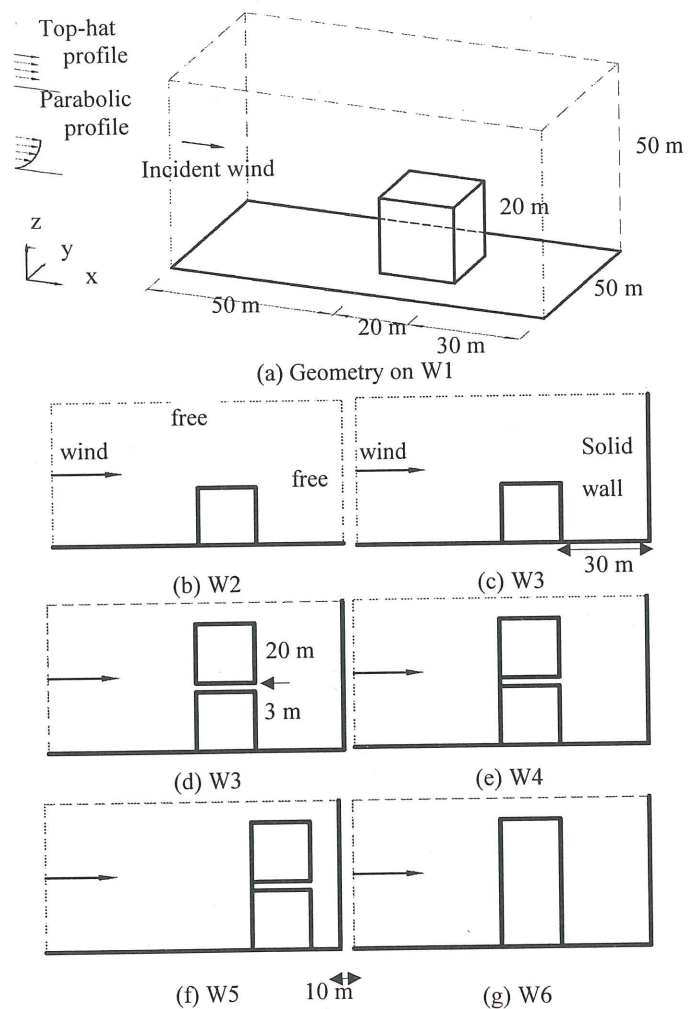


Fig. 2 Scenarios simulated

The CFD simulator selected is the Fire Dynamics Simulator FDS version 3.01[20,21]. Acoustic filtering technique was applied to remove the flow with high Mach number and LES applied to model smaller turbulent scales. Results of the velocity vectors across the central vertical plane along the y -direction at 20 s and 100 s are shown in Figures 3 to 6.

4. Results

It is observed that the two incident wind flow profiles with a 'top-hat' model and a parabolic model gave very different results. For W1, a very strong separate flow was induced at the sharp corner of the

building block for a 'top-hat' flow profile. That is because the incident wind speed is also very strong at lower level, but air movement was blocked by the cube. Even at early time of 20 s, air circulation was formed above the sharp corner, and interacted with the strong horizontal incident wind speed. This was not observed for the parabolic flow profile.

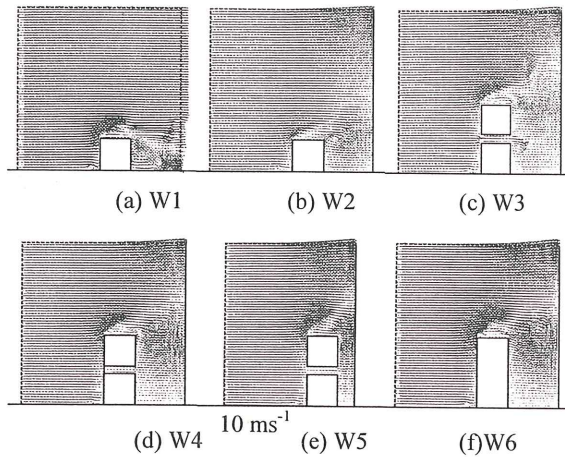


Fig. 3 Results across the central vertical plane for top-hat profiles at 20 s

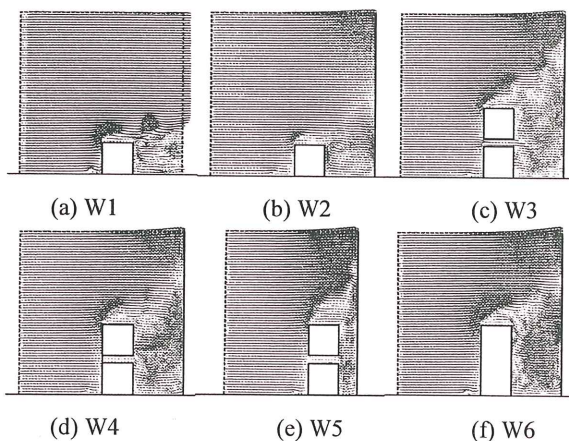


Fig. 4 Results across the central vertical plane for top-hat profiles at 100 s

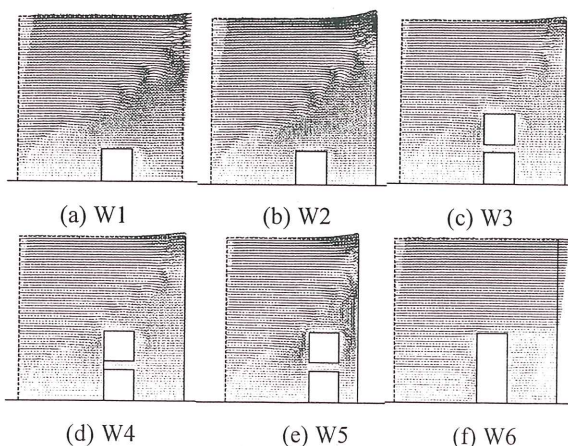


Fig. 5 Results across the central vertical plane for parabolic wind profiles at 20 s

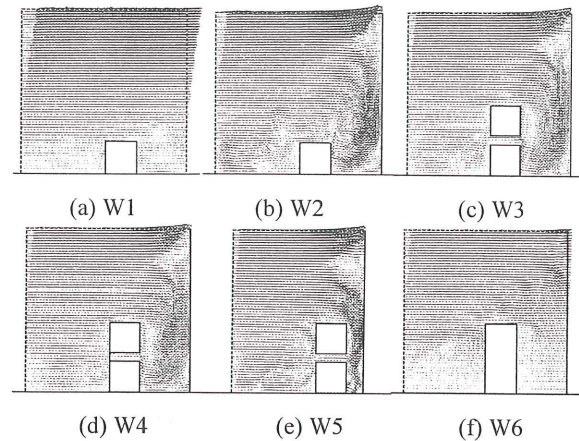
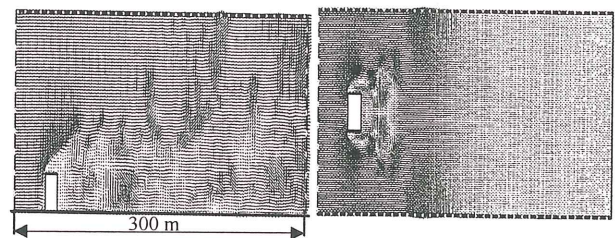
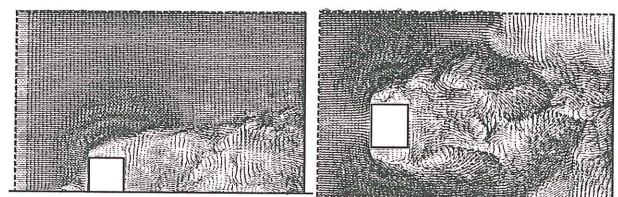


Fig. 6 Results across the central vertical plane for parabolic wind profiles at 100 s

An extended computing domain for the case W1 was repeated by taking a much longer distance 300 m away from the back of the cube as shown in the air flow pattern at 100 s in Fig. 7, and zoomed views at 80 s. The computing domain was divided into $192 \times 64 \times 64$ parts.



(a) Central vertical plane at 100 s (b) Central horizontal plane across the cube at 100 s



(c) Zoomed vertical plane at 80 s (b) Zoomed horizontal plane at 80 s

Fig. 7 W1 top-hat profile for longer computing domain

For scenario W2 with a vertical wall, horizontal incident wind would be directed downward along the vertical wall. Air would then flow towards the building from the back and then move up again along the backside wall. Very complicated flow was observed in the region between the cube and the wall.

For scenario W3 with a gap between the two cubes, directions of air flow in the gap are different for the two flow profiles. The configuration is similar to a refuge floor. This is clearly observed in the air flow pattern at 100 s across that gap (or the refuge floor) for the two wind profiles shown in Figures 8a and b.

Strong wind flow of 10 ms^{-1} with the top-hat model would induce air flowing in direction along the incident wind in the gap. But the parabolic flow profile would give air flowing opposite to the incident wind in the gap.

Results on closing the gap wall at the windward side for W4 would give opposite air flow directions for the two profiles as well. The horizontal air flow patterns in the gap are shown in Figures 8c and d. This is further verified by moving the entire building to the wall as shown in case W5 at 100 s in Fig. 9.

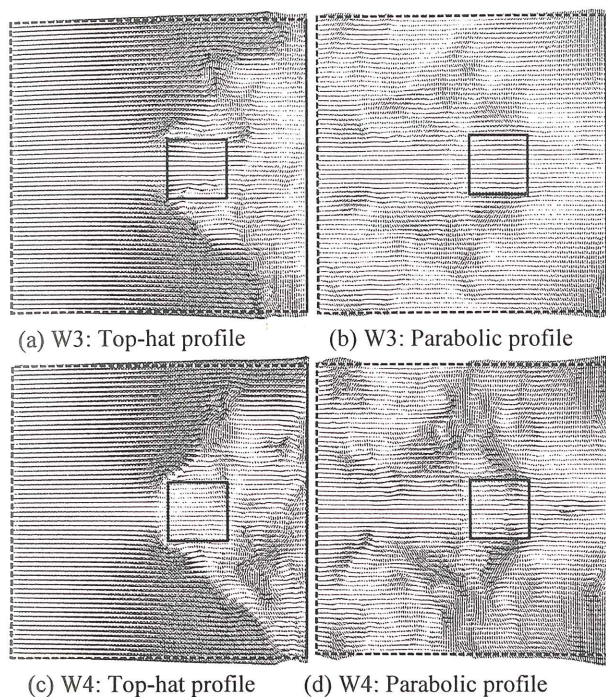


Fig. 8 Air speed across the gap at 100 s

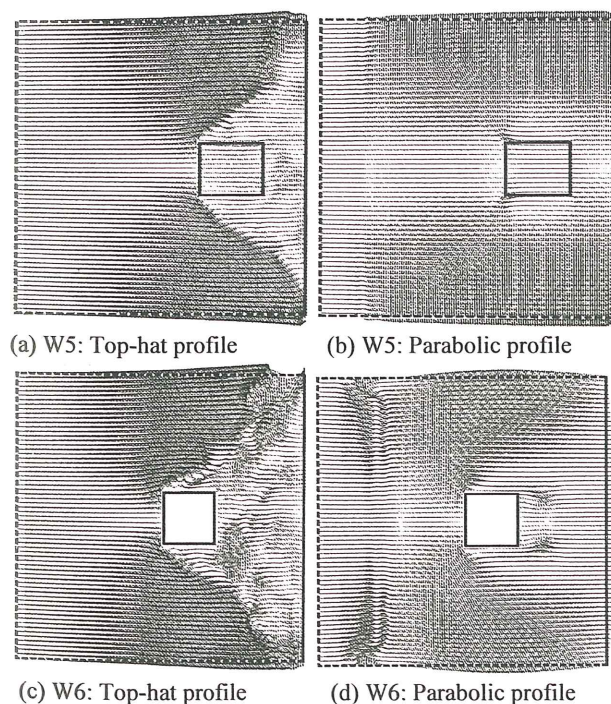


Fig. 9 Air flow pattern across the horizontal plane

Flow pattern for W6 would give a 'background' of how air would move for wind blowing towards a 43 m tall building, without opening a gap or refuge floor.

5. Discussions

With the application of CFD, effects of adjacent taller buildings or vertical barriers, and arrangement of refuge floors on indoor airflow were studied. The horizontal incident air flow would be diverted after striking at the vertical wall of the taller building or a hillside. Turbulence[10-12] generated in areas between the building and the vertical barrier would give a very complicated air flow pattern. It is observed that the indoor flow pattern depends on the height of the unit concerned, as illustrated by cases W3 to W6, with a gap at 20 m above the ground in simulating a refuge floor. Opening the right windows at the leeward window but closing the one at windward side as in W3 and W4 would 'catch the wind' to give better indoor air flow distribution. Whether this is good or not depends on the requirement.

For example, too strong the indoor air flow might give adverse effects. Air draft[29] might lead to thermal discomfort in cooler areas, but quite welcome in tropical countries[30]. When there is a fire, strong air flow pattern will lead to rapid rate of smoke (or even flame) spreading[31]. Therefore, some windows should be closed when an accidental fire is detected. Information on the air flow pattern as simulated in above will be useful for working out fire safety management[32].

6. Conclusion

As pointed out previously on supporting the codes review[33] on natural ventilation provision with CFD in Hong Kong[34], both the window areas and their positions[34,35] are important in inducing air flow inside. Wind-induced air flow is the main driver in places where stack effect is not significant and buoyant 'thermals' are not found. Buildings or hillside adjacent to that building would have significant effect on the indoor air flow of the unit concerned. Opening windows on the leeward side in units at some levels might give very strong indoor air flow induced by wind. To make good use of the incident wind field for providing natural ventilation, windows have to be designed properly so as not to 'short-circuit' the air flow.

Making good use of natural ventilation[6] will lead to better air quality and thermal environment. But the arrangement might give excessive wind loading under strong wind. Glass panels[36] in curtain walled buildings were pulled down during typhoon in Hong Kong. Fast rate of smoke (or even flame) spreading might be resulted when there is a fire.

If a fire occurs inside such a flat, heat will be mixed much more rapidly in a building with such good natural ventilation provision. Fire can then be spread out more rapidly to the adjacent areas. In fact, fire safety aspects[37] for green or sustainable buildings[1,2] should be considered carefully. For example, a proper fire action plan[32] should be worked out to instruct security guards to close some windows and open others to extract smoke as a natural vent when there is an accidental fire.

Acknowledgement

The work is supported by the subgroup 4 "Smoke movement and toxic harmful gases transfer in fire" of the National 973 project "Fire dynamics and fundamentals of fire safety" with project No. 2100CB409609.

References

- 1 R.J. Cole "Building environmental assessment methods: clarifying intentions" Building Research and Information, Vol. 27, No. 4-5, p. 230-346 (1999).
- 2 N.K. Larsson "Development of a building performance rating and labeling system in Canada" Building Research and Information, Vol. 27, No. 4-5, p. 332-341 (1999).
- 3 R.J. Cole and L. Mitchell "Customizing and using GBTool: two case-study projects" Building Research and Information, Vol. 27, No. 4-5, p. 257-275 (1999).
- 4 L. Yiu and P. Deo "Exploration of a Generic Vs Regional Specific Indicator Framework for Green Building Assessment: including example from N. China" Proceedings of the 3rd International Conference on Sustainable Building, Oslo, Norway, September 23-25 (2002).
- 5 C.M. Leung, C.M. Chung, M.F. Hui and M.S.H. Lau "The Green building initiatives" Sustainable Building 2002: Proceedings of the 3rd International Conference on Sustainable Building, Oslo, Norway, September 23-25 (2002).
- 6 D. Etheridge and M. Sandberg "Building ventilation: Theory and measurement" John Wiley & Sons, Chichester (1996).
- 7 K.T. Andersen "Theoretical considerations on natural ventilation by thermal buoyancy" ASHRAE Transactions: Symposia, Vol. 101, No. 2, p. 1103-1117 (1995).
- 8 W.Y. Hung and W.K. Chow "A review on architectural aspects of atrium buildings" Architectural Science Review, Vol. 44, No. 3, p. 285-295 (2001).
- 8 Paterson and C.J. Apelt "Simulation of flow past a cube in a turbulent boundary layer" Journal of Wind Engineering and Industrial Aerodynamics, Vol. 35, No. 2, p. 149-176 (1990).
- 10 R. Martinuzzi "Experimentelle untersuchung der umströmung wandgebundener rechteckiger prismatischer hindernisse" Dissertation, University Erlangen-Nürnberg, p. 176, Fig. 5.34 (1992).
- 11 W. Rodi "Large-eddy simulation and statistical turbulence models: Complementary approaches" Lecture 3 in New Tools in Turbulence Modelling, Les Houches School, May 21-31, 1996, Ed. by Olivier Métais & Joel Ferziger, Springer (1997).
- 12 Y. Gao and W.K. Chow "Review on wind-induced air movement around a cube" Architectural Science Review – Accepted to publish (2003).
- 13 W.K. Chow "Wind-induced indoor air flow in a highrise building adjacent to a vertical wall" Applied Energy – Accepted to publish (2003).
- 14 S. Murakami and S. Kato "Numerical and experimental study on room airflow – 3-D predictions using the k-ε turbulence model" Building and Environment, Vol. 24, No. 1, p. 85-97 (1989).
- 15 S.J. Emmerich "Use of computational fluid dynamics to analyze indoor air quality issues" Report NISTIR 5997, Building and Fire Research Laboratory, National Institute of Standards and Technology, Gaithersburg, Maryland, USA (1997).
- 16 M.G. Smith, R.R. Walker and M.D.A.E.S. Perera "Prediction of natural ventilation air flows in a non-urban office" ROOMVENT'92, Aalborg, Denmark, September, p. 463-478 (1992).
- 17 B.E. Launder and D.B. Spalding "Mathematical models of turbulence" Academic Press – London (1974).
- 18 K.B. McGrattan, H.R. Baum and R.G. Rehm "Large eddy simulation of smoke movement" Fire Safety Journal, Vol. 30, No. 2, p. 161-178 (1998).
- 19 S. Krajnović and L. Davidson "Development of large-eddy simulation for vehicle aerodynamics" Proceedings of IMECE2002, ASME International Mechanical Engineering Congress & Exposition, November 17-22, 2002, New Orleans, Louisiana (2002).
- 20 K.B. McGrattan, G.P. Forney, J.E. Floyd, S. Hostikka and K. Prasad, Fire dynamics simulator (version 3) user's guide. NISTIR 6784, National Institute of Standards and Technology, Gaithersburg, MD, U.S.A. (2002).
- 21 K.B. McGrattan, H.R. Baum, R.G. Rehm, A. Hamins, G.P. Forney, K. Prasad, J.E. Floyd and S. Hostikka, Fire dynamics simulator (version 3) - technical reference guide. NISTIR 6783, National Institute of Standards and Technology, Gaithersburg, MD, U.S.A. (2002).
- 22 R.G. Rehm and H.R. Baum "The equations of motion of thermally driven, buoyancy flows" J. Res. Nat. Bur. Stand. 83 297-308 (1978).
- 23 J. Tsutsumi, T. Katayama, T. Hayashi and P. He "Numerical simulation of cross-ventilation in a single-unit house" Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE, p. 447-451 (1992).
- 24 Z. Wang "Study of cross-ventilation in apartment with

- large openings – comparison between wind tunnel tests and simulations” Proceedings of the Second International Conference on Indoor Air Quality, Ventilation and Energy Conservation in Buildings, 9-12 May, Montreal, Canada, p. 449-455 (1995).
- 25 Z. Wang “Comparison of airflow within six apartments: cross-ventilation study with computational fluid dynamics” ROOMVENT’96, p. 271-277 (1996).
- 26 S.D. Lamining and J.R. Salmon “Wind data for design of smoke control system” ASHRAE Transaction, Vol. 104, Part 1, Paper RP-816 (1998).
- 27 J.H. Klote “Design of smoke control systems for areas of refuge” NISTIR 5132, National Institute of Standards and Technology, Gaithersburg, MD, U.S.A. (1993).
- 28 W.Z. Lu, S.M. Lo, Z. Fang and K.K. Yuen “A CFD study of air movement in designated refuge floor” International Journal of Computational Fluid Dynamics, Vol. 15, No. 2, p. 169-176 (2001).
- 29 P.O. Fanger, A.K. Melkov, H. Hanzawa and J. Ring “Air turbulence and sensation of draught” Energy and Buildings, Vol. 12, No. 1, p. 21-39 (1988).
- 30 W.K. Chow and W.Y. Fung “Investigation of the subjective response to elevated air velocities: Climate chamber experiments in Hong Kong” Energy and Buildings, Vol. 20, No. 3, p. 187-192 (1994).
- 31 M. Poreh and S. Trebukov “Wind effects on smoke motion in buildings” Fire Safety Journal, Vol. 35, p. 257-273 (2000).
- 32 H.L. Malhotra “Fire safety in buildings” Building Research Establishment Report, Department of the Environment, Building Research Establishment, Fire Research Station, Borehamwood, Herts, WD6 2BL, UK (1987).
- 33 Consultancy Agreement Brief “Review of lighting and ventilation requirements in buildings” Buildings Department, Hong Kong Special Administrative Region, China (1999).
- 34 W.K. Chow “A comment on studying the ventilation requirements for buildings in the Hong Kong Special Administrative Region” International Journal on Architectural Science, Vol. 1, No. 1, p. 1-13 (2000).
- 35 P.C. Chou, C.M. Chiang, Y.K. Chuah and W.A. Wang “Effects of window position on the air flow distribution in a cross-ventilated residential bedroom” Indoor + Built Environment, Vol. 7, p. 300-307 (1998).
- 36 Hung Wing-yan and Chow Wan-ki “Fire safety in new architectural design associated with the extensive use of glass” 2003 International Symposium on Fire Science and Fire-protecting Engineering (2003 ISFSFE) to be held on 12-15 October 2003, Beijing, China – Accepted to present (2003).
- 37 W.K. Chow and C.L. Chow “Awareness of fire safety for green and sustainable buildings” Paper 2, Session 3 on “Fire engineering and the community”, International Conference – Fire Asia 2003, 26-28 February 2003, Hong Kong (2003).