CFD Simulation of Natural Ventilation in Urban Buildings Due to Wind Effect

Z.T. Ai, The Hong Kong Polytechnic University, Hong Kong
C.M. Mak, The Hong Kong Polytechnic University, Hong Kong
D. Yang, The Hong Kong Polytechnic University, Hong Kong
H.M. Wong, The University of Hong Kong, Hong Kong

The European Conference on Sustainability, Energy & the Environment 2017
Official Conference Proceedings

Abstract
Previous studies regarding natural ventilation in buildings were mostly limited to isolated buildings. Considering an urban context, this study investigates the wind-induced single-sided natural ventilation in buildings near a long street canyon under a perpendicular wind direction using CFD method. Four aspect ratios (AR) of the street canyon, from 1.0, 2.0, 4.0 to 6.0, are investigated to examine the influence of street configuration. Ventilation rate of rooms in buildings is analyzed. AR influences ventilation rate and its distribution among rooms along height of buildings. The percentage decrease of ventilation rate of buildings reaches 67% when AR of a street canyon is increased from 1.0 to 6.0. The findings of this study are intended to increase the understanding of natural ventilation performance in urban buildings.

Keywords: Natural ventilation, urban environment, street canyon, aspect ratio, CFD simulation

iafor
The International Academic Forum
www.iafor.org

Introduction

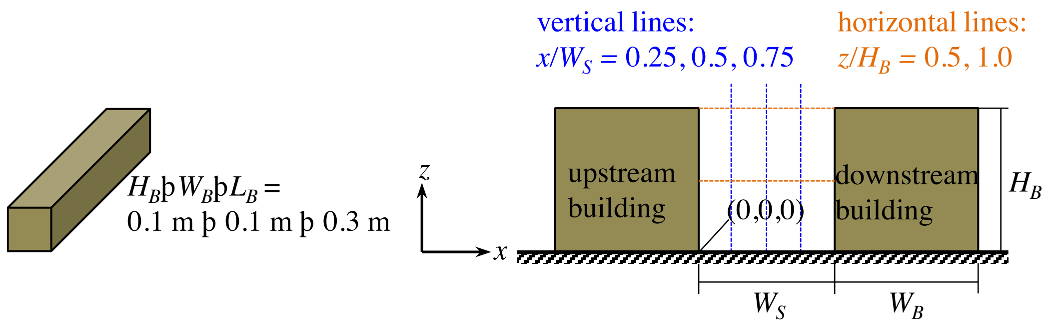
Given that few buildings in urban areas can be regarded as isolated buildings, the urban microclimate would directly influence the natural ventilation in buildings (Ai and Mak, 2015). Review of on-site measurements of wind speeds inside and outside (mostly above) street canyons by Ai and Mak (2015) suggests that, depending on AR, the ratio of wind speed inside a canyon to that outside the canyon ranges mostly between 10% and 30%. In addition, wind direction in vicinity of a building near a street canyon is dominated by the along canyon flow combined with upward and downward movements, while the normal-to-facade flows are very weak. The decreased wind speeds and substantially changed flow patterns inside street canyons would influence (mostly lower) wind-induced pressure difference for natural ventilation in buildings, which thus highlight the importance of taking into account urban context in natural ventilation studies.

A few studies examined the natural ventilation performance in buildings when considering the influence of surrounding buildings, which show that the wind speed near building facades could be lowered by up to 86.8% (Gao and Lee, 2012) and the natural ventilation performance in urban buildings could drop by up to 83% (Georgakis and Santamouris, 2006) when compared to isolated buildings. These findings are, however, case dependent and again may not be applicable to a different situation.

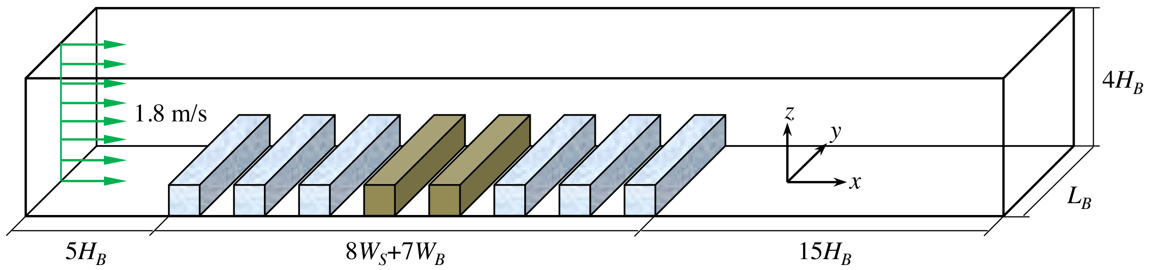
In general, current understanding of natural ventilation in urban buildings is far from sufficient, and there is still a strong need to provide a systematical investigation using a general urban geometry. From both street configuration perspective, the objective of this study is to investigate the wind-induced single-sided natural ventilation in urban buildings. A long street canyon flanked by two buildings is considered, while four AR values are investigated, including 1.0, 2.0, 4.0 and 6.0, which all correspond to the skimming flow regime (Oke, 1987; Ai and Mak, 2015). Ventilation performance of rooms is evaluated using air change rate per hour (ACH). CFD simulations are conducted and steady-state results are obtained by solving the Reynolds-Averaged Navier-Stokes (RANS) equations using the Renormalization group (RNG) $k-\varepsilon$ turbulence model.

CFD simulations: model validation

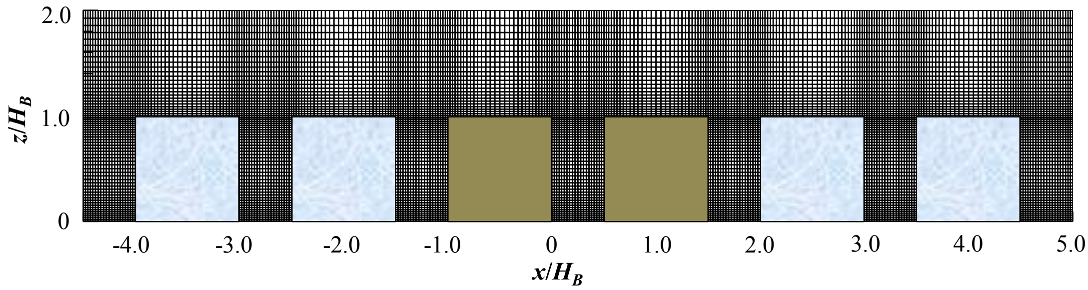
Li et al. (2008) conducted a water tunnel ($L_T \times W_T \times H_T$: 10 m × 0.3 m × 0.5 m) experiment to measure the flow field inside a street canyon. The street canyon of AR (H_B/W_S) equal to 1.0 was investigated, which was formed by eight identical building models ($L_B \times W_B \times H_B$: 0.3 m × 0.1 m × 0.1 m). The water flow approaches the street canyons in a perpendicular direction (Figure 1 (b)). The height of the buildings and the width of the street canyon was equal to 0.1 m. The depth of water in the experiment was fixed at 0.4 m. The Reynolds number based on the reference water speed (U_{ref}) in freestream at $z = 0.3$ m and the building height was 12,000, implying that U_{ref} was equal to 1.8 m/s. Velocity components in the streamwise and vertical directions along three vertical lines and two horizontal lines on the vertical centerplane ($y = 0$) of the target street canyon (see Figure 1 (a)) were measured.



(a) building model and the vertical centerplane ($y = 0$) of the target street canyon model



(b) computational domain



(c) mesh information on part of vertical centerplane (AR = 2)

Figure 1: The street canyon model, computational domain and mesh information.

The building model and street canyon model used in CFD simulations are the same with those in the experiments (see Figure 1 (a) and (b)). Computational domain and its dimensions (see Figure 1 (b)) are selected based on the existing best practice guidelines for CFD simulation of urban aerodynamics (Franke et al., 2007; Tominaga et al., 2008), except that the height and lateral length of the domain follow those in the experiments. The whole computational domain is constructed using structured hexahedral cells (see Figure 1 (c)). After a grid sensitivity test, a grid with 3,168,000 cells in total is used. Same with the experiments, a uniform wind speed at 1.8 m/s is defined at the inlet of the computational domain. A turbulent intensity of 5% and a turbulent length scale of 0.35 m are imposed for the inflow. At the domain outlet, pressure outlet with zero static pressure is specified. Zero normal velocity and zero normal gradients of all variables are defined at the lateral sides and the top of the domain. The domain ground and the building surfaces are imposed as non-slip walls.

ANSYS Fluent 13.0.0 (Fluent, 2010) is employed to conduct the CFD simulations. A steady-state two-equation RANS model, namely RNG $k-\varepsilon$ model (Yakhot and Orszag, 1986), is used to predict the flow and turbulence fields. RNG $k-\varepsilon$ model is selected due to its good performance in predicting flow in and around buildings (Ai et al., 2013). A two-layer model (Wolfshtein, 1969) and standard wall functions are combined to treat the near-wall regions. SIMPLEC algorithm is used for coupling pressure and momentum equations. The second-order schemes are used to discrete the convection and diffusion terms. Convergence is achieved when all scaled residuals are less than 10^{-5} and the average wind speeds at important locations within the street canyon are stable for over 50 iterations.

Figure 2 shows the velocity component in x direction along some vertical and horizontal lines on the vertical centerplane of the targe street canyon. In general, the CFD predictions show a good agreement with the experimental data. Overall, the CFD method used in this study including the turbulence model selected (namely, RNG $k-\varepsilon$ model) can predict acceptably the flow field in the street canyon, which justifies the use of it in the rest of this paper.

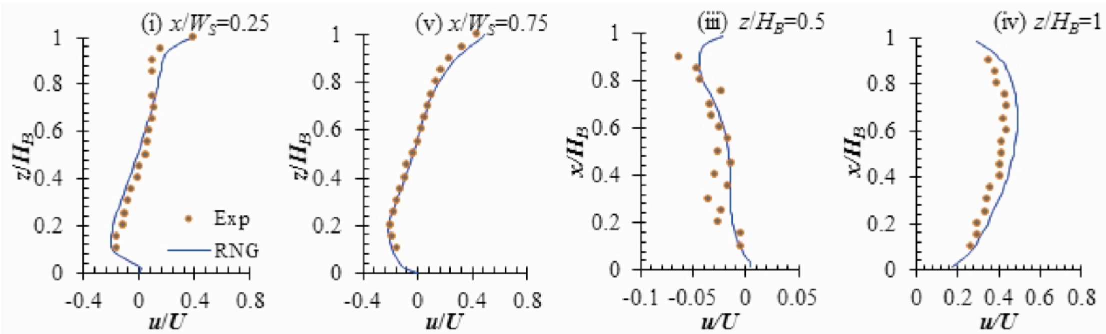


Figure 2: Velocity components in x and z directions along two vertical and two horizontal lines on the vertical centerplane of the target street canyon.

CFD simulations: geometry and computational settings

A street canyon model formed by two parallel slab-like buildings is investigated in this study (see Figure 3 (a)). Four aspect ratios (H/B), namely 1.0, 2.0, 4.0 and 6.0, are considered. The height of buildings (H) remains constant, while the width of the street canyon (B) is varied to form different AR values. The street canyon is included into a T-shape computational domain (see Figure 3 (b)). This T-shape computational domain configuration and its dimensions are selected, because many previous studies employed such a T-shape computational domain to investigate the atmospheric flow and related processes in a street canyon (Ai and Mak, 2016).

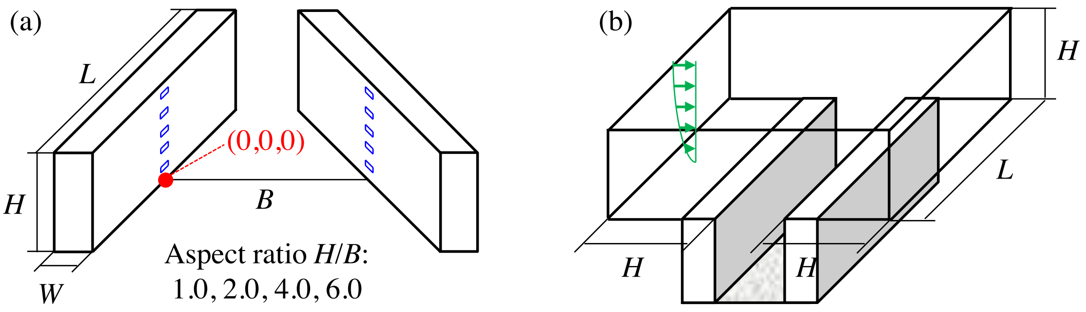


Figure 3: Street canyon model (a) and computational domain (b).

The dimensions of each building are 55.2 m × 29.7 m × 6.2 m ($L \times H \times W$). Considering that the dimensions of a single room are 2.4 m × 2.7 m × 3.1 m ($L_R \times H_R \times W_R$), the building models contain 23 × 11 rooms on both windward and leeward sides (see Figure 4). Five rooms on the second, fourth, sixth, eighth and tenth floors, respectively, at the horizontal centres of the leeward facade of the upstream building and the windward facade of the downstream building are investigated (see Figure 3 (a) and Figure 4). The dimensions of the openings are 1.2 m × 0.8 m ($H_O \times W_O$). Depending on the building facade and floor where a room is located, the rooms are named (see Figure 4 (a)).

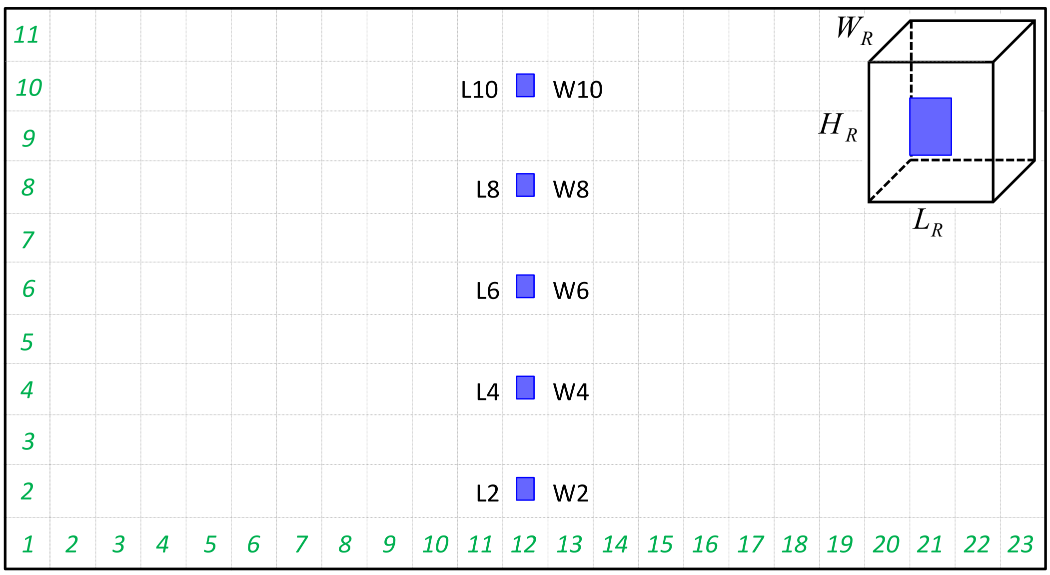


Figure 4: Details of building models of the street canyon; the 'L' and 'W' indicate the leeward facade of the upstream building and windward facade of the downstream building, respectively.

The street canyon is simulated as a 1:15 reduced-scale model, considering that a small model can save computational cost (Ai and Mak, 2014b). The Re number based on the wind speed and building height in the present study is around 2.4 × 10^5, which is sufficiently high to allow an independence of Re number (Snyder, 1981). In this study, hexahedral cells are used to construct the whole computational domain for all cases. As a result of grid sensitivity test, the number of cells used eventually for the four cases are summarized in Table 1.

Table 1: A summary of the number of cells used for the four cases.

AR	1.0	2.0	4.0	6.0
No. of cells	6,637,568	5,813,248	4,988,928	4,164,608

A logarithmic law velocity profile is defined at the domain inlet. The aerodynamic roughness height $z_0 = 0.001$ m, the reference velocity (U_{ref}) at the height of $Z_{ref} = 2m$, H are U_{ref} = 4.2 m/s and Z_{ref} = 59.4 m, the friction velocity of atmospheric flow above the building tops (u^*) is 0.17 m/s. The streamwise turbulence intensity I_u is defined as 15% above the building top, while the hydraulic diameter is 38.6 m. The other boundary conditions, solver settings and convergence criteria are the same with those used in the validation study.

CFD simulations: results and analysis

The integral method is employed to calculate the mean ventilation rate (Q_{mean}), which integrates the mean velocities on an opening that are extracted from a time-averaged flow field generated by the steady RANS simulations. The ACH value can be then obtained by: $ACH = Q_{mean}/V_{room}$, in which V_{room} is the volume of a room.

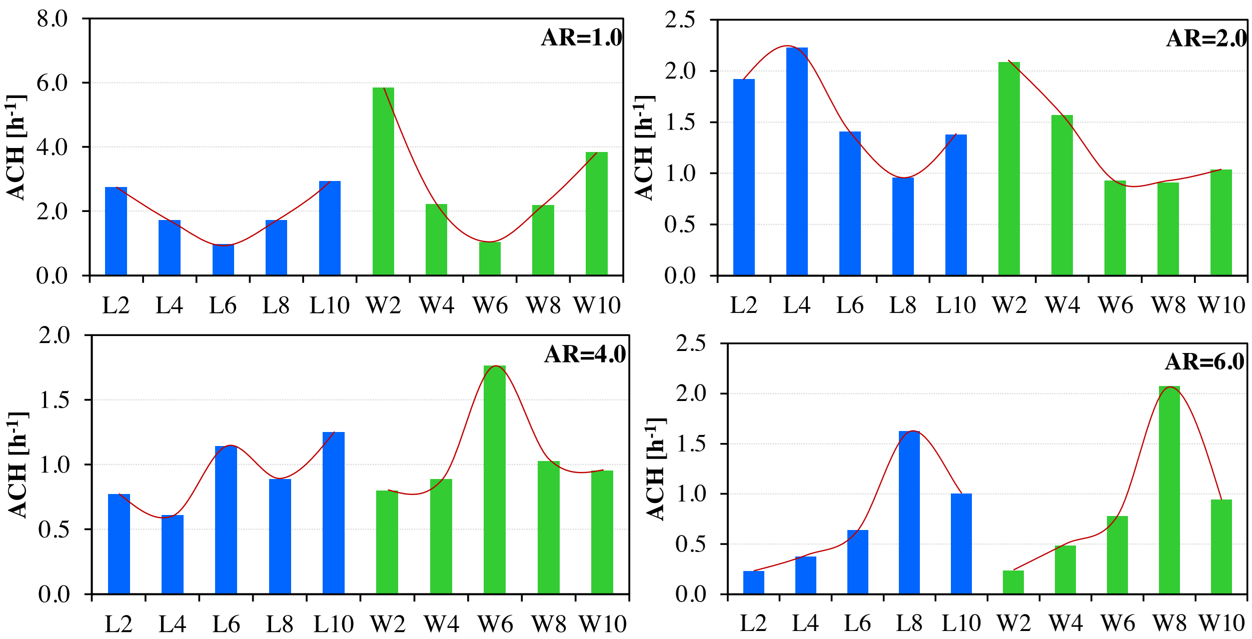


Figure 5: ACH values of rooms in buildings near the street canyon under different aspect ratios.

Figure 5 presents the ACH values of rooms at both leeward and windward sides of the street canyon under different aspect ratios. It is obvious that the ACH values along height are not uniformly distributed. For AR = 1.0, the rooms located on the lowest floor and the top most floor show the best ventilation performance. This distribution of ACH values is similar with that observed on an isolated building (Ai et al., 2013). The locations of the rooms that have the best ventilation performance shift with the increase of aspect ratio. The reason for the distributions of ACH values along height can be obtained from the analysis of the flow patterns inside street canyons (see Figure 6).

For AR = 1.0, a large and strong vortex is formed inside the street canyon. Rooms located at the lower and top parts of the street canyon would have the highest possibility to experience normal-to-facade near-wall flows, which contribute mostly to the indoor and outdoor flow exchange. Although it is a fact that along-facade flows still contribute to indoor ventilation due to their turbulent effects (Ai and Mak, 2014a), for the case with perpendicular wind direction studied in this paper, the normal-to-facade flows should be the main contributor of the indoor ventilation. When AR = 2.0, two vortices are formed and they interact at the lower part of the street canyon, which produces opportunities for nearby rooms to have higher ventilation rates (see Figure 5: AR = 2.0). Similar reasons can be found for the cases of AR = 4.0 and AR = 6.0. However, when AR = 6.0, the skimming flow above the street canyon cannot penetrate deeply into the lower part of the street canyon, which results in the very low ventilation rates for the rooms located at the lower part of the street canyon. These distributions of ACH values along height under different aspect ratios are important findings, which reveal the locations where the best and the worst ventilation could occur.

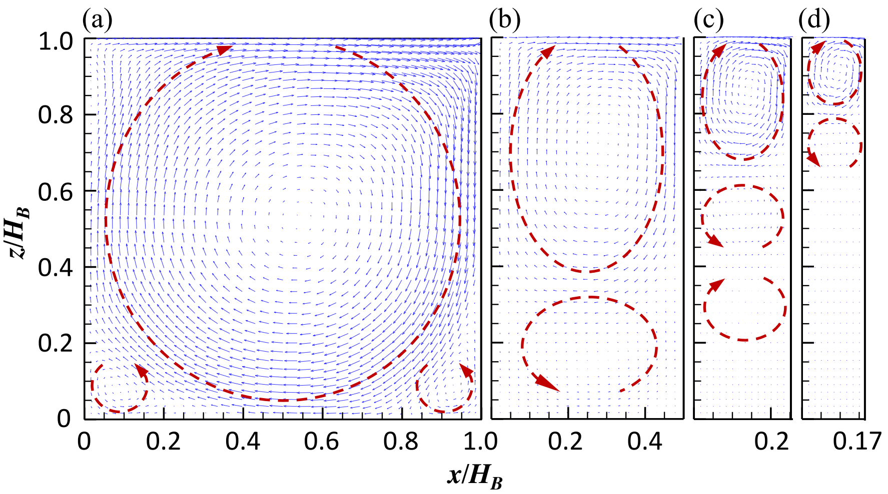


Figure 6: Flow vectors on the vertical centerplane of the street canyon under different aspect ratios: (a) AR = 1.0, (b) AR = 2.0, (c) AR = 4.0 and (d) AR = 6.0.

Figure 7 shows the average ACH values of rooms for different aspect ratios. ACH values on both the leeward and windward sides decrease with the increase of aspect ratio. Taking the case of AR = 1.0 as the base case, the percentage decreases of ACH of other cases with a higher AR are calculated. In general, a large decrease of ACH is observed when the aspect ratio is increased. However, such a decrease becomes slow gradually. Obviously, it is more difficult for the atmospheric flow above a street canyon to penetrate deeply into the inside of a deeper street canyon (namely, with a higher aspect ratio). The findings in this section suggest that on one hand describing the major surroundings in detail is important when assessing natural ventilation performance in buildings and on the other hand the aspect ratio of a street canyon is an important factor influencing the building natural ventilation.

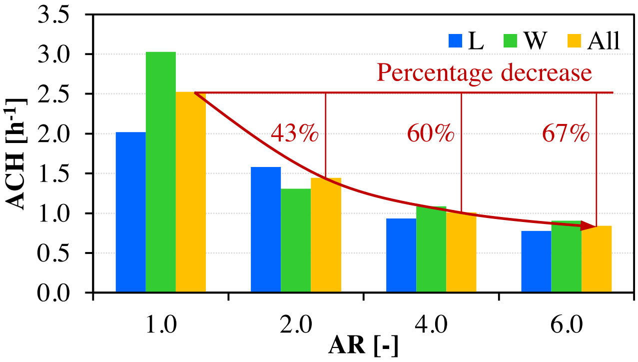


Figure 7: Average ACH values of all rooms at leeward facade (L), windward facade (W) and both facades (All), where the percentage decreases of ACH in comparison to the case of AR = 1.0 are also presented.

Conclusions

Considering four aspect ratios, this study investigates natural ventilation in buildings near a street canyon. Since the atmospheric flow above a street canyon is more difficult to penetrate deeply into a deeper street canyon, ventilation performance of buildings is decreased with the increase of aspect ratio of a street canyon. Compared to the case of AR = 1.0, the percentage decrease of ACH values are, on average, 43%, 60% and 67% for the cases of AR = 2.0, 4.0 and 6.0, respectively. Influenced by flow pattern inside a street canyon, ACH values of rooms along height of a building are not uniformly distributed. Such a distribution varies significantly with the change of aspect ratio. These findings (namely, ACH values and their distributions) suggest that aspect ratio is an important parameter that should be considered when designing natural ventilation of urban buildings.

Acknowledgement

The work described in this paper was financially supported by a grant from the Environment and Conservation Fund of Hong Kong (Project ECF 18/2014).

References

Ai, Z.T., Mak, C.M., Niu, J.L. (2013). Numerical investigation of wind-induced airflow and interunit dispersion characteristics in multistory residential buildings. Indoor Air, 23, 417-429.

Ai, Z.T., Mak, C.M. (2014a). Modeling of coupled urban wind flow and indoor airflow on a high-density near-wall mesh: sensitivity analyses and case study for single-sided ventilation. Environmental Modelling & Software, 60, 57-68.

Ai, Z.T., Mak, C.M. (2014b). Potential use of reduced-scale models in CFD simulations to save numerical resources: Theoretical analysis and case study of flow around an isolated building. Journal of Wind Engineering and Industrial Aerodynamics. 134, 25-29.

Ai, Z.T., Mak, C.M. (2015). From street canyon microclimate to indoor environmental quality in naturally ventilated urban buildings: Issues and possibilities for improvement. Building and Environment. 94, 489-503.

Ai, Z.T., Mak, C.M., (2017). CFD simulation of flow in a long street canyon under a perpendicular wind direction: evaluation of three computational settings. Building and Environment. 114, 293-306.

Fluent. (2010). ANSYS FLUENT 13.0 Theory Guide. Turbuelnce, Canonsburg, PA, ANSYS Inc.

Franke, J., Hellsten, A., Schlunzen, &H., Carissimo, B. (2007). Best practice guideline for the CFD simulation of flows in the urban environment. COST Action 732. COST office.

Gao, C.F., Lee, W.L. (2012). The influence of surrounding buildings on the natural ventilation performance of residential dwellings in Hong Kong. International Journal of Ventilation. 11(3), 297-310.

Georgakis, C., Santamouris, M. (2006). Experimental investigation of air flow and temperature distribution in deep urban canyons for natural ventilation purposes. Energy and Buildings. 38, 367-376.

Li, X.X., Leung, D.Y.C., Liu, C.H. (2008). Physical modelling of flow field inside urban street canyons. Journal of Applied Meteorology Climatology. 47, 2058-2067.

Oke, T.R. (1987). Boundary layer climates. 2nd ed. New York: Routledge.

Snyder, W.H. (1981). Guideline for Fluid Modeling of Atmospheric Diffusion, Meteorology and Assessment. Division Environmental Sciences Research Laboratory. U.S. Environmental Protection Agency, Research Triangle Park, NC.

Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., Shirawasa, T. (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. Journal of Wind Engineering and Industrial Aerodynamics. 96(10-11), 1749-1761.

Wolfshtein, M. (1969). The velocity and temperature distribution of one-dimensional flow with turbulence augmentation and pressure gradient. International Journal of Heat Mass Transfer. 12, 301-318.

Yakhot, V., Orszag, S.A. (1986). Renormalization group analysis of turbulence: 1 Basic theory. Journal of Scientific Computing. 1, 1-51.

Contact email: cheuk-ming.mak@polyu.edu.hk