A PRELIMINARY STUDY OF EFFECT OF THE GREEN FEATURE – WING WALLS ON NATURAL VENTILATION IN BUILDINGS

Cheuk Ming Mak [1], Jian Lei Niu [2], and Kai Fat Chan [3]

[1] Associate Professor, Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong
[2] Associate Professor, Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong
[3] Graduate, Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong
Correspond to becmmak@polyu.edu.hk

ABSTRACT : There is growing consciousness of the environmental performance of buildings in Hong Kong. The Buildings Department, the Lands Department and the Planning Department of the Hong Kong Government issued the first of a series of joint practice notes [1] to promote the construction of green and innovative buildings. Green features are architectural features used to mitigate migration of noise and various air-borne pollutants and to moderate the transport of heat, air and transmission of daylight from outside to indoor environment in an advantageous way. This joint practice note sets out the incentives to encourage the industry in Hong Kong to incorporate the use of green features in building development.

The use of green features in building design not only improves the environmental quality, but also reduces the consumption of non-renewable energy used in active control of indoor environment. Larger window openings in the walls of a building may provide better natural ventilation. However, it also increases the penetration of direct solar radiation into indoor environment. The use of wing wall, one of the green features, is an alternative to create effective natural ventilation. This paper therefore presents a preliminary numerical study of its ventilation performance using Computational Fluid Dynamics (CFD). The numerical results will be compared with the results of the wind tunnel experiments of Givoni.

Key words : Wing walls, Green Feature, CFD, Natural Ventilation, Wind Speed, Wind Tunnel

1. INTRODUCTION

There is growing concern of the environmental performance and sustainability of buildings in Hong Kong. In order to achieve a sustainable building development, the Government of Hong Kong Special Administrative Region has issued the first of a series of Joint Practice Notes [1] in February 2001 to promote the incorporation of green and innovative features into high-rise building design. These green features such as wing walls shown in Figure 1 are used to enhance natural ventilation. Figure 1 shows the provision of wing walls in a building façade vertically between two openings. Natural ventilation is considered to be an effective passive cooling strategy in building design. In 1962 and 1968, Givoni [2,3] conducted experiments on room models with and without wing walls in a wind tunnel so as to study its effect on natural ventilation. He found that single-sided ventilation incorporated with wing walls could greatly improve the internal air circulation compared with that without wing walls. Maximizing the utilization of natural ventilation therefore not only minimize the reliance on active means for environmental control, but also reduces the consumption of non-renewable energy.

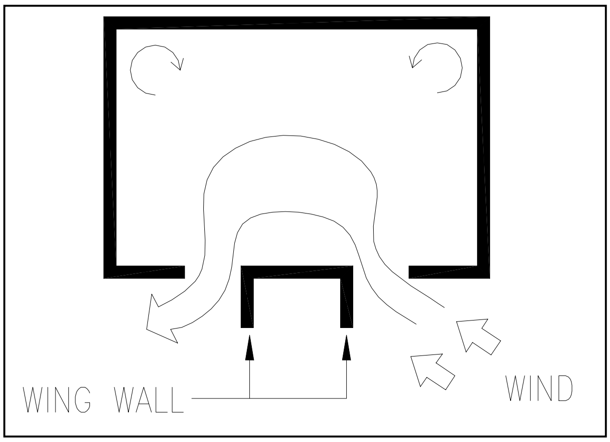

Figure 1. Physical configuration of wing walls

In recent years, Computational Fluid Dynamics (CFD) has been widely adopted to study building airflow and indoor environment [4-10], as well as the wind environment [11]. Several CFD studies in single-sided ventilation [12-13] in building and cross-ventilation [14] have also been reported.

2. EXPERIMENTS OF GIVONI

The experiments of Givoni [2,3] were conducted to study air flow through room models with and without wing walls in a wind tunnel. Figure 2 and Figure 3 show the plan and side elevation of the open-throat type wind tunnel respectively. It consists of an intake section, a working section, a distribution control section, a transit section, a fan and an exit.

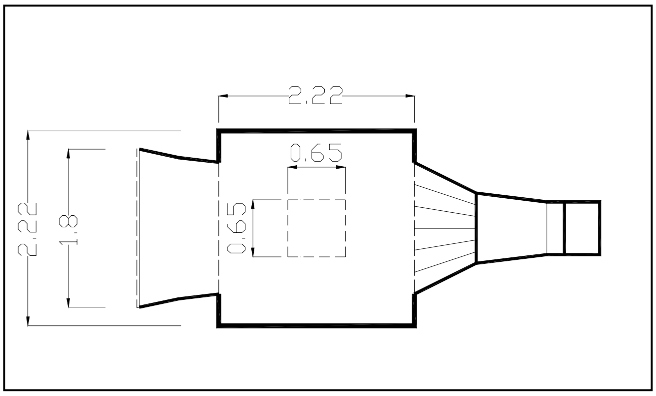

Figure 2. Plan elevation of the wind tunnel experiments

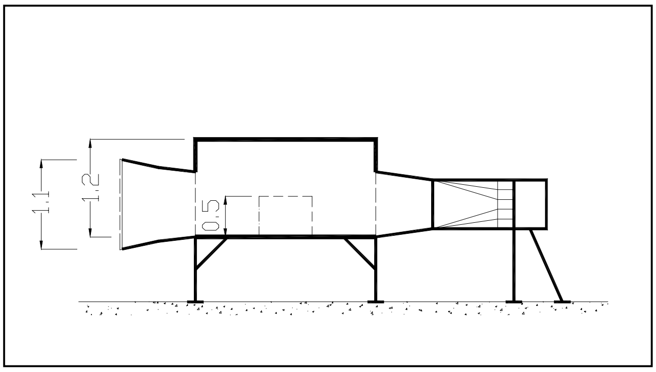

Figure 3. Side elevation of the wind tunnel experiments

It can be seen in Figure 2 and Figure 3 that the intake opening has a cross section of 1.8 x 1.1m(H) and contracts to a cross section of 1.5 x 0.8m(H), over a length of 0.9m. The working section is 2.22x1.2m(H). The distribution control section contracts in cross section from 1.5x0.8m(H) to 0.8x0.6m(H) and contains five vertical louvers whose angle can be adjusted so as to regulate flow distribution. The transit sections are used for connecting the rectangular and circular section.

The room model of dimension of 0.65x0.65x0.5m(H) with a centered window opening shown in Figure 2 and Figure 3 with dashed lines was located at the center of the wind tunnel. The window opening with dimension of 1/3 of that of the wall was created at the center of the wall of the room model facing the air flow.

The wind tunnel was then tested with different air (wind) flow velocities ranging from 1.27m/s to 3.35m/s. Different wind directions were tested with the angle of air flow incidence ranging from 0^o (opening facing the air flow) to 135^o with 22.5^o increment. The average internal velocity measured from the room model was expressed in percentage based the inlet air flow velocity (wind speed).

3. NUMERICAL SIMULATION

3.1 CFD code

Wind tunnel testing is a long-established engineering method in general aerodynamic and hydraulic study. It is expensive to construct such facilities. Besides, it is quite time-consuming to conduct such experimental studies. CFD is an alternative approach to study the performance of green and innovative features in buildings. A commercial CFD code FLUENT has been successfully applied in different research works [15-19]. FLUENT version 6.0 and the pre-processor software GAMBIT version 4.0 were adopted in this work to simulate the experiments of Givoni. The physical and air flow velocity data obtained from the experiments of Givoni were used as input parameters into the software.

3.2 Geometry

There are three major parts in the CFD code. They are: i) pre-processor, ii) solver and iii) post-processor. The pre-processor GAMBIT was applied to create a 2-dimensional physical room model that is based on the experiments of Givoni. Figure 4 shows the geometry of the computational domain. It can be seen in the figure that there is one centered opening (case 1) on the wall of the room model of 0.65mx0.65m. The width of the window opening at the centre of the wall is 0.22m. The wind angle is 0° when the opening on the wall is facing the wind and its normal is in parallel to the wind as shown in Figure 4.

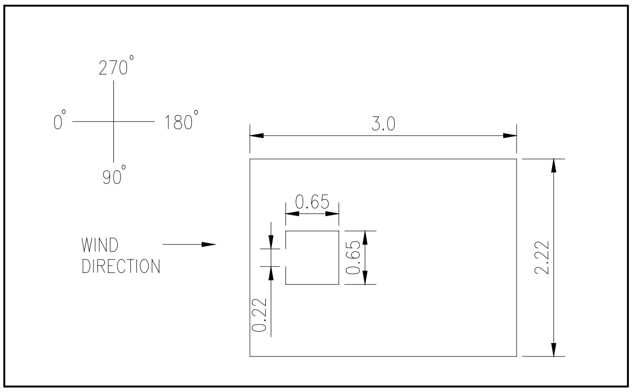

Figure 4. Physical model for computer simulation
(One single-sided window opening)

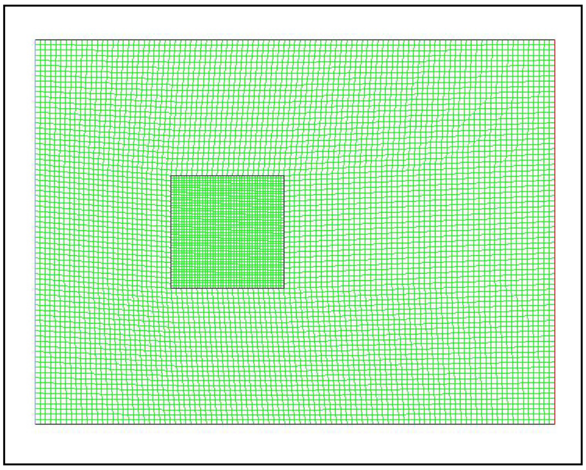

Figure 5. Grids of the simulation model

3.3 Numerical grids, turbulence model and boundary conditions

The FLUENT CFD package is used here in modeling the natural ventilation by solving the conservation equations for mass, momentum and energy using the finite volume method. A simple standard (two-equation) k- ε turbulence model was used though it was inevitable to introduce some errors [20]. The number of uniform structured grids is around 12700. Figure 5 shows the grids of the simulated model. Figure 6 shows the model configurations for all cases at different wind angles. Table 1 shows the different inlet mean wind speeds for all cases. The descriptions of all four cases for the 2-dimensional CFD simulation and the wind tunnel experiments of Givoni are as follows:

Case 1: One centered opening, with 1/3 width of wall, no vertical projections (wing wall)

Case 2: Two lateral openings, with total 1/3 width of wall, no vertical projections (wing wall)

Case 3: Two lateral openings, with total 1/3 width of wall and vertical projections (wing wall) of depth equal to the opening width

Case 4: Two lateral openings, with total 1/3 width of wall and vertical projections (wing wall) of depth double the opening width

Table 1. Different inlet mean wind speeds

Different inlet mean wind speeds (m/s)
1.27
1.68
1.83
2.0
2.95
3.35

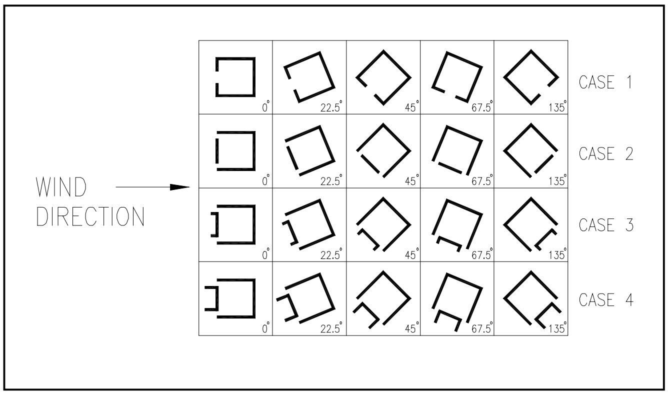

Figure 6. Model configurations at different wind direction (Wind angle ranging 0° and 135°)

4. ANALYSIS OF RESULTS

The CFD results of the 2-dimension models are compared with the experimental results of Givoni [2,3].

4.1 Effect of wing wall

Figure 7 and Figure 8 show the CFD results and the experimental results of Givoni respectively. The result of each point shown in the two figures was obtained by taking average at different wind speeds listed in Table 1. It can be seen in these figures that the percentage of mean indoor air speed inside the room to wind speed is increased by the incorporation of wing wall on the wall facing the wind. Significant improvement of natural ventilation is obtained in the cases with wing wall (i.e. case 3 and 4) in both experimental results and CFD results. The increase in the mean indoor velocity in two cases with wing wall can be reached up to 350% compared with those without wing wall. However, it can be seen that there is no significant improvement of natural ventilation when a longer wing wall is used.

4.2 Performance of wing wall under different wind directions

Figure 7 and Figure 8 show that there are large variations of the percentage of mean indoor air speed to wind speed in case 3 and case 4 under different wind directions. There is a highest value at around 45°. That means although wing wall can increase mean indoor air velocity that would promote natural ventilation, its performance is greatly affected by the wind directions. This result agrees with other findings [18]. The figures show that the wing wall has its best performance at wind angle of around 45^{o}.

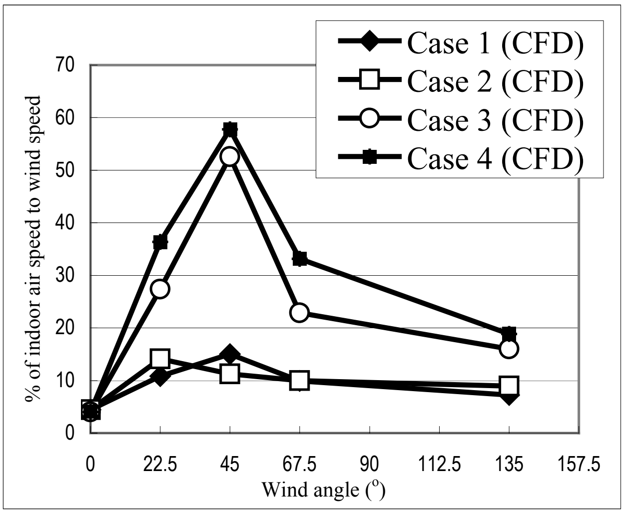

Figure 7. Percentage of mean indoor air speed to wind speed against wind angle ranging 0° and 135° (CFD results)

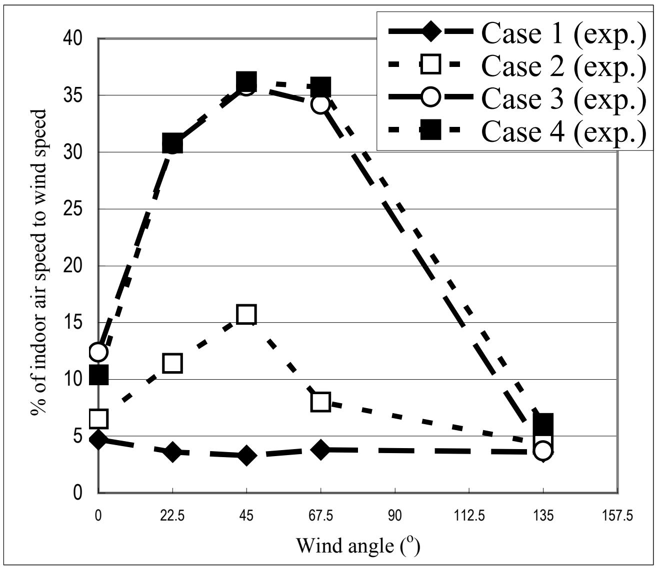

Figure 8. Percentage of mean indoor air speed to wind speed against wind angle ranging 0° and 135° (Experimental results of Givoni)

4.3 Comparison of CFD results and the experiments of Givoni

Figures 9-12 show the comparisons between CFD results and wind tunnel experiments of Givoni for cases 1-4. There are no wing wall in Case 1 and Case 2. Case 4 has a longer wing wall than Case 3.

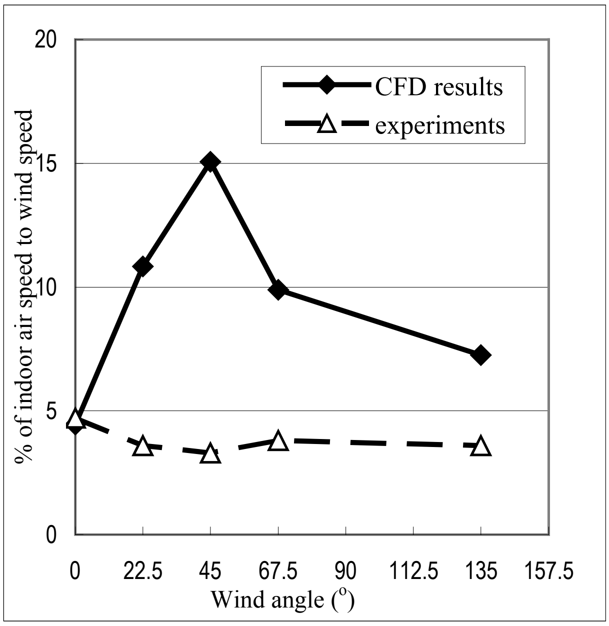

Figure 9. Percentage of mean indoor air speed to wind speed against wind angle ranging 0° and 135° (Case 1)

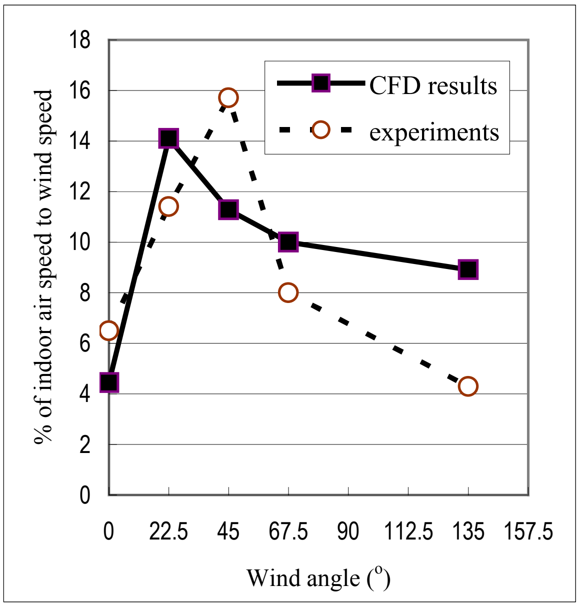

Figure 10. Percentage of mean indoor air speed to wind speed against wind angle ranging 0° and 135° (Case 2)

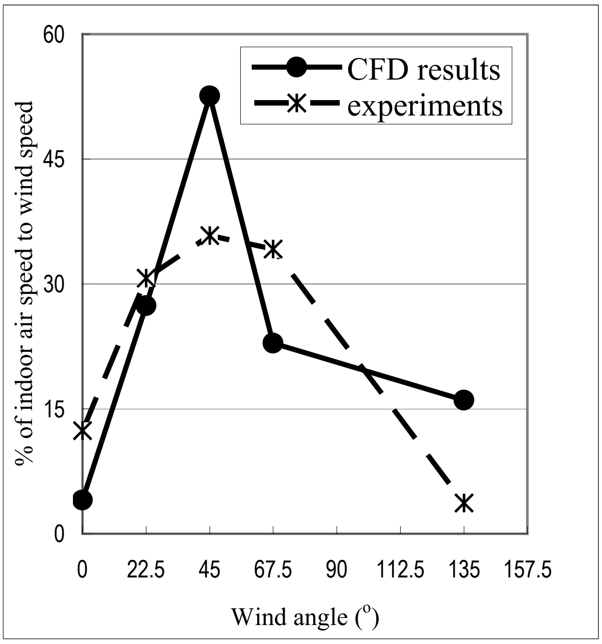

Figure 11. Percentage of mean indoor air speed to wind speed against wind angle ranging 0° and 135° (Case 3)

It can be seen in the figures that the CFD results generally have similar trend to the experimental results of Givoni. However, there are large discrepancies between CFD results and experiments for case 1 (no wing wall). It is found that the maximum error of about 200% occurs in case 1 while the minimum error of about 5% occurs in case 4. The average error obtained in the four cases is around 55%.

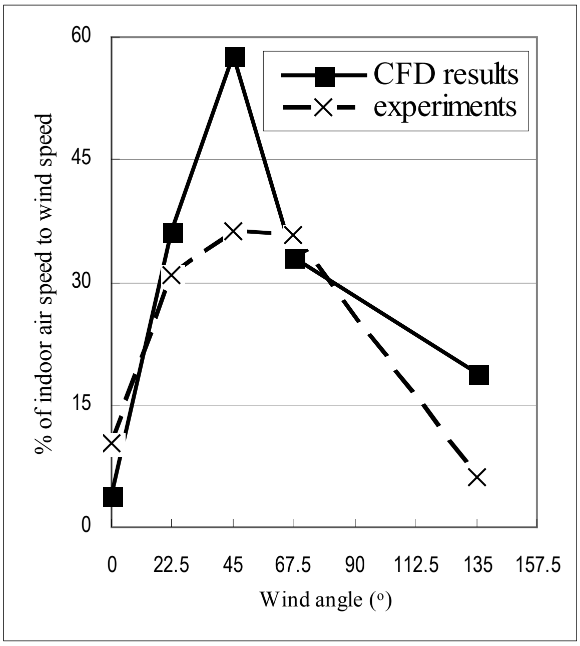

Figure 12. Percentage of mean indoor air speed to wind speed against wind angle ranging 0° and 135° (Case 4)

5. CONCLUSION

CFD technique has been applied to model four different configurations of the room model of the experiments of Givoni in order to study the performance of wing walls in terms of single-sided natural ventilation. Both the CFD results and the experiments of Givoni indicate that wing wall can promote natural ventilation by increasing the mean indoor air speed relative to wind speed at various wind speeds and wind directions. The best performance of wing wall is at the wind angle of around 45°. The study also shows that 2-dimensional CFD simulation produces similar trend to the experimental results though there are some discrepancies. Further work will be 3-dimensional CFD simulation of the experiments of Givoni.

ACKNOWLEDGMENT

The authors would like to acknowledge that the CFD simulation works were conducted by Mr. Chan Kai Fat who graduated in 2004.

REFERENCES

[1] Joint Practice Note No.1. *Green and Innovative Buildings*, Building Department, Lands Department and Planning Department, Hong Kong Special Administrative Region, 2001.

[2] Givoni, B., *Basic study of Ventilation problems in housing in hot countries*, Research Report to the Ford Foundation, Technion, Haifa, Israel: Building Research Station, Israel Institute of Technology, 1962.

[3] Givoni, B., *Ventilation problems in hot countries*, Research Report to the Ford Foundation, Technion, Haifa, Israel: Building Research Station, Israel Institute of Technology, 1968.

[4] Jones, P. J., and Whittle, G. E., "Computational fluid dynamics for building air flow prediction – current status and capabilities", *Building and Environment*, Vol. 27, No. 3, pp. 321-338, 1992.

[5] Niu, J. L., and Kooi, J. V. D., "Two-dimensional Simulation of the Air Flow and Thermal Comfort in a Room with Open-window and Indoor Cooling Systems", *Energy and Buildings,* vol. 18, No. 1, pp. 65-75, 1992.

[6] Chow, W. K., "Application of computational fluid dynamics in building services engineering", *Building and Environment*, Vol. 31, pp. 425-436, 1996.

[7] Chow, W. K., "CFD simulation on balcony spill plume." *Journal of Fire Sciences*, Vol. 16, pp. 468-485, 1998.

[8] Chen, Q., and Srebric, J., "Application of CFD tools for indoor and outdoor environment design.", *International Journal on Architectural Science*, Vol. 1, pp. 14-29, 2000.

[9] Awbi, H. B., "Application of computational fluid dynamics in room ventilation", *Building and Environment*, Vol. 24, No. 1, pp. 73-84, 1989.

[10] Gan, G., and Riffat, S. B., "Naturally ventilated buildings with heat recovery: CFD simulation of thermal environment.", *Building Services Engineering Research Technology*, Vol. 18, No. 2, pp. 67-75, 1997.

[11] Murakami, S., Ooka, R., Mochida, A., Yoshida, S., and Kim, S., "CFD analysis of wind climate from human scale

to urban scale", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 81, pp. 57-81, 1999.

[12] Kiranoudis, C. T., Markatos, N. C., and Papakonstantino, K.A., "Numerical simulation of air flow field in single-sided ventilated buildings", *Energy and Buildings*, Vol. 33(1), pp. 41-48, 2000.

[13] Allocca, C., Chen, Q., and Glicksman, L. R., "Design analysis of single-sided natural ventilation", *Energy and Building*, Vol. 1534, pp. 1-11, 2002.

[14] Kato, S., Murakami, S., Takahashi, T., and Gyobu, T., "Chained analysis of wind tunnel test and CFD on cross ventilation of large-scale market building", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 67&68, pp. 573-587, 1997.

[15] Mak, C. M., and Yik, Francis W. H., "A study of natural ventilation in a kitchen using computational fluid dynamics (CFD)", *Architectural Science Review*, Vol. 45, pp. 1-8, 2002.

[16] Mak, C. M., and Oldham, D. J., "The application of Computational Fluid Dynamics to the prediction of flow-generated noise in low speed flow ducts. Part 1: Fluctuating drag forces on a flow spoiler", *Journal of Building Acoustics*, Vol. 5, No. 2, pp. 123-141, 1998.

[17] Mak, C. M., and Oldham, D. J., "The application of Computational Fluid Dynamics to the prediction of flow-generated noise in low speed flow ducts. Part 2: Turbulence-based prediction technique", *Journal of Building Acoustics*, Vol. 5, No. 3, pp. 199-213, 1998.

[18] Mak, C. M., Cheng, C., and Niu, J. L., "The application of Computational Fluid Dynamics to the assessment of green features in buildings: Part 1: Wing walls", *Architectural Science Review*, Vol. 48, No. 1, pp. 1-14, 2005.

[19] Niu, J.L., Yuen, Y. M., and Mak, C. M., "The application of Computational Fluid Dynamics to the assessment of green features in buildings: Part 2: Communal sky gardens", *Architectural Science Review*, accepted for publication.

[20] Murakami, S., "Current status and future trends in computational wind engineers", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 67-68, pp. 3-34, 1997.