SURROUNDING BUILDINGS AND WIND PRESSURE DISTRIBUTION ON A HIGH-RISE BUILDING

Zhiwen Luo[1,*], Yuguo Li[1], Marcus Rosler[2] and Joachim Seifert[2]

1. *Department of Mechanical Engineering, The University of Hong Kong, Hong Kong, China*
2. *Technical University of Dresden, Dresden, Germany*

* zhwluo@hku.hk

Abstract

The effect of the surrounding lower buildings on the wind pressure distribution on a high-rise building is investigated by computational fluid dynamics (CFD). When B/H=0.1, it is found that the wind pressure on the windward side was reduced especially on the lower part, but for different layers of surrounding buildings, there was no great difference, which agrees with our previous wind tunnel experiment data. Then we changed the aspect ratio from 0.1 to 2, to represent different airflow regimes: skimming flow (SF), and wake interference (WI). It shows that the average Cp increases when B/H increases. For different air flow regimes, it is found that insignificant difference exists when the number of the building layers is more than 2. From the engineering point of view, it is sufficient to only include the first layer for natural ventilation design by using CFD simulation or wind tunnel experiment.

1. Introduction

The use of natural ventilation is highly constrained by its surrounding environments, e.g. a highly dense urban area may reduce the potential of wind-driven natural ventilation. The sheltering effect of the surrounding built-up environment can reduce pressure differences across a building which is necessary to produce adequate ventilation rates. Moreover, for a highly dense city like Hong Kong, a tall building is often surrounded by relative lower buildings. One question arises: when carrying out natural ventilation design in an urban area using CFD simulation or wind tunnel experiment, it is often difficult to determine the minimum amount of surroundings that should be included in the computational or experimental domain. The present paper is trying to answer this question.

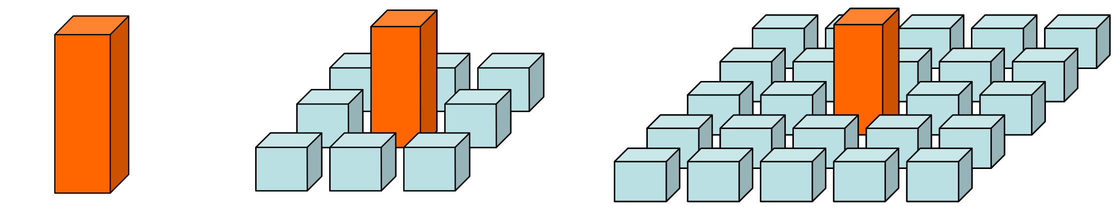
Most existing studies on the sheltering effect are for a single building with the surrounding buildings of the same height and shape. Wiren (1985) performed a wind tunnel study of the wind pressure effects on a 1 1/2-storey single family house surrounded by identical models in various regular arrays. The models used in Wiren's study had a roof pitch angle of 45 degrees. His tests indicated that the maximum reduction in ventilation airflow rate, obtained with three rows of houses surrounding the test house, was about 40%. Chang and Meroney (2003) investigated the sensitivity of high roof suctions on low-rise buildings with multiple surrounding building configurations. Both wind tunnel experiment and numerical simulation were conducted. Sheltering effects produced by the surrounding buildings on the central investigated building were found to be significantly different from the isolated case.

There are also studies considering one building higher than its surroundings. Cheng and Wang (2005) numerically simulated the pedestrian winds in a built-up area by varying the height of the central building, and found that it was difficult to simulate the pedestrian winds when there is a considerable height difference between the buildings. Yoshie et al (2007) reported a cooperative project for CFD prediction of pedestrian wind environment in the Architectural Institute of Japan. For predicting pedestrian winds, in the case of a high-rise building in the urban area, they found that it is practically sufficient to consider at least one layer of surrounding blocks, but no wind pressure data were reported. Most recently, Eipper et al (2007) used a wind tunnel experiment to study the effect of surrounding lower buildings on wind pressure distribution on the central high-rise building with a constant separation distance. They compared wind pressure results and found there was no significant difference between one-layer and two-layer surroundings.

The purpose of the present study is to extend the studies of Eipper et al (2007) into different flow regimes using computational fluid dynamics (CFD) simulations. We will first focus on a single high-rise building and compare the result with the wind tunnel data in Eipper et al 92007) to validate our numerical model. Then, we will consider the different cases of surrounding buildings with varying distances from the central building, representing two different flow regimes, i.e., skimming flow (SF) and wake interference (WI)) respectively. The isolated roughness (IR) situation is not considered here as it is regarded as the single high-rise building case.

2. Physical Model and Numerical Method

There are a lot of high-rise buildings in a populated and densely-built city like Hong Kong. A high-rise building is often surrounded by multiple lower buildings. We consider a simple model with the urban area as arrays of identical building blocks with one high-rise building in the centre as shown in Fig.1. Fig. 1a represents a single high-rise building at the fully exposed condition,. Fig. 1b describes one-layer layout while Fig. 1c with two layers. The different surroundings can be represented by adding different layers.



a) Single high-rise building   b) One-layer layout   c) Two-layer layout

Figure 1. An idealized physical model for a urban area

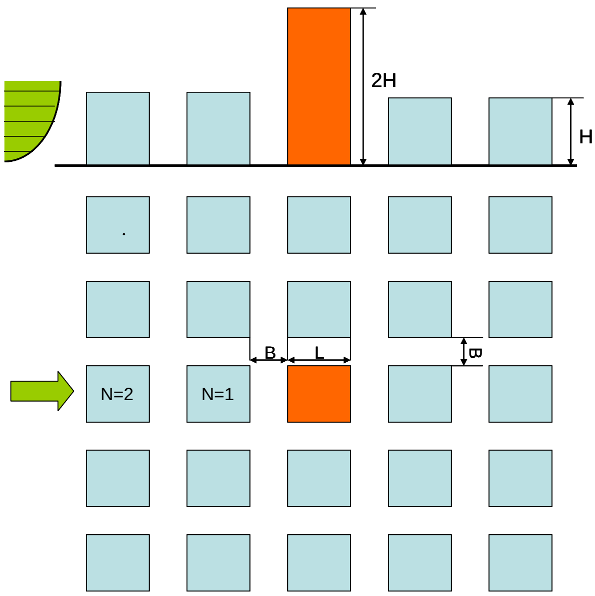


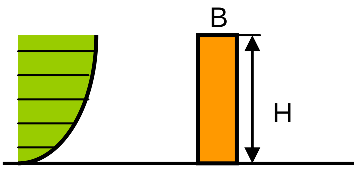
Figure 2. Building arrangement

Fig. 2 shows the plane and section view of the physical model. The middle building has a size of 40m (length) × 40m (width) × 200m (height) in full scale and the surrounding lower buildings have the same footprint as the middle one with reduced height of 100m. The scale used in wind tunnel model is 1:400. The aspect ratio B/H can be changed by varying different B values but keeping H constant. Different aspect ratios of 0.1, 1, 2, 6 are adopted representing different flow regimes in urban area according to Oke's (1998), i.e. skimming flow (SF) with B/H=0-1.2, wake interference (WI) flow with B/H=1.2-5, and isolated flow (IF) with B/H>5. The approaching flow is a typical atmospheric boundary flow with the direction normal to the buildings.

The choice among CFD methods is a kind of compromise between accuracy and the cost. It is known that the standard k-ε turbulence model overestimates the k value in the impinging region and cannot capture accurately separation and reattachment (Murakami 1993), although it requires less time. On the other hands, large eddy simulation (LES) is a more accurate, but not widely used in practice due to its complexity. In the present study, a RNG k-ε turbulence model is used (Seifert et al 2006). The commercial software of Fluent 6.5 is used.

3. Single High-rise Building Case

We first consider the single high-rise building case. The typical model domain with the model building is depicted in Fig. 3.



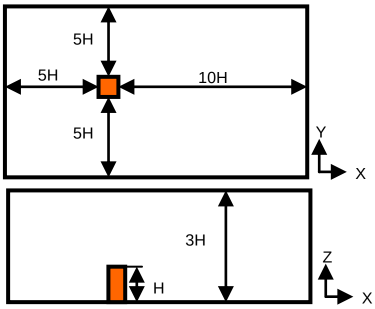


Figure 3 Typical computational model for a single high-rise building

3.1 Boundary Conditions

The power-law approaching wind profile is used to approximate the mean velocity profile by fitting the experimental data:

$$\frac{U}{U_{ref}} = \left(\frac{Z}{Z_{ref}}\right)^{0.16} \tag{1}$$

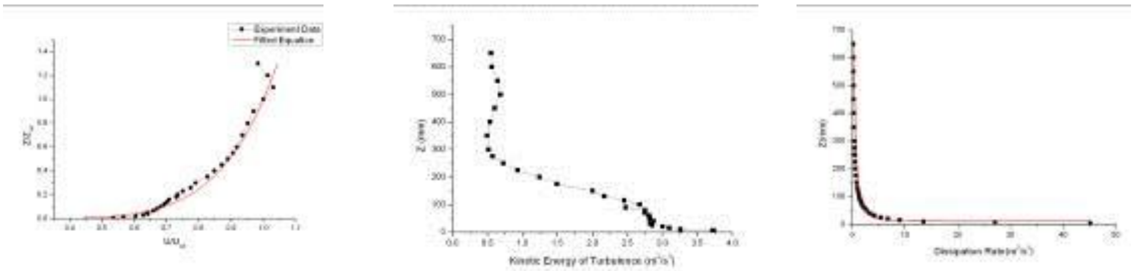
The reference height Z_{ref} is set at the height of the central high-rise building and U_{ref} = 9.54 m/s at the reference height which is in accordance with the experiment data. The experimental data on kinetic energy of turbulence and its dissipation rate at the inlet section are not available, and they are calculated using the following equations:

$$k(z) = \frac{3}{2}(U_{avg} I)^2 \tag{2}$$

$$\varepsilon(z) = \frac{u_*^3}{\kappa z} \tag{3}$$

$$k = \frac{u_*^2}{\sqrt{C_\mu}} \tag{4}$$

where U_{avg} is the mean velocity at inlet, I is the turbulence intensity at different heights, u_* is the friction velocity, κ is the von Kaman constant equals to 0.41 and C_μ =0.09. The inlet boundary conditions are illustrated in Fig. 4. The other boundary conditions are tabulated in Table 1.



a) velocity distribution b) kinetic energy of turbulence c) dissipation rate

Figure 4. Boundary conditions at the inlet

Table 1. The boundary conditions for the computing domain

Inlet	Power-law based on wind tunnel experiment
Outlet	Gauge pressure = 0, Outflow boundary condition is applied
Ground and building surface	Smooth wall using log-law wall function
Top	Free slip, flux normal to the boundary is zero
Lateral sides	Free slip, flux normal to the boundary is zero Symmetric boundary condition is applied

3.2 Grid Refinement and Domain Size Tests

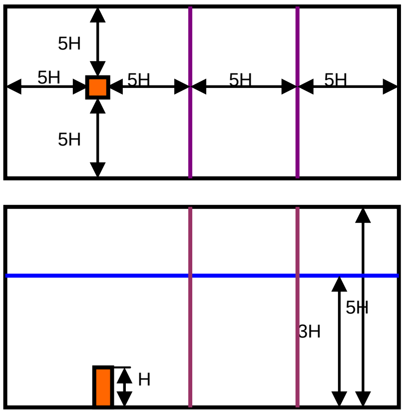
The computation grids and domain are two key elements to determine the accuracy of s CFD simulation. A grid-independent and domain-independent solution is worth seeking before making the comparison with the experimental data. The mesh generation is followed in a way that refined mesh is deployed in the vicinity of the target building and coarse mesh far away from it. Different mesh schemes are tested, shown in Table 2. The calculated wind pressure along the center line is compared in Fig.5 (a). The results show that the mesh2 is sufficient for obtaining a grid-independent solution. Therefore, the mesh2 is used in the subsequent studies.

Three different sizes of flow domain are tested from small to large, as in Table 3. The minimum domain is set to be the same as the tunable table in the wind tunnel test section. The wind pressure along the center line is compared and no significant difference is found among the three domain sizes. Conservatively, the middle domain is selected for this study.

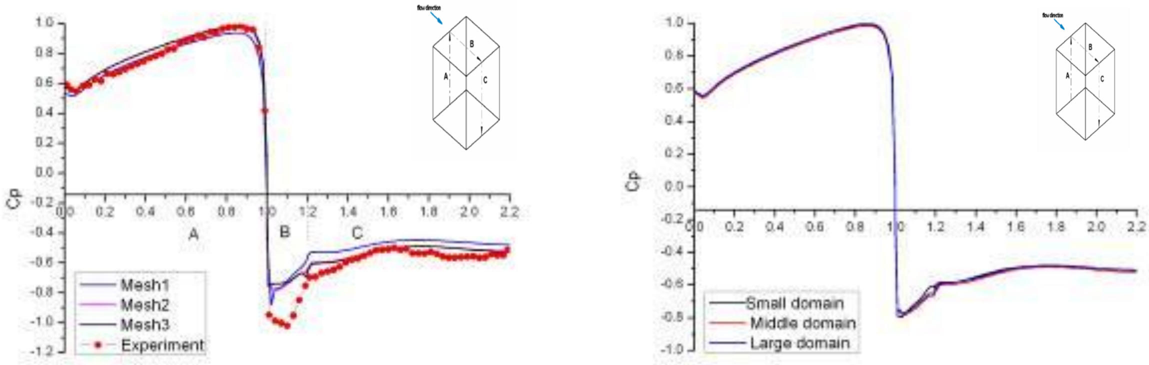
Table 2. Different mesh strategies for grid-independent testing

Mesh strategy	No of Grid cells	First cell near the wall
Mesh1	44,0000	b/7 in X direction, b/10 in Y and Z direction
Mesh2	55,6000	b/15 in X direction, b/20 in Y and Z direction
Mesh3	67,5000	b/30 in X direction, b/40 in Y and Z direction

Table 3. Different computational domain size



Flow domain	Domain Size
Domain A	$(10H+b)\times(10H+b)\times 3H$
Domain B	(10H+b)×(15H+b)×3H
Domain C	(10H+b)×(20H+b)×5H



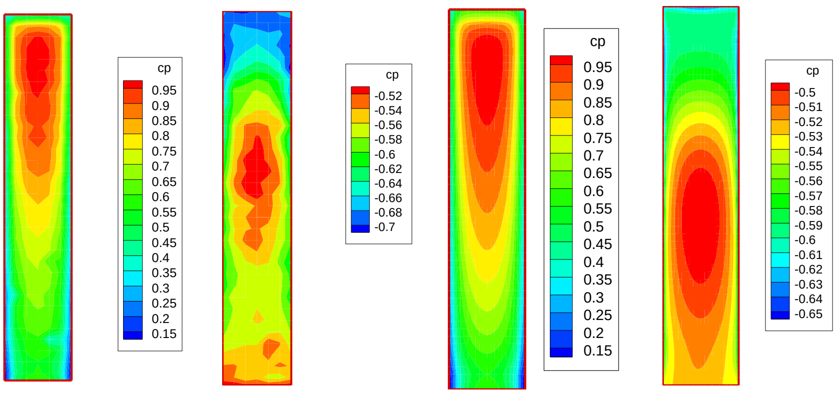
a) Grid refinement test results

b) Domain size test results

Figure 5. The effect of grid refinement and domain sizes on the center line wind pressure coefficient distribution.

3.3 Wind Pressure Comparison

By adopting mesh2 and domain size 2, the computed wind pressure contours on the windward and leeward façade of the central building are compared in Fig. 6. The wind pressure on the windward façade is well reproduced by the simulation, but larger discrepancy is found on the leeward. This discrepancy may be attributed to the limitation of the RNG turbulence model used here. The results confirm the accuracy and reliability of the present numerical method.



Front Back

(a) Experimental result

Front Back

(b) CFD result

Figure 6 Wind pressure contour on the windward and leeward sides of the building

4. Effect of Different Surroundings on the Wind Pressure

The computed and measured wind pressure coefficients are compared in Fig. 7 for B/H=0.1 in one-layer surroundings. The computed results agree well with the measured ones except for the roof. Compared with the fully exposed case in Fig. 5a, it is seen that the shielding effect from the one-layer surroundings is quite obvious, especially on the lower part of the high-rise building. The largest divergence on C_p could be as much as 0.8 at the height ratio of 0.3.

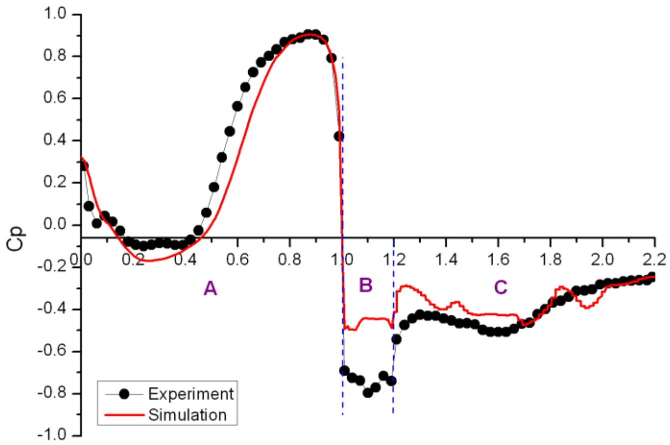


Figure 7. Wind pressure comparison for *B*/*H*=0.1 for one-layer surroundings

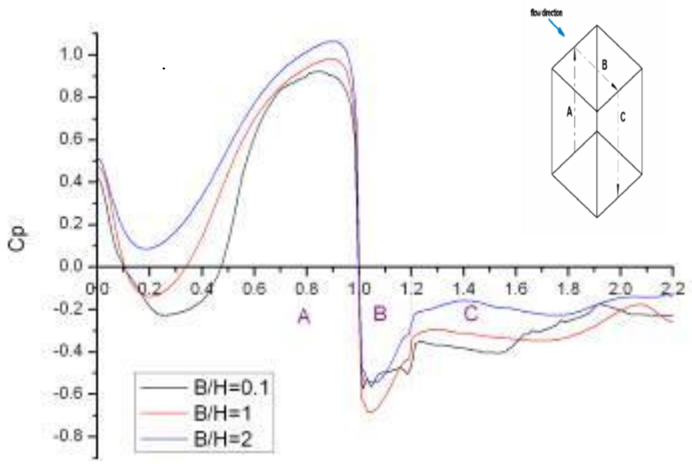
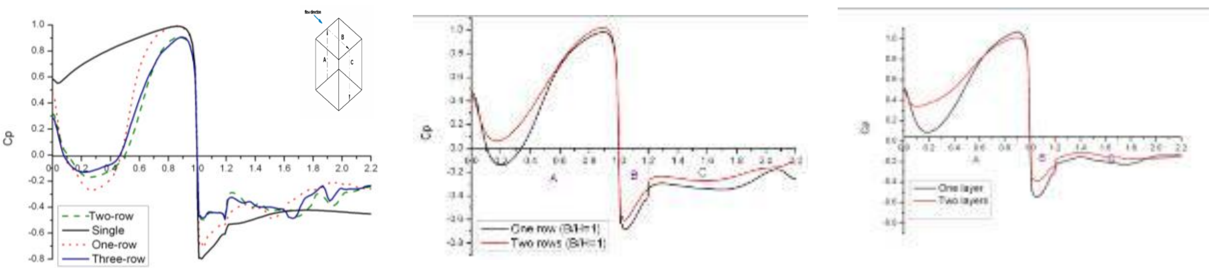


Figure 8. Wind pressure distribution on the center line under different aspect ratios

Different aspect ratios (*B*/*H*=0.1, 1, 2) are considered for one-layer surrounding buildings. The effect of aspect ratio on the wind pressure distribution on the central high-rise building is shown in Fig. 8. As expected, the shielding effect highly depends on the aspect ratios. The average C_p increases when B/H increases. That indicates that the sheltering effect of surrounding buildings is reduced when the separation distance increases as expected.



(a) B/H=0.1 (b) B/H=1 (c) B/H=2

Figure 9. C_p comparison for different layers when B/H=0.1, 1 and 2.

Fig. 9 examines the effect of different layers when the aspect ratio B/H=0.1, 1 and 2, respectively. There is very little difference between the results with two layers and three layers. But from the engineering's point of view, where the highest accuracy is not desired, it is reasonable to consider only the first layer in the simulation. This generally agrees with Lam et al (2006)'s conclusion from wind tunnel test that "effects on environmental wind conditions of a building being located in a row were largely confined to the first two building members."

5. Limitation of the Present Study

In our present study, only the regularly-spaced buildings and normal wind direction are considered. In the real urban context, the arrangement of the buildings differs from case to case and wind direction changes from time to time. It would be more realistic and helpful to include more representative arrangements usually found in the real urban area and take the different wind directions into account.

6. Conclusions

We attempted to understand the sheltering effect of the surroundings on the surface wind pressure distribution, which is needed for natural ventilation design. We considered a special case of urban area with a high-rise building in the centre surrounded by multiple layers of identical lower buildings. CFD simulation is applied to compute the wind pressure distribution along of the middle high building and we try to evaluate how much surroundings should be included for a reasonable prediction of the wind pressure coefficient. We conclude:

- The effects of the surroundings significantly reduce the surface pressure coefficients, especially when the width of the street canyon is small. The average C_p increases when aspect ratio B/H increases.
- For different air flow regimes, it is found that insignificant difference exists when the number of the building layers is more than 2. From the engineering point of view, it is sufficient to only include the first layer for natural ventilation design by using CFD simulation or wind tunnel experiment.

7. Acknowledgements

The work described in this paper was supported by a grant from the Research Grants Council of the Hong Kong Special Administrative Region, China (Project No. HKU 7154/05E).

8. References

Chang, C.H. and Meroney, R.N. 2003. The effect of surroundings with different separation distances on surface pressures on low-rise buildings. Journal of Wind Engineering and Industrial Aerodynamics, 91, 1039-1050

Cheng, H. and Fan, W. 2005. Using a CFD approach for the study of street-level winds in a built-up area. Building and Environment, 40, 617-631

Seifert, J., Li, Y., Axley, J. and Rosler, M. 2006. Calculation of wind-driven cross ventilation in buildings with large openings. Journal of Wind Engineering and Industrial Aerodynamics, 94, 925-947

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

Murakami, S. 1993. Comparison of various turbulence models applied to a bluff body. Journal of Wind Engineering and Industrial Aerodynamics, 46-47: 21-36

Yoshie, R., Mochida, A., Tominaga, Y., Kataoka, H., K.Harimoto, T and Shirasawa, T. 2007. Cooperative project for CFD prediction of pedestrian wind environment in the Architecture Institute of Japan. Journal of Wind Engineering and Industrial Aerodynamics, 95(9-11): 1551-1578

Eipper, T., Hildebrand,V., Rosler, M., Seifert, J. and Li, Y. 2007. Interaction between natural ventilation and flow around a multi-storey building. Roomvent 2007

Oke, T. 1998. Street design and urban canopy layer climate. Energy and Buildings, 11: 103-111

Wiren, B.D. 1985. Effects of surrounding buildings on wind pressure distribution and ventilation heat losses for single-family houses. Part1:1 1/2-storey detached houses. The National Swedish Institute for Building Research, Gavle, Sweden, Report No. M85:19