Numerical study on wind-pressure characteristics of a high-rise building in group of buildings

Wang Dayang\textsuperscript{a}, Zhou Yun\textsuperscript{a}, Li Qingxiang\textsuperscript{b}, Deng Xuesong\textsuperscript{a}, Zhu Yong\textsuperscript{a}

\textsuperscript{a}School of Civil Engineering of Guangzhou University, Guangdong Guangzhou, China
\textsuperscript{b}Guangdong Provincial Academy of Building Research, Guangdong Guangzhou, China

ABSTRACT: Wind pressure characteristics of a twin-tower high-rise structure disturbed by surrounding buildings were investigated using large eddy simulation (LES). The computational simulation technique were described initially. Studies have been performed in detail for the mean and fluctuating pressure coefficients. Detail explanations of each analysis results were given in the paper. To study further on the pressure coefficients on the building surfaces, parameter studies on shape coefficient and spatial correlation were performed and investigated subsequently. The numerical obtained in this study are expected to provide practicing engineers a better understanding of wind field around buildings.

KEYWORDS: Numerical prediction; CFD; Pressure coefficient; Disturbance effect; Spatial correlation, High-rise building

1 INTRODUCTION

Computational fluid dynamics has been a widely-used method of wind flow simulation around buildings for more than 20 years. Rapid development is observed especially in recent studies. Traditional standard turbulence models, such as eddy viscosity-based models and various second-order stress models, were found to be inadequate in predicting unsteady flows around bluff bodies (Castro and Graham, 1999; Cowan, 1997; Lim et al, 2009; Leschziner, 1993). Unsteady techniques such as discrete vortex methods and, particularly, large eddy simulation (LES) are found to be much more appropriate for simulating unsteady flow features (Murakami and Mochida, 1995; Shah and Ferziger, 1997; Murakami, 1998; Fasel et al, 2002; Nozawa and Tamura, 2002; Fureby, 2007; Tominaga, 2008). Rodi (1997) compared the performance of LES and Reynolds-Averaged Navier-Stokes (RANS) calculations of vortex-shedding flow passing a square cylinder at Reynolds number, Re=22,000 and that of the 3D flow passing a surface-mounted cube at Re=40,000. Results showed that turbulence fluctuations were severely underestimated by RANS, while LES was found more suitable and had great potential for complex flow calculations. Validation of LES in predicting flow around an obstacle under turbulent flow condition was also confirmed by Nozawa and Tamura (2002). Breuer et al (2003) investigated the variations of the predicted results among RANS, DES (detached-eddy simulation) and LES for the separated flow around a flat plate at high incidence. The RANS computations were not able to capture the unsteady vortex shedding behaviour and both two- and three-dimensional RANS predictions led to the same steady-state results. The asymmetric vortex shedding motion was well reproduced by DES and LES. But the free shear layer originating from the leading edge of the plate was not well reproduced by DES. Kose and Dick (2010) presented a numerical study of the flow around a cubical building by RANS, hybrid RANS/LES and LES in Re=4\times106. Results obtained by LES were the most accurate for coarse grid simulations. Detailed comparison between LES and experiment of both the inflow boundary layer and the flow field around a cube
was performed by Lim et al. (2009). Results confirmed that LES is a viable tool for use in wind engineering problems concerning flow over isolated bodies.

It is well-known that there is rarely a case that only one single high-rise building is built in a city district. Many more regional groups of high-rise buildings are built in major cities, which cause structural wind characteristics become much more complex because of interaction and obstruction effects among buildings. Moreover, existing study results on wind characteristics of flow around single building and also some design parameters stipulated in international design codes may not be applicable to buildings under such wind condition. So, the structural wind characteristics influenced by surrounding buildings have been a subject of interest to researchers and engineers because of its significance for the evaluation of building designs. Zhou (2011) studied wind characteristics around a tall structure, which is disturbed by surrounding buildings, using experimental and numerical methods. Results showed that pressure coefficients were greatly disturbed by surrounding buildings. Cheng et al. (2003) discussed the predictive performance of LES with various dynamic subgrid-scale models for a fully developed turbulent flow around a matrix of cubes. Su and Chen (2006) predicted the wind-induced surface pressures and wind environment around a complex-shaped high-rise building with annex by the standard k-ε model and the RNG k-ε model. Ma et al. (2007) investigated the wind environment around a single building and a building complex based on and the Reynolds averaged N-S equations and the RNG k-ε turbulence model. Wang et al. (2003) predicted wind-induced pressures on a low-rise single house with gable roofs and a building complex consisting of six low-rise houses with gable roofs by numerical simulation.

This paper aims to assess the wind pressure characteristics of a twin-tower high-rise structure under the interaction and obstruction effects of surrounding buildings. The study results will assist engineers and researchers having a better understanding of the risks involved, such as extreme pressure and unfavorable wind direction, when designing a similar twin-tower. To achieve this, the present paper contains CFD. The CFD simulations were performed using LES method. Studies on LES predictions were made in terms of the mean pressure coefficients for a number of pressure analysis around the outer and inner surfaces of the twin-tower building. Studies were also carried out for disturbance effect and spatial correlation of the fluctuating pressure and interaction and obstruction effect. In addition, the reliability of LES method in simulating flow around group of buildings was also revealed. So, the CFD analyses were investigated to assess the capability and accuracy of the LES method in simulating the wind flow in a group of building complex.

2 COMPUTATIONAL SIMULATION TECHNIQUE

2.1 Computational model
In this study, a 1:300 scaled numerical model was designed. In order to eliminate flow obstacle effect on the inflow and outflow boundary conditions (Murakami, 1998), the length of inflow, outflow and side-flow of the computational domain are 5H, 12H and 5H, respectively, as shown in Fig.1 (H is the maximum height of the bluff bodies, H=0.75m). The total height of the computational domain is 3H. The reason for such a choice is to eliminate the flow obstacle effect on the inflow and outflow boundary conditions, and keep the blockage ratio less than 3% as discussed by Murakami (1998). The Reynolds number based on the width L and U(z) (Inflow velocity at z=H) is about $5.4 \times 10^5$. Both structured and unstructured grids were used for mesh generation, as shown in Fig. 1. For zones near the bluff body, an unstructured mesh was generated while for zones outside the unstructured mesh, the structured mesh was applied. One
important advantage of this arrangement is that it is convenient to generate a mesh fine enough in
the neighborhood of the building surfaces while keeping the mesh in far field zones from the
building surfaces unchanged or in a proper coarser mesh. ADINA (2005) platform was adopted
in this study, which has the capacity of manage both structured and unstructured grids in its
solver. The average distance of the first layer of nodes from the surface is about $L/5000$ in $X$
direction and $B/5000$ (value of $L$ and $B$ can be found in Fig. 1(a)) in $Y$ direction with same
growth factor of 1.2, which are smaller than the values suggested by Murakami (1998) (i.e.
$D/1000$) and Huang (2006) (i.e. $D/4000$), so as to ensure that the wall unit $y^+<5$ is satisfied. The
maximum grid size used far away from the building is about $H$. Slip boundary conditions are
imposed on the velocity near the solid boundaries. The Reichardt wall-law is then used to derive
the shear stresses caused by the presence of the wall (Hinze, 1959; Camarri et al., 2002).
Advantage of this wall law is that it can describe the velocity profile not only in the laminar sub-
layer ($y^+<5$) but also in the logarithmic region of a turbulent boundary layer ($y^+\geq 40$) and in the
intermediate region, which will guarantee correct asymptotic behaviour at the wall of the SGS
terms in the Smagorinsky model. Totally, about 800,000 3D grid elements were generated in the
present simulation. Computations are carried out on 16 Xeon E5580 3.2GHz CPUs and require
about 16GB of Infineon memory and about 8 days of CPU time for the whole simulation. The
numerical time step for the transient simulation is $5\times 10^{-4}$ sec and 8000 steps, which is equivalent
to 4 seconds, are needed to collect data for computational stability purposes. The statistical
average of flow field was taken for the last 4000 steps (i.e. 2 sec).

(a) Computational domain (Unit: m)

(b) Grid distribution
2.2 Boundary conditions

Reasonable boundary conditions will be helpful of saving computational time. For the inflow boundary condition, both the mean and fluctuating wind velocity were considered. The mean wind velocity can be achieved by matching a log-law profile (Obasaju, 1992):

$$v(z) = 2.5v^* \log_{e} \left( \frac{z}{40Z_0 + D} \right)^{0.25} \left( \frac{z-D}{Z_0} \right) < 40$$

$$v(z) = 2.5v^* \log_{e} \left( \frac{z-D}{Z_0} \right) \quad 40 \leq \frac{z-D}{Z_0} \leq 1000$$

(1)

where, $v(z)$ is the horizontal wind speed at $z$, $v^*$ is the surface friction velocity defined as $\left( \frac{\tau}{\rho} \right)^{0.5}$ ($\tau$ being the surface shear stress), $D$ is the height of the zero plane above ground, $Z_0$ is the surface roughness length parameter. Height of the gradient wind is 450m.

Based on a procedure by Kraichnan (1970) for generation of an isotropic continuous flow field satisfying a spectrum of Dirac function, a new general inflow turbulence generation method for large eddy simulation, also called the discretizing and synthesizing random flow field generation (DSRFG) technique, is proposed by Huang et al (2010) and Li et al (2007). The isotropic fluctuating velocity in $X$ space was synthesized by the following superposition of harmonic functions (Huang et al., 2010):

$$u_i(X,t) = \sum_{n=1}^{N} \left[ p_{ni}^* \cos(k_i^*x_i + \omega_i t) + q_{ni}^* \sin(k_i^*x_i + \omega_i t) \right]$$

(2)

$$p_{ni}^* = \epsilon_{imn} \epsilon_{ji}^* k_m^*, \quad q_{ni}^* = \epsilon_{imn} \epsilon_{ji}^* k_m^*, \quad \epsilon_{ji}^*, \epsilon_{ji}^*, \omega_i \in N(0,1), \quad k_m^* \in N(0,1/2)$$

Where $k$ and $\epsilon$ are the turbulence kinetic energy and the turbulence dissipation rate, respectively, $\epsilon_{imn}$ is the permutation tensor used in vector product operation, $k_i^*$ and $\omega_i$, respectively, represent a sample of $n$ wave-number vectors and frequencies of the modeled turbulence spectrum. Detailed calculation about the Eq. (2) can be found in Huang et al (2010). The first superiority of the DSRFG is that continuity condition, $\nabla \cdot \mathbf{u} = 0$, can be ensured strictly. The second is that fluctuating velocity generated in inflow condition satisfies specified power spectrum density function. And the last one is that spatial correlation of inflow condition can be adjusted by scaling factor.

At the outflow, convective boundary condition is applied for velocity and pressure:

$$\frac{\partial u}{\partial t} + c \frac{\partial u}{\partial x} = 0$$

(3)

where $c$ is taken to be the bulk velocity so as to ensure global mass conservation. Slip condition was assumed for velocity on the two sides and top surfaces of the computation domain. With this kind of boundary condition, one has $\frac{\partial U}{\partial y} = \frac{\partial U}{\partial w} = V = W = 0$. And non-slip condition was used for velocity on the ground surface, that is $u_n=0$. Neumann condition for pressure was employed on surfaces of building and the computation domain as pressure gradient orthogonal to the surfaces equals to zero.
3 DATA PROCESSING

Pressure data obtained from computational simulation is processed using equations as shown in Table 1. External pressures were normalized with respect to the mean wind pressure at the reference height of $H$. Instantaneous pressure coefficients were determined at each pressure tap location and subsequently analyzed to determine maximum, minimum, mean, and standard deviation of pressure coefficients. Shape coefficient describes distribution status of static pressure on a building surface in wind action, which is influenced mainly by the shape and size of a building. Spatial correlation of fluctuating wind represents pressure dependency of two pressure taps, which is an important index of studying spatial flow status and corresponding transmission pattern.

Table 1  Data processing equations

<table>
<thead>
<tr>
<th></th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wind pressure coefficient</td>
<td>$C_{pi}(n) = \frac{p_i(n)}{\rho U_{ref}^2/2}$ $n=1\cdots N$</td>
</tr>
<tr>
<td>Mean wind pressure coefficient</td>
<td>$\bar{C}<em>p = \frac{1}{N} \sum</em>{n=1}^{N} C_{p_i}(n)$</td>
</tr>
<tr>
<td>Fluctuating wind pressure coefficient</td>
<td>$C_{pi, rms} = \sqrt{\frac{1}{N-1} \sum_{n=1}^{N} (C_{p_i}(n) - \bar{C}_p)^2}$</td>
</tr>
<tr>
<td>Shape coefficient</td>
<td>$\mu_s = \frac{\sum C_{p_i}A_i}{A}$</td>
</tr>
<tr>
<td>Correlation coefficient of fluctuating wind</td>
<td>$\rho_{ij} = \frac{\mu_{ij}^2\mu_{p_i}^2\mu_{p_j}^2}{C_{pi, rms}C_{pj, rms}}$</td>
</tr>
</tbody>
</table>

As shown in Table 1, $i$ is the number of pressure tap; $N$ is the sample length of the wind pressure time series, $N=4096$; $\rho$ is air density, $\rho=1.2$ kg/m$^3$; $V_{ref}$ is wind velocity of the reference height; $z_i$ is the height of the $i^{th}$ pressure taps; $\mu_{p_i}$ is wind pressure coefficient of reference height; $\mu_{p_i}$ is wind pressure coefficient of the $i^{th}$ pressure taps; $\mu_{p_i}$ is the shape coefficient of the $i^{th}$ pressure taps; $A_i$ is the covered surface area of the $i^{th}$ pressure taps; $A$ is the total surface area.

4 RESULTS AND DISCUSSION

4.1 Mean pressure coefficient

The mean pressure coefficient contours obtained from numerical simulations results are shown in Fig. 2. It is well-known that the mean pressure coefficients on the windward face are largely positive for isolated building (Huang et al., 2006; Gomes et al., 2005). However, in this study the mean pressure coefficients were greatly affected by the interaction and obstruction effects between the main and sub-main towers and that of the upstream buildings. For example, the mean pressure coefficients of the main tower on the front face were negative on the lower level for $0^0$ wind incident direction, attributing to the influence of the sub-main tower. The mean pressure on the side and back faces were also greatly disturbed, which are discussed in detail in the next sections.
4.2 Fluctuating pressure coefficient

Fluctuating pressure coefficients of the different taps on front faces are largely of similar values, except those at the corner, as shown in Fig. 3, which means that flow status on the front faces is relatively stable. At the corner, the fluctuating pressure coefficients increase as stronger vortex motion generates. On the side faces, larger fluctuating pressure coefficients relative to those on the front faces can be observed, especially for the region at the corner, due to the flow separation. The reason is that strong and orderly vortex motion appears on the side faces and greater vortex energy is then aroused, which causes the increase of the fluctuating pressure. On the back faces, the variation of the fluctuating pressure coefficients are complex relatively. The reason is that irregular and disorder vortex is generated in wake flow, which causes the pressure on back faces variable and no precise laws can be used to describe the pressure status.

![Figure 2. Mean pressure coefficient contours](image)

(a) 0°  (b) 90°
4.3 Interaction and obstruction effect

To research the interaction and obstruction effect of among buildings, interaction and obstruction factor $\gamma$ is defined as:

$$\gamma = \frac{\mu_s}{\mu_{sn}}$$

(4)

Where: $\mu_s$ is shape coefficient with the interaction and obstruction effect, $\mu_{sn}$ is shape coefficient without the interaction and obstruction effect.

Variation of the interaction and obstruction factor along the height of both the main and sub-main towers in different wind directions is shown in Fig. 4. It can be easily found in Fig. 5 that the maximum shape coefficients of the front faces of the twin-tower building is 0.77, which is close to the value of 0.8 stipulated in the Chinese code (2002). However, greater discrepancy have been noticed for the shape coefficients of back and side faces when comparing to the Chinese wind code (2002). The maximum shape coefficients were -1.25 and -0.83 for the side and back faces respectively, which were higher than the values of the Chinese code (i.e. -0.7 and -0.5, respectively). So, it can be deduced that noticeable difference of pressure distribution on building faces exist relative to signal building (no interaction and obstruction effect exist in signal building).
4.4 Spatial correlation of fluctuating pressure

Spatial correlation of fluctuating wind represents pressure dependency of two pressure taps, which is an important index of studying fluctuating pressure characteristic and also spatial flow status. The lateral correlation coefficient contours of fluctuating wind pressure are given in Fig. 6. As revealed in these figures, the correlation coefficients decrease gradually as the distance of two pressure points increases. The correlation coefficient is influenced by vortex structure and its motion status of the surrounding flow field. Regular vortex structure (such as vortex shape and dimension) and similar motion status will contribute to enlarge the correlation coefficient in condition of the same distance of two points. E.g., two-point correlation in the middle region is relatively larger than that of the side region of the front faces in same distance.

5 CONCLUSIONS

The maximum mean pressure coefficients are found to be at the middle of windward surface with height of about 0.8H-0.9H in 24 wind directions, and the minimum pressure generates at the corner of flow separation. Flow with strong and orderly vortex motion yields large fluctuating pressure, such as the fluctuating pressure on side faces and especially at the corner of flow.
separation. On the contrary, smaller fluctuating pressure yields if irregular and disorder vortex motion exist, such as the pressure on back faces.

Interaction obstruction effect of shape coefficients of the twin-tower building is mainly from the interference disturbance between the main and sub-main towers, and the disturbance effect of surrounding buildings. In this study, the disturbance effect of surrounding buildings on the fluctuating pressure of the twin-tower building is smaller than that of the interference disturbance between the main and sub-main towers.

The lateral spatial correlation of fluctuating pressure among pressure taps on the front and side faces are relatively larger than that of the back faces. As for the spatial correlation between two faces, larger correlation can only be found at the corner between the front and side faces. The vertical correlation decreases with increasing of distance between two pressure taps.

6 ACKNOWLEDGEMENTS

This project is supported by the Doctor Special Fund of Higher School (200810780001), Natural Science Foundation of Guangdong Province (8351009101000001), Production and Research Project of Ministry of Education of Guangdong Province (20090912) and Innovation Team Project of Education System of Guangzhou City (09T003).

7 REFERENCES
