Building and Environment 132 (2018) 30–44

Contents lists available at ScienceDirect

Building and Environment

journal homepage: www.elsevier.com/locate/buildenv

Experimental and numerical study on mean pressure distributions around an isolated gable roof building with and without openings

Fengda Xing, Damith Mohotti*, Kapil Chauhan

School of Civil Engineering, Faculty of Engineering and IT, The University of Sydney, Darlington, NSW 2006, Australia

A R T I C L E I N F O

Keywords:
Gable roof structures
Wind tunnel study
Computational fluid dynamics (CFD)
Steady RANS.A

A B S T R A C T

In this study, the influence of opening position and wind direction on pressure distribution around isolated gable roof buildings with and without openings were investigated. Computational Fluid Dynamics (CFD) simulations were performed and compared with wind tunnel experimental results. One of the aims of this study was to develop well-defined design guidelines for typical gable roof structures, identifying critical localised pressure rises due to different wind attack angles. Existing international standards provide limited design guidelines for localised wind pressure distribution of such buildings, especially in cases where there are openings. Wind tunnel experiments were conducted to obtain mean pressure distributions of critical areas of the building under different wind directions with four different opening configurations, namely, an enclosed building, a building with one windward opening, a building with one windward opening and one sidewall opening, and a building with one windward opening and two sidewall openings. A CFD-based numerical simulation approach was used to create a virtual wind tunnel in the computational domain. Sensitivity analyses for grid resolutions and turbulence models of simulations were performed for the building with a windward opening.

Results from the numerical simulations show good agreement with the experimental results on pressure distribution. The validated models were used to identify critical areas of the buildings that must be considered in the design stage. The relevant pressure coefficients are presented and compared with the standards. The importance of performing a comprehensive wind study using a numerical approach or wind tunnel tests is highlighted.

1. Introduction

When a low-rise building undergoes severe weather conditions such as from a tornado or thunderstorm, the high pressure around the building and flying debris caused by the high winds can easily damage fragile parts of the building (e.g., doors and windows), which means that an enclosed building will turn into a building with openings. This can change the external and internal pressure distributions on the surfaces of the building and rises in localised pressure are observed. Current wind design standards have not included sufficient provisions to cope with this possibility.

The transient response of internal pressure brought by the sudden opening has been the focus of various studies [2–4]. Ginger and Letchford [5] not only studied internal pressure fluctuation, but also took net pressure into consideration and compared this with Australian wind load standard AS 1170.2:1989 [6]. The results of this study showed that the net pressure from AS 1170.2:1989 was conservative for sealed buildings but underestimated buildings with openings. However, their study was limited to a windward opening, which was further developed by Sharma and Richards [7], who also added a corner wall opening and compared the net pressure with both the provisions in the Australian/New Zealand wind loading code AS/NZS 1170.2:2002 [8] and the American wind loading code ASCE7-02 [9]. Sharma and Richards (2005) also found a highly correlated relationship between fluctuating internal and roof external pressure, which showed good agreement with a previous study by Beste and Cermak [10]. In more recent studies, Guha, Sharma [11] investigated a variety of factors influencing fluctuating internal pressure, including building volumes, opening sizes and wind speeds, using a covariance integration approach in a wind tunnel. The results were used to empirically develop design equations of influence factors for low-rise buildings with a dominant opening. The internal pressure dynamics in a building with multiple openings on a single wall was also studied experimentally and compared with a single wall configuration by Guha, Sharma [12]. From a review of these studies, it can be summarised that internal pressure fluctuation and its relationship with external pressure has attracted

* Corresponding author.
E-mail address: damith.mohotti@sydney.edu.au (D. Mohotti).

https://doi.org/10.1016/j.buildenv.2018.01.027
Received 20 November 2017; Received in revised form 13 January 2018; Accepted 15 January 2018
0360-1323/

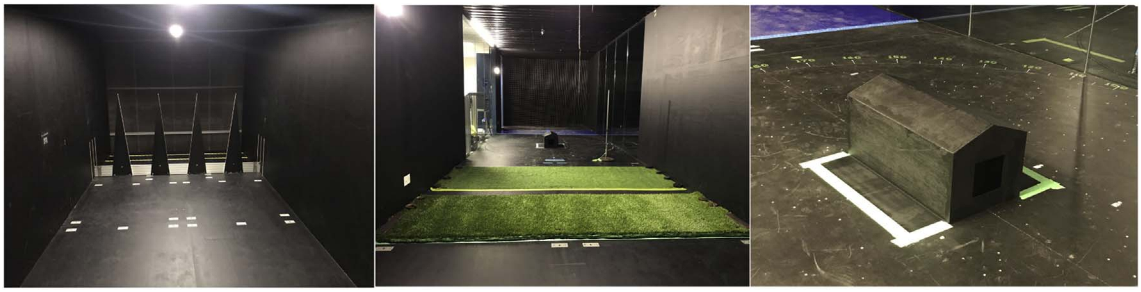

Fig. 1. (a) Spires; (b) carpets to create boundary layer; (c) model of building with windward opening on the turntable in the wind tunnel.

attention from researchers; however, there is a lack of study on the influence of the number of openings and opening position on external and internal pressure. Further, the wind direction is another topic of concern, especially in high wind-prone areas such as coastal cities where the wind direction can be a critical variable in severe weather conditions. When the wind direction is oblique to the edges of a building, the high suction caused by the corner vortex can cause serious damage to the roof. This topic has also caught the attention of wind engineers who have investigated the effect of suction on different kinds of roofs [13–15].

Some studies have taken the CFD-results into consideration in order to validate its applicability to pressure prediction under oblique wind directions. Tamura, Kawai [16] employed both the $k-\varepsilon$ model and large eddy simulation (LES) for turbulent flows on a low-rise building (breath:depth:height = 1:1:0.5). They suggested use the $k-\varepsilon$ model for complex flows around a building, and LES for unsteady problems. He and Song [17] investigated roof corner vortex of the Texas Tech University (TTU) building with large eddy simulation and concluded that the simulation could capture fluctuating eddies at all scales and would be close to wind tunnel data if the mesh was fine enough. The unsteady characteristics and unsteady motions of the conical vortex were studied by Ono, Tamura [18] using LES and a flow visualisation technique, which also found a good match between the simulation and wind tunnel results. Generally, LES is preferred by wind engineers for the study of corner vortex due to the advantage of this simulation in unsteady flow problems. However, the disadvantages of LES are also very obvious, including expensive computational costs, complexity of simulation techniques, and the absence of best practice guidelines [19]. These limitations mean that LES cannot replace Reynold-averaged Navier-Stokes equations (RANS) as the most widely used simulation approach in wind engineering. RANS has been applied with a satisfactory degree of accuracy in many areas, such as pedestrian-level wind [20,21], natural ventilation [22,23], and wind-driven rain [24,25]. The whole flow field data could be provided at a reasonable computational cost and several sets of guidelines have been developed for RANS over the past 15 years, which increase the confidence of practical applications [19,20]. The family of $k-\varepsilon$ turbulence models has been shown to have some deficiencies on pressure predictions under oblique wind directions [16,26,27]. In this study, the performance of the $k-\omega$ shear stress transport (SST) turbulence model, which combines the advantages of both the $k-\varepsilon$ and $k-\omega$ turbulence models [28,29], will be evaluated under various wind directions. If the accuracy of $k-\omega$ SST turbulence model could be proven on pressure prediction around low-rise buildings, this simulation method is believed to be a better alternative for building design than wind tunnel experiments and LES at the present stage.

The study presented in this paper mainly focuses on the change of pressure distribution around buildings brought by different opening positions and wind directions of a gable roof building based on the assumption that fragile parts of the building have been broken in severe weather conditions. A boundary layer wind tunnel is used in this study in conjunction with numerical simulations performed with steady RANS. In this study, an isolated gable roof building is selected to explore the influence of opening position and wind direction on the mean pressure distribution around the building. Section 2 presents a detailed description of the wind tunnel experiments. Numerical settings are discussed in Section 3, followed by the results and discussions on the sensitivity analyses for turbulence models and grid resolutions in Section 4. In Section 5, the performance of CFD is evaluated by comparing the results with the wind tunnel data.

2. Experimental setup

The experiments were carried out in the Boundary Layer Wind Tunnel within the school of civil engineering at the University of Sydney. The test section of the boundary layer wind tunnel is 20 m long, 2.5 m wide and 2 m high. The atmospheric boundary layer was created by a combination of spires as shown in Fig. 1(a), and grass carpets as shown in Fig. 1(b). The building models were constructed from plywood, which was painted black to give a smooth surface, as shown in Fig. 1(c). The models were placed on the turn-table with scales near the edge, which are used to record the exact rotation angle of the turntable. Different wind attack angles could be achieved by rotating the turn table. The wind direction is defined as 0° when the direction is parallel to the roof ridge. Six directions were investigated in this study, ranging from 0° to 75° at 15° intervals. The building models were made at a scale of 1:20 and had the dimensions $0.25\ \text{m}\times 0.5\ \text{m}\times 0.25\ \text{m}\ (W\times L\times H)$ corresponding to $5\ \text{m}\times 40\ \text{m}\times 5\ \text{m}\ (W\times L\times H)$ in full-scale. Four different models were investigated including an enclosed model, a model with one windward opening, a model with one windward opening and one sidewall opening, and a model with one windward opening and two sidewall openings, as shown in Fig. 2. The opening positions were selected considering the typical locations of windows and doors which are vulnerable to flying debris and high suction pressure. All openings are of the same size ($0.1\ \text{m}\times 0.1\ \text{m}$). The dimensions of the model with windward opening and the locations of external and internal pressure taps are shown in Fig. 3. The same dimensions and pressure tap distributions were utilised for the other three models. The mean wind velocity and turbulence intensity of the approaching flow were measured with a hot-wire probe. Time averaging was conducted for a period of 120 s with a sampling rate of 1000 Hz. Fig. 4 shows the measured mean wind velocity and turbulence intensity profiles as well as fitted lines, which will be used as the simulation inlet conditions of this study. The aerodynamic roughness length was estimated to be 0.0003 (0.006 in full-scale). The reference point was selected at the model height ($z_{ref}=H$), with the reference mean velocity $U_{ref}=10.7\ m/s$ and a reference turbulence intensity of 18%. The relative uncertainty in the measurements of mean velocity was around 3.5%, and mean pressures have relative uncertainties of less than 6.7% at the reference point [30]. The reproducibility of experimental results could be achieved within these uncertainty ranges.