Building and Environment 143 (2018) 240–257

Contents lists available at ScienceDirect

Building and Environment

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

Study on localised wind pressure development in gable roof buildings having different roof pitches with experiments, RANS and LES simulation models

Fengda Xing, Damith Mohotti*, Kapil Chauhan

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

ARTICLE INFO

Keywords:
Gable roof structures
Wind tunnel study
Computational fluid dynamics (CFD)
Steady RANS
Large eddy simulation

ABSTRACT

In this study, the influence of different roof pitches on the mean pressure distributions around isolated buildings subject to different wind directions were investigated with wind tunnel tests, 3D steady Reynolds-averaged Navier-Stokes (RANS) simulation and large eddy simulation (LES). Wind tunnel experiments were conducted to obtain mean pressure distributions around buildings having three commonly used roof pitches, namely, 1:5, 2:5 and 3:5. The critical high suction areas created by conical vortices were identified, and the influence of roof pitch on these critical areas with high localised pressures was investigated under various wind directions. In addition, computational fluid dynamics (CFD) analysis was performed, and the performance of RANS and LES were evaluated and compared with the results measured from the wind tunnel tests by considering both the accuracy of results and computational cost. The assessment on RANS and LES was used to determine the best guidelines for the flow problems involved in this study.

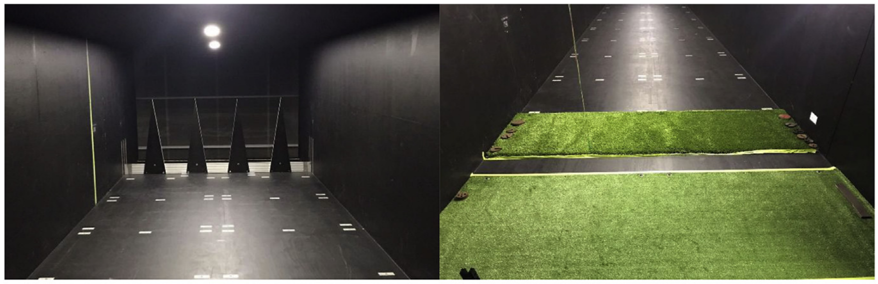
Results from the RANS show good agreement with the experimental results on pressure distribution when the building is subject to perpendicular wind directions. However, a significant improvement was found using LES over RANS in the prediction of near-building flow field and localised pressure under oblique wind directions but with an increased computational cost by a factor of more than 80. Furthermore, the high suction pressures are observed to be more critical on buildings with a lower roof pitch under both perpendicular and oblique wind directions. This indicates that a low roof pitch should be applied with caution especially in windstorm-prone areas.

1. Introduction

Wind is one of the most critical factors that needs to be considered in the design of a building. When wind approaches a building, the localised pressure on the building's surfaces can be affected by many parameters such as building geometry and wind direction. This is especially critical for low-rise buildings such as canopy-roofed warehouses. As wind loadings are originally due to the obstruction effect from a building, it will vary significantly with different building geometries. Wind direction can also bring remarkable variations on pressure distribution around a building, especially under oblique wind attack angles where conical vortices can easily occur. The large suctions caused by conical vortices can cause severe damages to roofs and should be given sufficient attention during building design. Among the different factors related to building geometries, the roof pitch is a well-known parameter that can significantly influence the flow pattern around a building. The pressure distribution around a building with different roof pitches has been extensively investigated over the last several decades [1–3]. Hoxey et al. [4] investigated gable roof models with roof angles of 14° and 26°, and summarised that the pressure changes brought by building geometries were not reflected in current wind loading design codes. Xu and Reardon [5] tested three hipped roof models with 15°, 20° and 30° roof angles and found that the 30° hip roof model experienced the highest suction at the roof corner. Prasad et al. [6] conducted a more comprehensive study and took gable, hipped and flat roof buildings into consideration. A critical roof angle of 45° was found for gable and hipped roof buildings that had the best performance, and the peak suction pressures on the roof were also reduced by 85% and 91%, respectively, compared to a flat roof building. The conical vortices were not included because only perpendicular wind directions were considered in these studies. Due to the high suction pressures brought by conical vortices under oblique wind directions,

* Corresponding author. School of Civil Engineering, Faculty of Engineering and IT, The University of Sydney, Darlington, NSW, 2006, Australia.
E-mail address: damith.mohotti@sydney.edu.au (D. Mohotti).

https://doi.org/10.1016/j.buildenv.2018.07.026
Received 15 May 2018; Received in revised form 14 July 2018; Accepted 17 July 2018
Available online 18 July 2018



(a) (b)

Fig. 1. (A) Spires; (b) carpets at the inlet of the wind tunnel to create boundary layer.

pressure distributions, especially on the roof, can be very complicated and even more critical than under perpendicular wind directions. There was a large number of investigations related to conical vortices on different building geometries such as a flat roof building [7], curved roof building [8], and saddle roof building [9]. The motion of corner vortices has been proved to be closely related to the wind direction as well as the geometry of the building model. Based on these abundant previous studies, it is found that the influence from the roof pitch on the wind flow and pressure distribution around low-rise buildings under both perpendicular and oblique wind directions has rarely been explored [10]. Most of the studies on roof pitch are limited to perpendicular wind directions. It is necessary to conduct more comprehensive studies and identify the critical areas, especially the high suction, caused by conical vortices on building roofs with different roof pitches, which is also of great importance to ensure safe building design [11].

In addition, wind tunnel tests were used in these studies to determine the pressure coefficient. With the rapid development of CFD (computational fluid dynamics) over the last few decades, numerical results from CFD are gaining increasing attention and acceptance from researchers. Typically, Reynolds-averaged Navier-Stokes (RANS) simulation has been the most widely used method, mainly because of its reasonable computational cost and well-developed best practice guidelines [12]. This approach has wide applications on the study of flow characteristics around low-rise buildings, with a satisfactory degree of accuracy in more recent studies [13–15]. However, the disadvantages of RANS simulations still cannot be neglected, especially on the modelling of complex flow and the unsteadiness of flow structures compared to large eddy simulation (LES) [12,16]. The large eddies, which contain most of the turbulent energy, can be directly computed by LES but can only be modelled using turbulence models in RANS [17]. Therefore, LES is theoretically a better numerical tool for the modelling of turbulent and transitional flows than RANS. Some previous studies have assessed the different performances from RANS and LES on transitional flow problems. Tamura et al. [18] suggested use of $k-\varepsilon$ model for complex flows around a low-rise building and LES for unsteady problems. Tominaga and Stathopoulos [19] evaluated the performance of LES and RANS on the dispersion problem around an isolated cubic building, and found that LES could always give better results than RANS on the distribution of concentration. Van Hooff et al. [20] presented a validation of cross-ventilation flow through a generic enclosure with five different RANS turbulence models and LES. It was concluded that the transient feature could be better captured by LES, resulting in a better reproduction of all the measured parameters, including velocity, turbulent kinetic energy and volume flow rate. Generally, LES could provide better results than RANS on transient flow problems but with considerably higher computational cost. However, the number of this type of publication which includes both RANS and LES is very limited. Especially for buildings under oblique wind directions, a transient flow problem with high turbulence, there are few related investigations on the difference in performance between RANS and LES. Although literature could be widely found on this topic, the research methods are always limited to a single simulation method, or only experimental study [10,21,22]. The different performances of RANS and LES have never been compared and validated by experimental data in pressure prediction when the buildings are under oblique wind attack angles. The relative study is meaningful to determine a suitable CFD approach for this flow problem, especially considering that some discrepancies from RANS simulation results have been found with the wind-tunnel measurements in previous investigations [23,24].

The study presented in this paper mainly focuses on the change of pressure distribution around buildings brought by different roof angles, especially high suction on the roof under oblique wind directions. A boundary layer wind tunnel is used in this study in conjunction with numerical simulations performed with steady RANS and LES. In this study, three gable roof models with three different roof pitches, namely, 1:5, 2:5 and 3:5, are selected to explore the influence of roof pitch 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 grid resolutions and the lengths of sampling time in Section 4. In Section 5, simulation results will be compared and validated with experimental results from wind tunnel tests.

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). Fig. 2 shows the building models with roof pitches of 1:5, 2:5 and 3:5, respectively. The models were constructed from plywood with smooth surfaces. They were placed on the turn-table with scales near the edge, which are used to record the exact rotation angle of the turn-table. 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 and 90° when the direction is perpendicular to the roof ridge, as shown in Fig. 3. Seven directions were investigated in this study, ranging from 0° to 90° at 15° intervals. The building models were made at a scale of 1:20 and the height of the eave H was kept constant for all the models. The scaled-down models had the dimensions $0.25\ \text{m}\times 0.5\ \text{m}\times 0.2\ \text{m}\ (W\times L\times H)$ corresponding to $5\ \text{m}\times 10\ \text{m}\times 4\ \text{m}\ (W\times L\times H)$ in full-scale as shown in Fig. 3. The total heights of the building models were 0.225 m, 0.250 m and 0.275 m, respectively, for roof pitches 1: 5, 2: 5 and 3: 5, corresponding to the heights of 4.5 m, 5.0 m and 5.5 m in full-scale. A total of 90 pressure taps were used on the external surfaces, and the pressure tap distribution is shown in Fig. 4. The same 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