Building and Environment 84 (2015) 204–213

Contents lists available at ScienceDirect

Building and Environment

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

Air flow around isolated gable-roof buildings with different roof pitches: Wind tunnel experiments and CFD simulations

Yoshihide Tominaga [a, *], Shin-ichi Akabayashi [b], Takuya Kitahara [a], Yuki Arinami [b]

[a] Department of Architecture and Building Engineering, Niigata Institute of Technology, 1719 Fujihashi, Kashiwazaki, Japan
[b] Environmental Science and Technology, Graduate School of Science and Technology, Niigata University, 8050, Ikarashi 2-no-cho, Nishi-ku, Niigata, Japan

ARTICLE INFO

Article history:
Received 4 October 2014
Received in revised form
8 November 2014
Accepted 12 November 2014
Available online 21 November 2014

Keywords:
Gable roof
Roof pitch
Computational fluid dynamics (CFD)
Wind tunnel experiment
Steady RANS

ABSTRACT

The air flow around isolated gable-roof buildings with different roof pitches was investigated by wind tunnel experiments and computational fluid dynamics (CFD) simulations based on a steady Reynolds-averaged Navier–Stokes equations (RANS) model. Firstly, wind tunnel experiments on the air flow around building models with three different pitches, specifically, 3:10, 5:10, and 7.5:10, were conducted to create a measurement database of the time-averaged velocity, turbulent kinetic energy, and pressure coefficient around the building. Next, sensitivity analyses for the grid resolutions and turbulence models of the CFD simulations were performed for the 5:10 roof pitch model. The performance of the CFD simulation with the selected grid resolution and turbulence model was examined and validated by comparing the results of the simulation with the measured data for all the roof pitches. Generally, for the streamwise velocity, the simulation results were found to be in good agreement with the measured values, with an average deviation of less than 15%. For points behind the building, however, the prediction accuracy showed as much as 30% deviation. This discrepancy was closely related to the fact that the transient fluctuations caused by vortex shedding around the building are not reproduced by the steady-RANS simulations used in this study. Finally, the effect of the roof pitch on the flow field around the building was investigated using the CFD simulations. We clarified that the difference in the flow fields of the 3:10 and 5:10 roof pitches is large, relative to the difference between the 5:10 and 7.5:10 pitches.

1. Introduction

It is well known that a roof's pitch has a major influence on the flow environment surrounding a building. To determine the pressure coefficient, which is important to structural and ventilation design, wind tunnel experiments to examine buildings with different roof pitches have been widely conducted [1–3]. From the perspective of the urban environment, it has been shown that the roof pitches of buildings significantly affect the pollutant environment in street canyons [4–8]. Moreover, the influence of roof pitches on air flow [9–11], heat transfer [12], and snow drifting [13] around buildings has been examined. Although these problems have been investigated mainly by physical experiments in previous decades, numerical approaches based on computational fluid dynamics (CFD) are commonplace nowadays. Furthermore, even if we are not interested in the roof pitch itself, it is becoming more common for CFD analyses to be applied to complex building configurations that include inclined roofs [14–17].

To confirm the applicability of CFD analyses, the validation and verification processes are crucial [18,19]. To validate the application of CFD analysis to the wind flow around a building, generic building models with flat roofs (basically cubes and cuboids) have usually been used [20–22]. There have been very few studies that have examined the applicability of CFD analysis when applied to the air flow around a building with a pitched roof [23], although some studies have applied the pressure coefficient for validation [14]. This can be attributed to the fact that there have been very few detailed studies that have obtained the mean velocity and turbulent statistics around a building with a pitched roof. Recently, Ntinas et al. [24] applied a direct numerical simulation (DNS) to the prediction of the air flow around obstacles with arched and pitched roofs in a wind tunnel. Although the computational results were examined by comparing them with the mean velocity values obtained by particle image velocimetry (PIV), the Reynolds number

* Corresponding author. Tel./fax: +81 (0) 257 22 8176.
E-mail address: tominaga@abe.niit.ac.jp (Y. Tominaga).

http://dx.doi.org/10.1016/j.buildenv.2014.11.012
0360-1323/

was quite low ($Re = 1284$, based on the height of the obstacles and the inlet velocity) and the flow field was two-dimensional. These conditions would be very basic from an engineering aspect. Therefore, it is important for detailed wind tunnel results to be available to evaluate the applicability of CFD analysis to the air flow around a pitched-roof building within a turbulent boundary layer. Furthermore, for various applications of interest, e.g. design of photovoltaic panels [23,25], it would be useful to clarify the influence of the roof pitch on the air flow and pressure fields, as well as their features, based on experiments and CFD simulations.

In this study, the air flow around isolated gable roof buildings with different roof pitches was investigated by wind tunnel experiments and CFD simulations based on a steady Reynolds-averaged Navier–Stokes equations (RANS) model. As the most fundamental condition, only wind striking the facade perpendicularly was considered. Firstly, wind tunnel experiments to determine the air flow and pressure fields around building models with three different roof pitches, namely, 3:10, 5:10, and 7.5:10, were conducted to create a measurement database. Next, sensitivity analyses of the grid resolutions and turbulence models for the CFD analyses were performed for the 5:10 roof pitch. Based on the selected grid resolution and the turbulence model, the flow and pressure fields for the three roof pitches were predicted by applying CFD. The performance of the steady RANS CFD analysis was examined and validated by comparing with the measured data. Finally, the effect of the roof pitch on the air flow field around a building was clarified using the CFD results.

2. Materials and methods

2.1. Flow field

A schematic view of the building model used for the tests is shown in Fig. 1. An isolated gable-roof building for which the height of the eaves $H_e = 6$ m and the width $W = 6.6$ m was placed such that it was perpendicular to the approaching flow. The model could be given one of three different roof pitches, specifically, 3:10 (=16.7°), 5:10 (=26.6°) and 7.5:10 (=36.9°). These roof pitches were drawn from traditional Japanese architecture. H_e was identical for all three versions of the model.

2.2. Wind tunnel experiments

The experiments were carried out in the wind tunnel at Niigata Institute of Technology. The test section of the boundary layer wind tunnel is 13 m long, 1.8 m high, and 1.8 m wide. A combination of spires and surface roughness was used to create an approach wind profile. The mean streamwise velocity of the approaching flow obeyed a power law with an exponent of 0.25, which corresponds to a suburban terrain. The vertical profiles of the mean velocity and turbulent kinetic energy k in the approaching flow are shown in Fig. 2. The aerodynamic roughness length was estimated to be $z_0 = 1.0 \times 10^{-4}$ m for the profile of the approaching flow. A 1:30 scaled model, i.e., $H_e = 200$ mm, was adopted. Although the model size is rather large relative to the roughness length, priority was given to improving the resolution of the velocity measurement around the building. The velocity at H_e, i.e., U_{He}, was 2.6 m/s. A specific dynamic scale was not considered, although care was taken to ensure that the Reynolds number was sufficiently high. The Reynolds number based on H_e and U_{He} was approximately 35,000.

The wind velocity was measured by a split fiber probe (SFP) (Dantec Dynamics; 55R55) and a CTA module (Dantec Dynamics; 90C10), which could identify the three-dimensional components of a velocity vector. Time averaging was conducted for a period of 60 s to obtain statistical values with a sampling rate of 1000 Hz. Additionally, PIV measurements were also conducted to obtain the instantaneous and time-averaged flow field for the 5:10 roof pitch. The PIV system consisted of a Nd:YAG laser (with a wavelength of 532 nm and a maximum power of 2 W) and a CCD camera (Photron FASTCAM SA3) with a 1024 × 1024 pixel resolution. The laser was mounted above the building model and was positioned to create a laser sheet in the vertical center plane. The CCD camera was positioned perpendicular to the plane of the laser sheet. The maximum frame rate of the CCD camera was 500 fps. This means that each frame consisted of two interlaced video fields with an interval of 2.0 ms. A smoke generator (Dainichi Porta Smoke; PS-2002) was installed far enough upwind of the building model to enable the evaporated oil "smoke" to be sprayed into the wind tunnel through the generator's nozzle. The size of the oil droplets was approximately 50 μm. The PIV post-processing was provided by image-interrogation and post-interrogation systems. A direct autocorrelation technique was used to determine the mean displacement of any one particle between consecutive images and to calculate the flow velocity vectors. The interrogation window measured 27 × 27 pixels. The mean velocity was obtained by combining 8000 frames, corresponding to 16 s in real time, of the vector fields. Furthermore, the static pressures on the building surfaces were measured using a multi-point transducer (Kyowa Electronic Instruments; F94-2206).

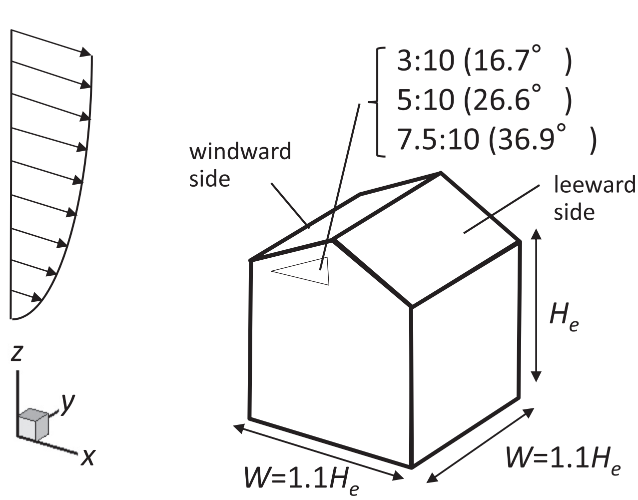

Fig. 1. Schematic view of model.

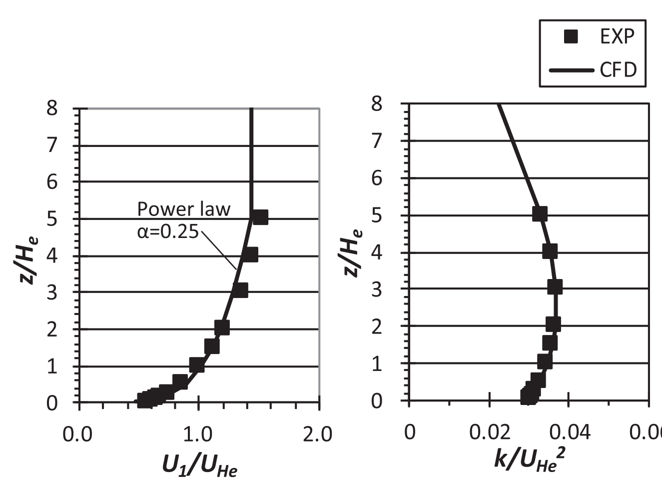

Fig. 2. Vertical profiles of mean velocity U_1 and turbulent kinetic energy k in the approaching flow.