Building and Environment 95 (2016) 63–74

Contents lists available at ScienceDirect

Building and Environment

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

Wind flow over the low-rise building models with gabled roofs having different pitch angles

Y. Ozmen [a,*], E. Baydar [a], J.P.A.J. van Beeck [b]

[a] Department of Mechanical Engineering, Karadeniz Technical University, Trabzon 61080, Turkey
[b] Environmental and Applied Fluid Dynamics Department, von Karman Institute, Brussels B-1640, Belgium

ARTICLE INFO

Article history:
Received 3 June 2015
Received in revised form
28 August 2015
Accepted 9 September 2015
Available online 12 September 2015

Keywords:
Gable roof
Roof pitch
Low-rise building
Wind tunnel experiment
Turbulence modeling

ABSTRACT

In this study, the turbulent flow fields on the low-rise building models with gabled roofs having different pitch angles immersed in atmospheric boundary layer have been investigated experimentally and numerically. The models of the Belgian Building Research Institute (BBRI) test building with a scale of 1:100 were studied with 15°, 30° and 45° roof pitches for the wind direction of 90°. Flow visualization, measurements of velocity and surface pressure around the models placed in wind tunnel were made. 3D solutions of the flow fields were obtained with two different turbulence models. The mean velocity and turbulence kinetic energy profiles are influenced by the roof pitch. Recirculation regions occur on the leeward part of roofs and at the behind of the models due to flow separation. These regions are much stronger and spread up to the roof ridge with increasing roof pitch. Largest values of turbulence kinetic energy for entire flow field occur at height of the roof level and they prove the presence of the mixing layer between the free stream flow and reverse flow region. It is seen from the surface mean pressure distributions that the 15° roof pitch causes more critical suction on the roofs than those of the 30° and 45° roof pitches. The numerical results shows that Realizable k-ε turbulence model exhibit better agreement at the prediction mean velocity and turbulence kinetic energy while Standard k-ω turbulence models exhibit better agreement at the prediction of mean pressures coefficients.

1. Introduction

The flow fields over the three-dimensional surface mounted building models are dominated by flow separation. This event has a great deal of importance in large number of applications such as the effects of surface roughness on boundary layer characteristics, wind loads on structures, and dispersion of pollutants emitted from conventional and nuclear power plants. The loading effects of the natural wind on buildings are rather complicated interactive process between the wind flow and the various components of the building. Damage to the buildings results from aerodynamic wind pressure that develop as air flow over and around the building. Depending on the past damage investigation reports [1], most of the wind damage was on the envelope of buildings, in particular at the roof sheathing. For this reason, a detailed understanding about the wind effects on low-rise buildings, and in particular, on roof sheathing is necessary. Wind tunnel experimentation plays an important role in the evaluation of design roof wind loads. Several fundamental experiments carried out with three-dimensional bluff bodies to better understand the flow structure around the buildings. These experiments attempted to define the effect of various parameters of the approach flow like the turbulence intensity and its scale on the surface pressure distributions. An experimental investigation of the flow around surface mounted cubes in both laminar and turbulent flows was conducted by Castro and Robins [2]. They presented the measurements of body surface pressures and mean and turbulence velocities within the recirculation regions which result from the flow separation. Castro [3] investigated the dynamics of the shear layer separating from the front edge of two dimensional, surface mounted, square sectioned blocks. He concluded that as the thickness of the upstream boundary layer increases, the shear layer grows more rapidly and moves downwards. Holmes [4] described the variation of the flow separation associated with the flow turbulence. He indicated that the turbulence characteristics in the flow have strong influence on the roof wind loads. Fackrell [5] studied the roof flow behavior and the resulting nature of the separated region downstream of the body.

* Corresponding author.
E-mail address: yozmen@ktu.edu.tr (Y. Ozmen).

http://dx.doi.org/10.1016/j.buildenv.2015.09.014
0360-1323/

Atli [6] measured the mean and turbulence velocities around surface mounted flat plates and visualized the surface flow using the oil technique. He showed that the height of the plate thickness of the boundary layer and Reynolds number are the parameters, which affect the flow field and structure of turbulence. Agui and Andreopoulos [7] concluded from the pressure fluctuation measurements that the large scale structures play an important role in the dynamics of separated flow. Ginger and Letchford [8] conducted point and area-averaged pressure measurements on a building roof with a scale of 1:100 immersed in a simulated suburban atmospheric surface layer in wind tunnel. They noted that large magnitude mean and fluctuating pressures were measured within regions of flow separation on low rise building roofs. Baydar and Onur [9] investigated the flow field around the obstacles immersed in boundary layer experimentally and numerically. They found that flow separated leading edge of the obstacle attaches to the top surface of the obstacle for $L > 4H$. Becker et al. [10] investigated the structure of the flow field around three-dimensional obstacles for different aspect ratios (wall length to wall width), in two different types of boundary layers in wind tunnel. They found that the flow structure around the obstacle is affected from aspect ratio, the angle of attack, the Reynolds number, and the type of boundary layer. A review of existing and new considerations for the assessment of wind loads on low-rise structures from wind tunnel experiments is presented by Tieleman [11]. Sousa and Pereira [12] investigated the effect of a gable roof (30° roof pitch) in the mean and turbulent flow structure around a surface-mounted cubic obstacle by the use of a 2D-DPIV (Digital Particle Image Velocimetry) system. They noted that the addition of a gable roof to the cubic obstacle has a strong impact both in the mean and turbulent flow fields. An experimental study using PIV (Particle Image Velocimetry) to quantify the characteristics of a tornado-like vortex and to reveal the dynamics of the flow-structure interactions between a low-rise, gable-roof building model and swirling tornado like winds was conducted by Hu et al. [13]. The wind loads acting on the gable-roof building model in tornado-like winds were found to be at least 3 times higher compared with those in a straight-line, atmospheric boundary layer wind at all the compared orientation angles. Mahmood [14] carried out an experimental study on 1:100 scale models of TTU (Texas Tech University) Test Building in a wind tunnel under different flow conditions. He conducted flow visualization, flow and turbulence measurements and pressure measurements on both sharp-edged and round-edged models and found that the type of edge affected the turbulent levels and flow characteristics. Kim and Tamura [15] investigated the effect of incident flows on the wind loads and their combinations effects on a target low-rise building. They found that the pressure coefficients of the isolated model differ significantly depending on the incident flows. Tominaga et al. [16] experimentally and numerically investigated the flow fields around isolated gable roof buildings with different roof pitches. They found good agreement between measured values and simulation results for streamwise velocity profiles in front of buildings. Flowe and Kumar [17] solved the flow fields around the various 3D building shapes by using Standard k-ε turbulence model with FLUENT. They concluded that the length of reverse flow region is a function of building dimensions. Oliveira and Younis [18] computed the flow field around a building with gable roof by using Standard k-ε and Reynolds Stress turbulence models. They found that the use of a Reynolds-stress closure enables the prediction of flow separation on the windward side of the roof while no flow separation is obtained with the Standard k-ε model. Tutar and Oguz [19] investigated the effects of wind flow around a group of buildings with different wind directions and building arrangements by using large eddy simulation. The numeric results show that the large eddy simulation with the finite volume method is more successful to calculate the wind effects on buildings than conventional turbulence models. Lien et al. [20] used four different k-ε turbulence-closure models applied with wall functions to calculate the disturbed mean flow and turbulence through and above an array of two-dimensional buildings. They found that the non-linear k-ε model give the best performance among four different turbulence closure models examined. Flow field around a cube using Standard k-ε turbulence model is numerically predicted by Gao and Chew [21]. They examined the linkage between reverse flow region and turbulence kinetic energy. Virk and Holdo [22] numerically analyzed flow field around a low-rise pitched roof church building with a spirelet using Standard k-ε turbulence model. Results revealed that lifting up of roof tiles is due to the presence of a localized low pressure and swirling turbulent flow near the spirelet section of the church.

When the current literature is examined, it is seen that studies of pressure distributions on the roofs contains almost all existent studies. The studies related to determine the velocity fields are really inconsiderable. However, it is known that the pressure distributions are directly affected from separated flow regions. This study aims to show the relation between the reverse flow regions and the critical pressures on the roofs by determining the velocity and turbulence profiles and pressure distributions on the roofs. This study clarifies the relationship between velocity and pressure distributions as different from previous studies in this field.

2. Experimental study

The experiments carried out in a low speed, open circuit L-2B wind tunnel at von Karman Institute. The wind tunnel has a working section of 350 mm wide, 350 mm high and 2000 mm long. The combination of vortex generators and roughness elements at the entrance to the test section is used to simulate atmospheric boundary layer [23]. A turbulent boundary layer of 150 mm thickness is obtained at the free stream velocity (U_o) of 15 m/s, giving a Reynolds number based on building height of Re = 40000. Fig. 1 indicates a schematic diagram of the wind tunnel test-section. δ and H represent the boundary layer thickness and characteristic height of models, respectively. The ratio of boundary layer thickness to model height (δ/H) is 3.75. A smoke-wire technique is used to visualize the flow structure around models. A 0.2 mm diameter stainless steel wire is vertically located at the entrance of test section. Before each test the smoke-wire was coated by paraffin oil and then heated by Joule effect of DC current. The flow pattern visualized is photographed successively by a video camera. Flow visualization is performed at Re = 5000 since this technique is limited to the small Reynolds numbers. The mean velocity, turbulence and mean surface pressure measurements conducted with the measurement chain systems as shown in Fig. 2. Mean velocity and turbulence measurements are obtained together with TSI IFA-100 hot-wire anemometer interfaced to a data acquisition system using TSI 1211 hot wire probe. The probe is calibrated with TSI Model 1125 calibration apparatus. Anemometer system consisted of a constant temperature anemometer, a hot wire probe with support and cable, connector box, A/D board and computer. A two dimensional traversing holds the probe in the streamwise and cross flow direction. Pitot tube is used to measure free stream velocity at the entrance to the test section. During the experiments, the velocity signals are sampled using DAS20 data acquisition system. Each point was taken 8192 data samples. All signals were filtered at 300 Hz, and sampled at 1000 Hz. A difficulty in the anemometer arises when the flow reverses its direction which is the case in the present study. The mean velocity output of the anemometer is always positive, that is, the output of an anemometer does not indicate the flow direction. The reverse flow in the recirculation