Building and Environment 60 (2013) 137–149

Contents lists available at SciVerse ScienceDirect

Building and Environment

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

CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: Validation and sensitivity analysis

H. Montazeri*, B. Blocken

Building Physics and Services, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

ARTICLE INFO

Article history:
Received 10 July 2012
Received in revised form
11 October 2012
Accepted 20 November 2012

Keywords:
Computational fluid dynamics (CFD)
Building aerodynamics
Wind pressure coefficient
Sensitivity study
Verification and validation

ABSTRACT

Knowledge of the pressure distribution on building walls is important for the evaluation of wind loads and natural ventilation. Wind-induced pressure distributions are influenced by a wide range of factors including approach-flow conditions, urban surroundings and building geometry. Computational Fluid Dynamics (CFD) can be a valuable tool for determining mean wind pressure coefficients on building facades. However, while many CFD studies of mean wind pressure on buildings have been performed in the past, the vast majority of these studies focused on simple building geometries without facade details such as balconies. These details however can drastically influence the flow pattern and the overall pressure distribution on the facade. This paper presents a systematic evaluation of 3D steady Reynolds-Averaged Navier–Stokes (RANS) CFD for predicting mean wind pressure distributions on windward and leeward surfaces of a medium-rise building with and without balconies. The evaluation is based on a grid-sensitivity analysis and on validation with wind-tunnel measurements. It is shown that building balconies can lead to very strong changes in wind pressure distribution, because they introduce multiple areas of flow separation and recirculation across the facade. The results show that steady RANS, in spite of its limitations, can accurately reproduce the mean wind pressure distribution across the windward facade of the building. The average deviations from the wind-tunnel measurements are 12% and 10% for the building with and without balconies, respectively. In addition, also the important impact of the reference static pressure and the turbulence model are demonstrated.

1. Introduction

Knowledge of the pressure distribution on building walls is essential to evaluate wind-induced natural ventilation and to assess wind loads on building walls and building components (e.g. [1–8]). As an example, Building Energy Simulation (BES) programs require pressure coefficient data as input for analysing ventilation and infiltration flow rates [2]. Similarly, design standards need data with a high accuracy for effective-cost designs and reduction of wind damage and cost to building components [9,10].

The pressure distribution on building walls is influenced by a wide range of factors including approach-flow conditions [11–13], urban surroundings [14], building geometry [1] and wind direction [15]. In particular, building facade details such as balconies and other protrusions can affect the peak and mean surface pressure distributions on buildings walls and roofs [16–18].

Pressure coefficients can be determined using full-scale on-site measurements [15,19–26], reduced-scale wind-tunnel measurements [27–32] or numerical simulation with Computational Fluid Dynamics (CFD) [13,16,33–37]. Full-scale measurements offer the advantage that the real situation is studied and the full complexity of the problem is taken into account. However, full-scale measurements are usually only performed in a limited number of points in space. In addition, there is no or only limited control over the boundary conditions [38]. Reduced-scale wind-tunnel measurements allow a strong degree of control over the boundary conditions, however at the expense of – sometimes incompatible – similarity requirements. Furthermore, wind-tunnel measurements are usually also only performed in a limited set of points in space [13]. CFD on the other hand provides whole-flow field data, i.e. data on the relevant parameters in all points of the computational domain [5,39,40]. Unlike wind-tunnel testing, CFD does not suffer from potentially incompatible similarity requirements because simulations can be conducted at full scale. CFD simulations easily allow parametric studies to evaluate alternative design configurations, especially when the different configurations are all a priori embedded within the same computational domain and grid (see e.g. [41]). CFD is

* Corresponding author. Tel.: +31 (0) 40 247 5790; fax: +31 (0) 40 243 8595.
E-mail address: h.montazeri@tue.nl (H. Montazeri).

0360-1323/$ – see front matter
http://dx.doi.org/10.1016/j.buildenv.2012.11.012

increasingly used to study a wide range of atmospheric and environmental processes. Examples are pedestrian wind comfort and wind safety around buildings [40,42–46], natural ventilation of buildings [5,41,47–53], air pollutant dispersion [54–58], convective heat transfer [59–61], etc. In some of these studies, CFD was applied and evaluated in detail, including verification, validation and sensitivity analyses. CFD has also been used on many occasions in the past to determine mean wind-induced pressure distributions on building facades. However, the vast majority of these studies focused on relatively simple building shapes and plane, smooth facades without protrusions or recessions (e.g. [34–36,62,63]). Nevertheless, many historical and contemporary building facades are characterized by protrusions and recessions. To the best of our knowledge, a detailed evaluation of steady Reynolds-averaged Navier–Stokes (RANS) CFD has not yet been performed for mean wind pressure distributions on such building facades.

This paper therefore presents a systematic and detailed evaluation of 3D steady RANS CFD for predicting mean wind pressure distributions on building facades with and without balconies for both normal and obliquely approach-flow conditions. The evaluation is based on a grid-sensitivity analysis and on validation with wind-tunnel measurements by Chand et al. [17]. The impact of several computational parameters is also investigated, including the resolution of the computational grid, the reference static pressure and the turbulence model.

In Section 2, the wind tunnel experiments by Chand et al. [17] are briefly outlined. Section 3 presents the computational settings and parameters for the reference case, and the validation of the CFD results with the wind-tunnel measurements. In Section 4, the sensitivity analysis is performed, including the influence of building balconies on the wind pressure distribution. A discussion on the limitations of the study is given in Section 5. The main conclusions are presented in Section 6.

2. Description of wind tunnel experiments

Atmospheric boundary layer wind-tunnel measurements of wind-induced surface pressure on the facades of a medium-rise building were conducted by Chand et al. [17]. The open-circuit wind tunnel was 14 m long and had a test section of 2.5 × 1.8 m^2. The atmospheric boundary layer was generated by a combination of three devices: vortex generators, a grid of horizontal rods and a set of roughness elements on the floor of the test section. The resulting vertical profile of mean wind speed at the location of the building (but without building model present) is represented by a log law with aerodynamic roughness length $z_0 = 0.008$ m (model scale, corresponding to 0.24 m in full scale) and a friction velocity $u^*_{ABL} = 0.73$ m/s. The measured incident longitudinal turbulence intensity ranges from 13% near ground level to about 3% at gradient height. Because these profiles were measured at the (virtual) location of the building, they represent the incident, rather than the approach-flow conditions. Using the incident-flow conditions in CFD is important for simulation accuracy [64]. The upstream wind velocity, measured at building height, was equal to 7.1 m/s, yielding a building Reynolds number of 250,000 which is well above the critical value of 11,000 for Reynolds number independent flow [65].

The building at scale 1:30 had dimensions width × depth × height = 0.60 × 0.25 × 0.50 m^3 (reduced scale, see Fig. 1) corresponding to full-scale dimensions 18 × 7.5 × 15 m^3, resulting in a blockage ratio of about 6.6%. To evaluate the effect of building balconies on the mean pressure coefficient, measurements were carried out for a building with and without balconies. Three balconies with width 0.15 m, depth 0.05 m and height 0.03 m were positioned at every one of the five floors, except the ground floor (Fig. 1).

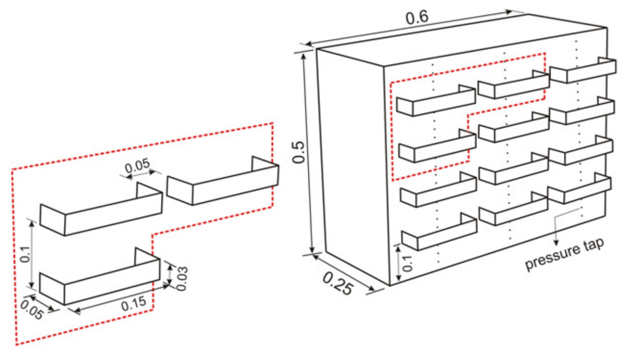

Fig. 1. Geometry of building model and balconies (dimensions in metres at model scale).

Mean surface pressures were measured along three vertical lines on the windward and leeward facade. Each measurement line was positioned in the middle of the balconies and 45 holes were drilled at equidistant points along it (Fig. 1). In the remainder of this paper, we will refer to these vertical lines as "edge lines" and "centre line". The measurements were performed with a scanning valve and a digital micro-manometer. Upstream static and dynamic pressures were measured with a Pitot tube mounted 0.90 m upstream of the model and at building height. During the surface pressure measurements, the static tube was connected to the negative port of the scanning valve. So, the manometer indicated the pressure differences of surface pressure and free stream static pressure. The results of the wind-tunnel measurements will be shown together with the validation in the next sections.

3. CFD simulations: reference case

A reference case is defined as a starting point for the sensitivity analysis. It includes a fixed choice for the computational geometry and grid, boundary conditions and turbulence model, as outlined below.

3.1. Computational geometry and grid

A computational model was made of the reduced-scale building model used in the wind-tunnel measurements. The dimensions of the computational domain were chosen based on the best practice guidelines by Franke et al. [66] and Tominaga et al. [67]. The upstream domain length is $5H = 2.5$ m. The resulting dimensions of the domain were $W \times D \times H = 10.6 \times 10.25 \times 3$ m^3, which corresponds to 318 × 307.5 × 90 m^3 in full scale. The computational grid was created using the surface-grid extrusion technique presented by van Hooff and Blocken [41]. The procedure was executed with the aid of the pre-processor Gambit 2.4.6, resulting in a hybrid grid with 2,102,250 prismatic and hexahedral cells. The grid is shown in Fig. 2a–c. 20 and 10 cells are used along the width and depth of the balconies, respectively, as shown in Fig. 2c. A maximum stretching ratio of 1.2 controls the cells located in the immediate surroundings of the building model. The grid resolution resulted from a grid-sensitivity analysis that will be outlined in Section 4.1. The minimum and maximum cell volumes in the domain are approximately 5×10^{-8} m^3 and 7.5×10^{-2} m^3, respectively. The distance from the centre point of the wall adjacent cell to the wall, for the windward, leeward and ground plane is 0.0017 m, 0.0022 m and 0.0025 m, respectively. This corresponds to y^* values between 20 and 350. As standard wall functions are used in this study, these values ensure that the centre point of the wall-adjacent cell is placed in the logarithmic layer. The domain shape (Fig. 2a) allows