Building and Environment 141 (2018) 166–181

Contents lists available at ScienceDirect

Building and Environment

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

Experimental investigation of the atmospheric boundary layer flow past a building model with openings

Marinos Manolesos[a,b,*], Zhiqiu Gao[a], Demetri Bouris[c]

[a] State Key Laboratory of Atmospheric Boundary Layer Physics and Atmospheric Chemistry, Institute of Atmospheric Physics, Chinese Academy of Sciences, Beijing, 100029, China

[b] School of Engineering, Swansea University, Bay Campus, Fabian Way, Swansea, SA1 8EN, UK

[c] Laboratory for Innovative Environmental Technologies, School of Mechanical Engineering, National Technical University of Athens, 9 Iroon Polytechneiou str., 15780, Athens, Greece

A B S T R A C T

As modern building design moves towards more sustainable solutions the use of natural ventilation is one of the options considered to improve indoor air quality and to minimize the energy cost of the buildings. The present cross-ventilation study is an experimental investigation of the atmospheric boundary layer flow past a cubic building model with vertical openings. Wind tunnel experiments were performed for two different simulated upstream boundary layer conditions and for two different cube options (with and without openings). Pressure measurements on the building model surface are in very good agreement with benchmark measurements. Stereo Particle Image Velocimetry measurements were performed to examine the effect of both the upstream condition and the openings. It is found that both conditions significantly alter the pressure and flow structure around the building model. Ventilation rate is estimated using two methods, the orifice equation and the measured velocity profile in the vicinity of the apertures. The comparison shows that the orifice equation overpredicts the ventilation rate and the effect of the upstream boundary layer. All data in the present report are freely available for validation purposes.

1. Introduction

Knowledge of the details of the flow field in and around buildings and of the pressure distribution on their external surfaces provides crucial information for numerous applications. These include those related to occupant comfort, the operation, structural integrity and energy performance of the buildings themselves (natural ventilation, wind loads, infiltration, heat losses etc.) and those related to energy systems that depend on the building geometry (urban wind turbines, solar collectors etc.). When it comes to sustainable or zero energy buildings and healthy environments, natural ventilation is a crucial factor [1,2]. It is an integral part of modern building design and it is usually combined with active ventilation features in hybrid systems in order to achieve higher energy efficiency [3].

Wind induced ventilation and more specifically cross-ventilation is both pressure and momentum driven [4,5] so, for an effective design, the performance of different building components and the interaction among and with the wind environment should be known accurately. Numerous studies have dealt with the issue of ventilation performance assessment and recent reviews have been provided by different authors [6–9].

Wind tunnel tests can be used for natural ventilation design [10], or for CFD validation purposes [11–14], which is a requirement from the numerical point of view [15]. In the latter exercise, also known as *validation application* [16,17], experimental results are considered benchmark cases for Computational Fluid Dynamic (CFD) simulations [5,18–22] or analytical descriptions [23–26]. Although the comparison of experimental results with mathematical model predictions seems like a straightforward task, it should be performed with great care and vigilance from both sides (numerical and experimental) in order to avoid systematic errors. As highlighted in the past, caution should be exercised to avoid comparing *apples with oranges* [27]. On the other hand, a fruitful application of the cross-comparison between wind tunnel measurements and CFD simulations can lead the development of best practice guidelines for CFD applications [28–30].

Among the wind engineering experimental investigations, the effect of the upstream boundary layer (BL) characteristics, such as ground shear velocity and Turbulence Intensity (TI), on the structure of the flow past buildings has been a subject of previous research [31–33]. However, to the authors' knowledge, there is a scarcity of experimental

* Corresponding author. School of Engineering, Swansea University, Bay Campus, Fabian Way, Swansea, SA1 8EN, UK.
E-mail address: marinos.manolesos@swansea.ac.uk (M. Manolesos).

https://doi.org/10.1016/j.buildenv.2018.05.049
Received 21 February 2018; Received in revised form 30 April 2018; Accepted 21 May 2018
Available online 23 May 2018

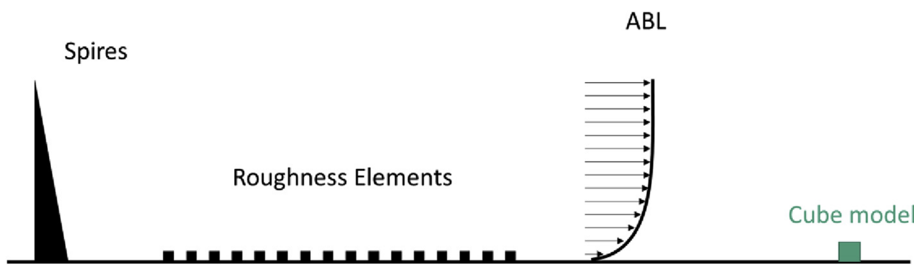

Fig. 1. Schematic side view of the experimental set up. The Spires and roughness elements used to create the atmospheric boundary layer (ABL) are shown along with the cube model.

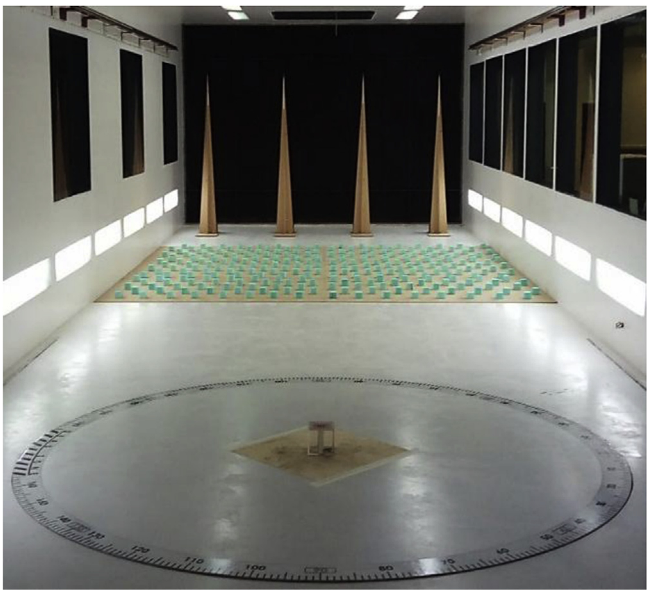

Fig. 2. View of the test section, looking upstream. The spires, and roughness elements are visible with the Plexiglas model located at the centre of the turntable.

studies that include scaled atmospheric BL simulations in the study of the flow past buildings with a porous envelope, with reference to natural ventilation applications.

One of the limiting factors of relevant flow field experimental campaigns up to now was that they provided point measurements [34], with very limited use of non-intrusive field methods, such as Particle Image Velocimetry (PIV) [11] and practically no use of Stereo PIV [35]. In addition, all ventilation studies concerned horizontal, square or round openings, that did not necessarily scale with the upstream BL and/or the building itself and therefore their vertical positioning had little relevance to the natural ventilation application [11,14,36].

The additional challenge in the latter case is that the pressure distribution along the opening is not constant and hence the orifice equation cannot be used for airflow rate prediction. The orifice equation is widely used in cross-ventilation studies, despite the fact that it has been known to give questionable predictions even for simple aperture geometries [12,37]. In general, natural ventilation flow rate prediction is considered a particularly challenging task and even more sophisticated methods can have a significant error margin. In fact, significant discrepancies between the flow rate measured at the windward and the leeward opening have been reported [38–40].

The present study is a wind tunnel investigation of wind driven cross-ventilation. The aim is to provide further insight into the interaction of basic cross-ventilation flow characteristics in and around the building and, additionally, to generate an experimental data base that may be used to validate CFD codes. The building model is a cube with vertically distributed openings and a clearly defined interior geometry. The cube is embedded in a controlled simulated atmospheric BL. Pressure and Stereo PIV measurements are presented in order to examine the effect of upstream conditions on cross-ventilation and the effect of the latter on the flow around the building.

It is noted that a generic cubic building shape has been chosen and that the vertical openings are not intended to simulate a scaled version of a single opening on the building face. At the achieved scale of 1:400 (see section 2.3), the dependence of the opening's discharge coefficient on the opening Reynolds number would not permit scale similarity [1]. The vertical openings were chosen in order to study their effect on and interaction with the simulated upstream atmospheric boundary layer and the building surface pressure distribution. While there have been studies that investigated the effect of the relative position of single openings on cross ventilation [9,11,20], experimental investigations of

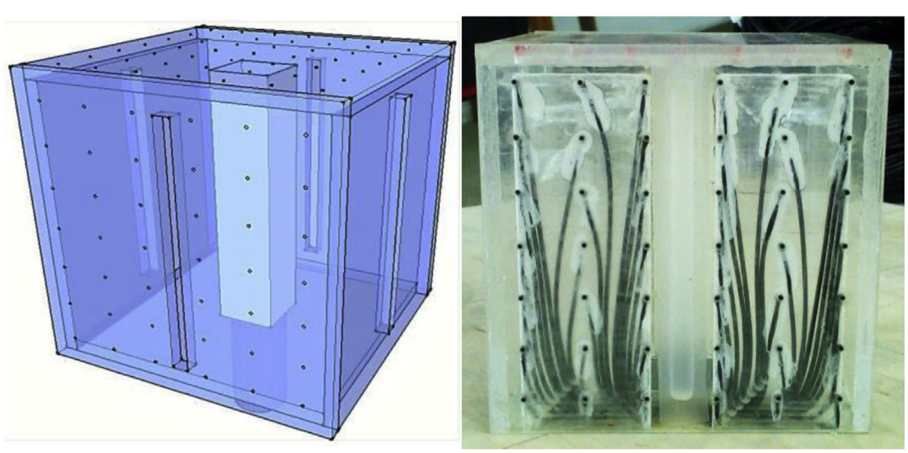

Fig. 3. (Left) 3D view of the plexiglass cube with visible slot openings on all sides, pressure tap positions on the roof and a single sidewall and the inner column concealing the pressure tubing and (right) front view of the actual cube model.