J. Wind Eng. Ind. Aerodyn. 133 (2014) 18–26

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

Study on the surface pressure distribution of cubes in cross-wind arrays

Hee-Chang Lim [a,*], Kenji Tsukamoto [b], Masaaki Ohba [b], Kunio Mizutani [b]

[a] School of Mechanical Engineering, Pusan National University, San 30, Jangjeon-Dong, Geumjeong-Gu, Busan 609-735, South Korea
[b] Department of Architecture, Faculty of Engineering, Tokyo Institute of Polytechnics, Atsugi, Kanagawa, 243-02/3, Japan

ARTICLE INFO

Article history:
Received 22 December 2013
Received in revised form
11 July 2014
Accepted 17 July 2014
Available online 8 August 2014

Keywords:
Interference effect
Side-by-side cubes
Surface pressure
Effects of the gap
Wind tunnel measurement

ABSTRACT

In this study, effects of the gap on equal-spaced cubical bodies (150 mm × 150 mm × 150 mm) placed in a turbulent boundary layer were investigated inside an atmospheric boundary layer wind tunnel. This study includes various measurements of surface-pressure distribution around a building in close proximity to surrounding obstacles. In addition, we examined the surface-pressure variation with cube distances (G) at 75 mm (0.5 h), 150 mm (1 h), 225 mm (1.5 h) and ∞ (i.e., a single cube). The experiments conducted included some boundary layer wind tunnel tests with Hot Wire Anemometry (HWA) and mean and fluctuating surface-pressure measurements around a set of cubes aligned in parallel. The tunnel tests were carried out at two different Reynolds numbers ($\mathrm{Re}=4.6\times10^4$ and 6.7×10^4), based on wind velocity U_h (4.5 m/s and 7.3 m/s) at a cube height h. On analyzing the results, we discovered that the gap effect of surrounding models has a significant influence on the pressure variation around the central model. The overall surface-pressure coefficient around the central structure was generally found to increase as the gap (G) between the structures was increased.

1. Introduction

Over the past decades, the wind load characteristics around bluff bodies have been of fundamental interest in the study of fluid dynamics. In addition, these characteristics have long been considered as a critical design parameter from various engineering points of view civil, fluid mechanical, and architectural. Numerous empirical and simulated data and comparisons of the flow around buildings of all kinds have therefore been carried out. Most of the past studies fall into one of three categories: tandem arrangements, in which one building is directly in the wake of the other; side-by-side arrangements, in which the buildings are arranged transverse to the incoming flow; and staggered arrangements, in which the buildings are arbitrarily configured. The flow field, pressure coefficients, force variation, and intensification or suppression of vortex shedding are highly dependent on the configuration of the building, and on the shape and spacing of the building group due to both wake and proximity-induced interference effects.

A frequently cited paper in this area, especially cylindrical models, Zdravkovich (1977, 1987), described wind-tunnel experiments that measured the flow around two surface-mounted cylinders. In that paper, Zdravkovich analyzed the problem of flow interference that arises when two cylinders are placed side-by-side, in tandem, and in staggered arrangements in steady state. It was observed that the vortex-induced force and the vortex shedding pattern were completely different from those found on a single cube in the case of the same Reynolds number. It was one of the first demonstrations of the crucial importance of modeling appropriately the design details of a building configuration. However, that early work only focused on the importance of ensuring appropriate simulation of the building arrangements.

A number of findings on interference have been obtained even though most of them involve only two bodies in close proximity. The most common interference mechanisms include shelter effect, flow channelling, flow asymmetry, and wake buffeting. When two buildings are in a tandem arrangement, the upstream building generally provides shielding for the downstream building. This normally leads to a reduction in the mean along-wind force on the downstream building. However, fluctuating wind force may become larger due to turbulence buffeting (Bailey and Kwok, 1985). The presence of a neighboring building introduces asymmetry in wind flow pattern around the target building, leading to the possibility of highly magnified wind-induced torsion (Zhang and Kwok, 1994). An upstream building is generally not significantly affected by a downstream building but when two buildings are in very close proximity, wind flow is channelled to sweep through the building gap.

Above all, the study of two bluff bodies placed close together is generally considered to be important due to their interaction effects on each other. Ricciardelli and Vickery (1998) investigated the aerodynamics forces acting on a pair of square cylinders in

---

* Corresponding author. Tel.: +82 515 102 302; fax: +82 515 125 236.
E-mail address: hclim@pusan.ac.kr (H.-C. Lim).

http://dx.doi.org/10.1016/j.jweia.2014.07.009
0167-6105/

a tandem and in a side-by-side arrangement. It was measured that the pressure in smooth and turbulent flows with angles ranging from 0° to 90° and wide-range separations from 2 H to 13 H. It was observed that for large angles of attack, the highest values of the root mean square (rms) force coefficients were found for large separations. In addition, To and Lam (2003) reported some interesting interference effects not previously observed in past investigations on two buildings in the case of a larger separation. However, with regard to the effect of the gap on groups of buildings, interference effects on a group comprising three or more buildings have not been studied in detail so far, and there are many problems still left to be investigated.

The immediate emphasis in this paper is on a group of cubical bodies with various gaps. The gap between bodies is responsible for the type of wake generated and, ultimately, for the structural loading, pressure, and (especially) structural excitation. For example, with the inclusion of another building in a side-by-side arrangement, the loading pattern becomes quite complex. The buildings may experience increased or reduced wind loads depending on their geometries and spacing, as well as the characteristics of wind flow and upstream terrain. This paper consists of a carefully designed set of experiments on boundary layer flow over surface-mounted side-by-side cubes. These experiments were conducted under various speeds in an atmospheric boundary layer wind tunnel to allow variations in Re, while the upstream boundary layer characteristics were kept relatively similar. Data is also presented with the previous wind-tunnel measurements, yielding a further change in Re and validating the present wind tunnel experiments.

The flow inside a wind tunnel was designed to be similar to the (rural) atmospheric boundary layer. Here, the main emphasis is primarily on the extent to which the flows are affected by Re, and attention is concentrated on the effect of the gap defined by distances between the bodies corresponding to 0.5 h, 1.0 h, 1.5 h and ∞ (i.e., a single cube); where h is the height of the cube. This paper is organized as follows. Section 2 outlines wind tunnel details and techniques. Section 3 describes the generation of a simulated turbulent boundary layer and the surface-pressure variation around a single cube. Section 4 describes the surface-pressure variation resulting from gaps in the models, and also presents and discusses the major results. Finally, Section 5 presents the major conclusions.

2. Design of the wind tunnel experiment

2.1. Configuration of the wind tunnel

The wind tunnel tests were conducted in the turbulent boundary layer wind tunnel (Fig. 1) of the Wind Engineering Research Center at Tokyo Polytechnic University (TPU) in Japan. This wind tunnel is an open-circuit, low-speed wind tunnel designed for wind environmental assessment and ventilation studies. Most of the experiments were conducted in the end-part test section of the tunnel, where the sectional dimensions were 1.2 m width, 1.0 m height, and 14 m length, with a maximum wind speed of approximately 30 m/s. However, for reasons of structural stability and safety, the wind tunnel usually operates at a speed lower than rated. Table 1 gives the dimensions of the group of surface roughness blocks used in the wind tunnel to generate the simulated turbulent boundary layer. The details of the generated turbulent boundary layer are illustrated in Fig. 1. The combination of turbulence-generating spires at the entrance and arrays of various rectangular blocks on the tunnel floor is a general way of generating a deep turbulent boundary layer [see Cook, 1973, 1977 in detail].

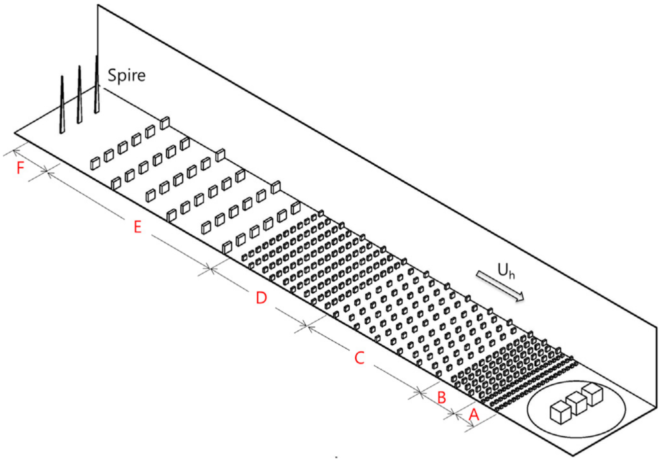

Fig. 1. Schematic diagram of the wind tunnel.

Table 1
Group of surface roughness blocks used in the tunnel.

Configuration	A	B	C	D	E	F
Size [W × H] (mm^2)	30 × 30	50 × 50	50 × 50	50 × 50	98 × 98	70 × 700
No. elements	60	60	128	108	36	3
Length (mm)	240	525	1780	1435	2450	450

2.2. Description of the measurement system

Flow measurements were conducted using hot wire anemometry (in our case, a multi-channel Constant Temperature Anemometry (CTA) module and a split-fiber probe of DANTEC (55H31)), which is highly sensitive to turbulent wind flow. For calibrating this precise measurement system, a well-known, less-sensitive, more-robust calibrator was used with a Pitot tube and pressure-difference manometer (MKS270). Surface pressure variation around the models is also an important indicator of the surrounding characteristics. It was obtained using a multi-channel pressure-measurement system in which each channel is connected to an analogue/digital (A/D) converter in such a way that all 32 channels of surface pressure were directly and simultaneously acquired and transferred to a personal computer for storing the data. The reference static and total pressures were monitored using a Pitot tube located 4 m upstream at a height of 0.5 m above the roof of the model. A multi-channel pressure-measurement system was utilized during the measurement and 65,536 sequential samples of pressure at all the pressure taps were obtained simultaneously at a sampling frequency of 1024 Hz. In order to reduce the effect of the diameter and length of the Teflon® pressure tube (approximately 1 mm and 0.5 m, respectively) (the diameter and length are approximately 1 mm and 0.5 m, respectively), the mean and fluctuating pressures were calibrated before the main measurement. In this case, the calibration was based on the correction of distortion effects caused by tubing systems. In particular, the frequency response of the tubing effect was numerically compensated for frequencies ranging up to 250 Hz using the gain and phase-shift characteristics of the pressure measuring system (Irwin et al., 1979). In addition, the pressure records were digitally low-pass filtered at 300 Hz.

2.3. Cube models and surface-pressure coefficient

As shown in Fig. 2, the models (150 mm in height, 150 mm in width, and 150 mm in length), were all made from 10-mm-thick acrylic. One of them was used to measure the surface pressure and