Building and Environment 92 (2015) 452–461

Contents lists available at ScienceDirect

Building and Environment

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

Wind tunnel experiments on cross-ventilation flow of a generic building with contaminant dispersion in unsheltered and sheltered conditions

Yoshihide Tominaga [a, *], Bert Blocken [b, c]

[a] Department of Architecture and Building Engineering, Niigata Institute of Technology, 1719 Fujihashi, Kashiwazaki, Japan
[b] Department of the Built Environment, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands
[c] Department of Civil Engineering, Leuven University, Kasteelpark Arenberg 40 — Bus 2447, 3001 Leuven, Belgium

ARTICLE INFO

Article history:
Received 2 April 2015
Received in revised form
21 May 2015
Accepted 22 May 2015
Available online 23 May 2015

Keywords:
Wind tunnel experiment
Cross-ventilation
Contaminant dispersion
Validation
Experimental data set

ABSTRACT

Recently, computational fluid dynamics (CFD) has been widely used for the prediction and analysis of cross-ventilation flows in buildings. In this study, detailed wind tunnel experiments were performed on the cross-ventilation flow of a generic single-zone building in order to compile a validation database for CFD methods. Both the velocity fields and the contaminant concentration fields were measured and investigated. First, the fundamental characteristics of the velocity and concentration fields in a cross-ventilated flow were investigated for the building in unsheltered conditions. Next, the distributions of turbulent scalar fluxes in a cross-ventilated flow, which have been rarely reported, were also measured, and the scalar transport mechanism was examined based on the results. Finally, the effect of the surrounding buildings on the cross-ventilation flow was investigated. This study shows that the turbulent velocity fluctuations and concentration fluctuations are clearly generated by different mechanisms. These results can be used to effectively and successfully validate CFD methods applied to the flow and concentration fields of cross-ventilation flows.

1. Introduction

The cross-ventilation of buildings with large openings is characterized by intricate interactions between the outdoor wind flow around the building and the indoor air flow inside the building. Although numerous studies have been conducted using wind tunnel experiments and on-site measurements in order to grasp the complicated mechanism of cross-ventilation [1–10], this approach is limited in its ability to clarify such a complicated phenomenon because of its three-dimensionality, unsteadiness, multiplicity, and so on. Computational fluid dynamics (CFD) is an effective approach for overcoming such limitations and has already been used in many studies [11–17].

An additional clear merit of the CFD approach is that it can easily consider contaminant dispersion, which is sometimes difficult to capture with physical modeling because of limitations with regard to the measurement resolution and similarity constraints. Since one of the main purposes of cross-ventilation is removing pollutants and other environmental hazards that affect the indoor air quality, CFD simulation of cross-ventilation flows including contaminant dispersion is an important topic. Although several studies have analyzed the dispersion of CO_2 or other tracer gases in cross-ventilated buildings using CFD [18–22], only few have compared their results with experimental data [21,22]. Therefore, the performance of CFD analysis with regard to contaminant dispersion in a cross-ventilated building has not been clarified so far.

The accuracy and reliability of CFD simulations should be confirmed through validation and sensitivity studies, including solution verification [23–29]. Naturally, CFD simulations should be validated with high-quality experimental data. However, there have been very few studies on experimental measurements of the contaminant distribution in a cross-ventilated building [30,31]. Furthermore, previous experimental studies on cross-ventilation flow have mainly focused on the mean (time-averaged) velocities and rarely on the turbulent fluctuation inside the building, even though in cases of large openings, cross-ventilation is characterized by the preservation of the total kinetic energy (mean kinetic energy + turbulent kinetic energy) through openings [32]. Since the transport of pollutant concentration

---

[*] Corresponding author.
E-mail address: tominaga@abe.niit.ac.jp (Y. Tominaga).

http://dx.doi.org/10.1016/j.buildenv.2015.05.026
0360-1323/© 2015 Elsevier Ltd. All rights reserved.

is known to be strongly influenced by the mean and turbulent flow fields, detailed experimental results on contaminant dispersion in a cross-ventilated flow in association with a velocity field should be provided to validate CFD methods.

In this study, detailed wind tunnel experiments were performed on the velocity and concentration fields in the cross-ventilation flow of a generic single-zone building in order to compile a validation database for CFD methods. Both the velocity fields and the contaminant concentration fields were measured and investigated. Section 2 outlines the wind tunnel experiments. Section 3 presents the fundamental characteristics of the velocity and concentration fields in the cross-ventilated flow of the isolated (unsheltered) building. The distributions of the turbulent scalar fluxes, which have been rarely reported, are also presented, and the scalar transport mechanism is considered based on the results. Section 4 clarifies the effects of surrounding buildings on the cross-ventilation flow. Finally, Section 5 summarizes the findings of the present study and discusses future prospects.

2. Outline of wind tunnel experiment

2.1. Building configuration

A generic single-zone building with two opposite openings was considered. This configuration was made similar in shape to the model used in the extensive experiments by Karava et al. [7] so that the results could be compared. The 1:100 scale building model had dimensions $W \times D \times H = 0.20\ \mathrm{m} \times 0.20\ \mathrm{m} \times 0.16\ \mathrm{m}$ Fig. 1(a) indicates the size and location of the openings in the facades perpendicular to the wind direction. The opening positions were central in the windward and leeward facade, at height $h = 80$ mm. Fig. 1(b) presents a perspective view. This configuration corresponds to 'Configuration E1' of the experiment by Karava et al. [7]. The area of one opening was $3.3 \times 10^{-3}\ \mathrm{m}^2$ at reduced scale. A gas outlet, which has a dimension $8\ \mathrm{mm}\ (0.05H) \times 8\ \mathrm{mm}\ (0.05H)$, was installed at the center of the floor.

Measurements were also taken under the condition that the cross-ventilated building was surrounded by buildings of the same size without openings, as indicated in Fig. 2. To investigate purely the influence of the adjacent building, only one block was arranged as a neighboring building in all directions, because surrounding buildings spread to the upwind direction would change the property of the approaching flow significantly. To minimize parameters to be considered, the surrounded buildings have no openings. The street widths were all equal to the building width W.

2.2. Experimental settings

The experiments were carried out in the atmospheric boundary layer wind tunnel at Niigata Institute of Technology [27,33–35]. The test section 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-flow wind profile representative of the lower part of a neutral atmospheric boundary layer. Fig. 3 shows the experimental set-up for the concentration measurements. Fig. 4 shows the vertical profiles of the mean velocity $\overline{u}$ and turbulent kinetic energy k for the incident flow, i.e. measured at the center of the empty turntable. The mean streamwise velocity of this approaching flow obeyed a power law with an exponent of 0.25:

$$\frac{U(z)}{U_H} = \left(\frac{z}{H}\right)^{0.25} \tag{1}$$

where $U(z)$ and U_H are the mean streamwise velocity at height z and that at the reference height H, respectively.

The turbulent kinetic energy k was exactly obtained by three-component measurement of the variances in the velocity fluctuations. This distribution can be approximated by the following relation:

$$\frac{k(z)}{U_H^2} = 0.033 exp^{-0.32\left(\frac{z}{H}\right)} \tag{2}$$

The wind speed at the building height H (i.e., U_H) was measured to be 4.3 m/s, yielding a building Reynolds number of about 45,000. Due to the constraints on the concentration measurement, a lower velocity of 2.2 m/s was required to measure the concentration in the preferred measurement range. This value corresponds to a building Reynolds number of about 23,000. These building Reynolds numbers are higher enough than the previously reported critical ones to obtain Reynolds-number independence [36]. The differences between the vertical profiles of $\overline{u}$ and k with the two experimental velocities were within 5%. The aerodynamic roughness length z_0, deduced from the line fitted to the mean velocity profile except for the effect of the internal boundary layer on the turntable, was 7×10^{-3} m. Appling this z_0 value to the logarithmic law, the friction velocity u^* for the experimental conditions can be calculated as approximately 0.3 and 0.5 m/s, respectively. Therefore, the roughness Reynolds number [37] based on z_0 and u^* were approximately 13 and 25, respectively. It is confirmed that these experimental conditions satisfy the criteria for a fully rough surface [9,36,37].

Ethylene (C_2H_4), which has a density very similar to air, was used as the tracer gas. The emitted mass flow rate of the tracer gas was set to 2.0 L/min for the concentration measurement. This flow rate corresponded to the emission velocity W_e of 0.52 m/s. Therefore, the emission velocity ratio W_e/U_H was 0.25 for the series of

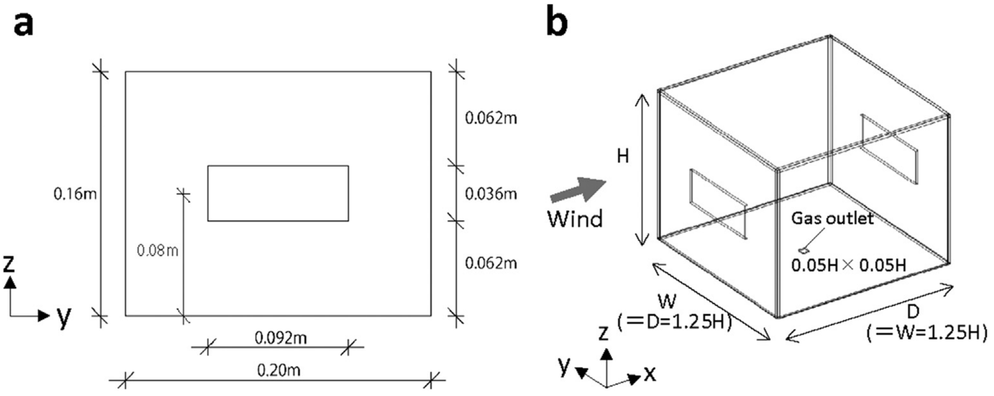

Fig. 1. (a) Front view and (b) perspective view of building model.