Energy and Buildings 43 (2011) 1149–1158

Contents lists available at ScienceDirect

Energy and Buildings

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

CFD simulations of natural ventilation behaviour in high-rise buildings in regular and staggered arrangements at various spacings

James O.P. Cheung, Chun-Ho Liu*

Department of Mechanical Engineering, The University of Hong Kong, Pokfulam Road, Hong Kong

ARTICLE INFO

Keywords:
Building interference
Building disposition
Computational fluid dynamics (CFD)
Cross ventilation
High-rise buildings
Natural ventilation

ABSTRACT

Natural ventilation, which is in line with the concepts of sustainability and green energy, is widely acknowledged nowadays. Prevailing winds in urban areas are unavoidably modified by the increasing number of closely placed high-rise buildings that significantly modify the natural ventilation behaviour. This paper explores the effects of building interference on natural ventilation using computational fluid dynamics (CFD) techniques. The cross-ventilation rate (temporal-average volumetric airflow rate) of hypothetical apartments in a building cluster under isothermal conditions was examined using the standard two-equation $k-\varepsilon$ turbulence model. The sensitivity of ventilation rate to wind direction, building separation and building disposition (building shift) was studied. Placing buildings farther away from one another substantially promoted the ventilation rate, cancelling the unfavourable interference eventually when the building separation was about five times the building width (the optimum separation). The characteristic flow pattern leading to this behaviour was revealed. With the adoption of building disposition, the optimum separation could be reduced to three times the building width. In addition, the airflow rates could be doubled with suitable shifts. Building disposition is therefore one of the feasible solutions to improve the natural ventilation performance in our crowded environment.

1. Introduction

Supply of fresh air, together with the removal of aged air through an indoor space, by natural means is defined as "natural ventilation" [1]. Utilizing natural forces, such as wind and temperature difference, natural ventilation was the principal method driving fresh air into an indoor environment [2]. Yet in megacities, its effectiveness has been reduced by the blockages from adjacent buildings. The local wind field is tremendously modified within a building cluster, e.g. sheltering that causes recirculations behind a building or suppressed ventilation in urban canopies [3]. Natural ventilation should actually be promoted and better utilized due to indoor health and environmental concerns [4,5]. The capital cost of a building integrated with natural ventilation is about 10–15% lower than that of its air-conditioned equivalent [6]. In this paper, we attempt to gain some insights on the ways to implement this natural resource in dense urban regions. Computational fluid dynamics (CFD) was used to investigate how the clearance among buildings would modify the micro-climate and the natural ventilation within a building cluster. In view of the precious land resources in urban areas, we, using an analytical approach, determined the optimum separation between buildings for more favourable natural ventilation performance. Different staggered building arrangements were also considered.

Research on natural ventilation can be dated back to the 1940s [7]. Many studies on natural ventilation across a range of building types, e.g. low- to mid-rise residential apartments, hospitals and livestock farm, etc. [8–10], have been performed over the past several decades. The natural ventilation performance of low-rise buildings was evaluated based on the pressure distributions on the façades [11]. Various pressure coefficients were formulated depending on prevailing wind direction, pressure distribution and building geometry [12]. The sizes and locations of openings were included in discharge coefficient and ventilation rate prediction as well [13,14]. The wind pattern and pressure distribution around and inside a low-rise cross-ventilated model were thoroughly investigated under different wind directions [15]. Apart from wind tunnel measurements, various CFD results were contrasted [16]. Ventilation effectiveness was recognized to be a function of the sizes and positions of openings for both cross and single-sided ventilations [17]. Ayata and Yıldız [18] found that, using suitable building orientation with respect to the prevailing wind, the natural ventilation performance could be markedly improved. The benefits of cross ventilation over single-sided or hybrid ventilations were discussed through the comparison of the air change rate and air change efficiencies elsewhere [19]. CFD is commonly found

* Corresponding author. Tel.: +852 2859 7901; fax: +852 2858 5415.
E-mail address: liuchunho@graduate.hku.hk (C.-H. Liu).

0378-7788/$ – see front matter
doi:10.1016/j.enbuild.2010.11.024

in ventilation studies [20,21]. The number of papers concerning CFD-predicted ventilation performance in buildings found in major journals has dramatically increased since 2006 [22].

The effect of building interference was also recognized. For instance, wind tunnel measurements were performed to study the variations of airflow in the passage in-between two buildings [23–25]. Lee et al. [26] conducted a series of wind tunnel experiments to examine the wind load on a low-rise structure within an array of buildings of various spacings. A severe drop in wind pressure was observed due to the upstream building blockage. Sawachi et al. [27] attempted to parameterize the pressure coefficients for a pair of buildings in proximity. Owing to the substantial modification in the flow structures, the conventional pressure coefficients determined based on an isolated building should be applied cautiously on a building cluster [28]. The change in micro-climate (especially the wind direction) by a group of buildings analogously affects the natural ventilation performance. Chang and Meroney [29] discussed the variations in surface pressures on low-rise buildings with five different separations in details. Furthermore, the turbulence distributions in and around two cross-ventilated buildings separated apart at different distances were studied [30].

Using pressure coefficients or discharge coefficients is the industrial practice to estimate the natural ventilation rate. However, the accuracy is often in doubt because of the persistent kinetic energy of airflow through the openings [21,31,32]. Aynsley [33] found that neglecting the velocity pressure component is a source of error. Under this circumstance, an explicit calculation of the flow around and inside the buildings is necessary. Kobayashi et al. [34] discussed the limitations of using chamber method in the prediction of discharge coefficient, which often led to an average under-estimation of 35%.

Specific research on high-rise buildings has been biased towards the structural perspective such as wind load and moment [35–37]. For instance, a substantial increase in the spanwise component of the wind-induced loading was observed when two adjacent buildings were aligned in the streamwise direction [38]. Lately, a few studies switched their focuses to ventilation. Chow [39] reported that under the influence of a nearby structure, the openings and incident wind could be utilized for improving the natural ventilation in high-rise apartments.

Although natural ventilation has been attracting researchers' attentions for long, the effects of building interference on natural ventilation in high-rise buildings have been merely explored. Major guidelines [40–42] do not have sufficient information covering this topic. Building interference indeed plays a crucial role in modern environmentally friendly building design. In view of the aforementioned benefits, this pilot study was conceived to reveal how neighbouring tall structures modify the wind and natural ventilation performance in a building cluster using CFD techniques. In particular, the sensitivity to building arrangement and prevailing wind direction was examined.

2. Mathematical model

Taking into account the computational cost and the requirement of modelling accuracy, the pseudo steady-state incompressible Reynolds-averaged Navier–Stokes (RANS) approach of the standard two-equation $k-\varepsilon$ turbulence model [43] was chosen in this study. All the simulations were carried out in an isothermal condition, i.e. only wind-driven ventilation was considered. The governing equations are listed below.

Continuity

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

Momentum conservation

$$\frac{\partial \overline{u_i u_j}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_i}{\partial x_i}\right)\right) \tag{2}$$

Turbulent kinetic energy (TKE, k)

$$\frac{\partial k\bar{u}_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + \nu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\frac{\partial \bar{u}_j}{\partial x_i} - \varepsilon \tag{3}$$

Energy dissipation rate (ε)

$$\frac{\partial \varepsilon\bar{u}_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}\nu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\frac{\partial \bar{u}_j}{\partial x_i} - C_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{4}$$

The turbulent viscosity (ν_t) is calculated in terms of k and ε by

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

In the equations, $\bar{u}_i$ and $\bar{p}$ represent the ensemble-averaged velocity components and pressure, respectively. The five modelling constants $C_{1\varepsilon}, C_{2\varepsilon},, C_\mu, \sigma_k, \sigma_{k\varepsilon}$ are 1.44, 1.92, 0.09, 1.0 and 1.3, respectively. The simulations were performed by the commercial CFD code FLUENT 6.3.26 [44].

3. Model validation

We are aware of the growing concern, partly due to the popularization of large-eddy simulation (LES) and direct numerical simulation (DNS), on the precision of the standard $k-\varepsilon$ turbulence model. Therefore, the model validation has to take up a crucial role in this study. A number of validation exercises were performed. As almost all the experimental measurements of cross-ventilation available in the literature were taken on low-rise structures, we had to configure our CFD accordingly in the model validation exercise.

The cross-ventilated building model used in the wind tunnel experiment in Ohba et al. [45] was replicated by the CFD. The experimental model was sized 300 mm × 300 mm × 150 mm (length × width × height. The building height served as the reference length scale H). Two small openings with dimension 60 mm × 30 mm (width × height) were installed at the middle of both the windward and the leeward walls. The origin was assigned at the middle of the windward façade on the floor (Fig. 1). The experiment was conducted in an open-return type wind tunnel of size 14.0 m × 1.2 m × 1.0 m (length × width × height). A wind profile in

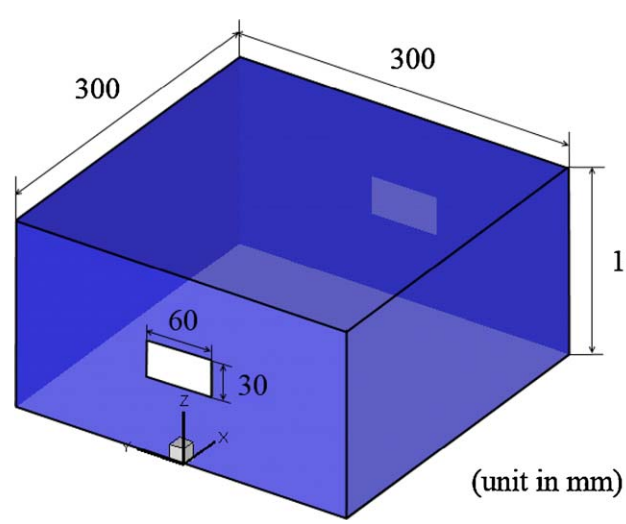

Fig. 1. The cross-ventilated building model studied by Ohba et al. [45].