J. Wind Eng. Ind. Aerodyn. 104–106 (2012) 408–418

Contents lists available at SciVerse ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

CFD simulation of cross-ventilation flow for different isolated building configurations: Validation with wind tunnel measurements and analysis of physical and numerical diffusion effects

R. Ramponi [a,b], B. Blocken [b,*]

[a] Building Environment Science and Technology Department, Politecnico di Milano, via Bonardi 3, 20133 Milano, Italy
[b] Building Physics and Services, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

ARTICLE INFO

Available online 13 March 2012

Keywords:
Computational Fluid Dynamics (CFD)
Complex airflow
Experimental validation
Artificial diffusion
Numerical dissipation
Parametric analysis

ABSTRACT

Computational Fluid Dynamics (CFD) has become one of the most important tools for the assessment of natural cross-ventilation of buildings. To ensure the accuracy and reliability of CFD simulations, solution verification and validation studies are needed, as well as detailed sensitivity studies to analyse the impact of computational parameters on the results. In a previous study by the present authors, the impact of a wide range of computational parameters on the cross-ventilation flow in a generic isolated single-zone building was investigated. This paper presents the follow-up study that focuses in more detail on validation with wind tunnel measurements and on the effects of physical and numerical diffusion on the cross-ventilation flow. The CFD simulations are performed with the 3D steady Reynolds-Averaged Navier–Stokes (RANS) approach with the SST k–ω model to provide closure. Validation of the coupled outdoor wind flow and indoor airflow simulations is performed based on Particle Image Velocimetry (PIV) measurements for four different building configurations. The analysis of numerical diffusion effects is performed in two parts. First, the effect of physical diffusion is analysed by changing the inlet profiles of turbulent kinetic energy within a realistic range. Second, the effect of numerical diffusion is investigated by changing the grid resolution and by applying both first-order and second-order discretisation schemes. The results of the validation study show a good to a very good agreement for three of the four configurations, while a somewhat less good agreement is obtained for the fourth configuration. The results of the diffusion study show that the effects of physical and numerical diffusion are very similar. Along the centreline between the openings, these effects are most pronounced inside the building, and less pronounced outside the building. The velocity-vector fields however show that increased physical and numerical diffusion decreases the size of the upstream standing vortex and increase the spread of the jet entering the buildings. It is concluded that diffusion is an important transport mechanism in cross-ventilation of buildings, and that special care is needed to select the right amount of physical diffusion and to reduce the numerical diffusion, using high-resolution grids and using at least second-order accurate discretisation schemes.

1. Introduction

Natural ventilation of buildings is an important approach towards a sustainable and energy-efficient built environment. Natural ventilation can be driven by wind-induced pressure differences or by thermally-induced pressure differences or by a combination of both (e.g. Linden, 1999; Hunt and Linden, 1999; Li and Delsante, 2001; Heiselberg et al., 2004; Larsen and Heiselberg, 2008; Chen, 2009; van Hooff and Blocken, 2010a). A distinction can be made between cross-ventilation and single-sided ventilation (e.g. Jiang and Chen, 2002; Jiang et al., 2003; Evola and Popov, 2006; Tablada et al., 2009; Caciolo et al., 2012). In the present paper, the focus will be on cross-ventilation.

In the past decades, Computational Fluid Dynamics (CFD) has become one of the most important tools in ventilation research (Chen, 2009). This is also true for natural cross-ventilation of buildings, as illustrated by the very large number of CFD studies that have been published in the past 20 years (e.g. Kato et al., 1992; Straw et al., 2000; Jiang and Chen 2002; Jiang et al., 2003; Murakami et al., 2004; Heiselberg et al., 2004; Mochida et al., 2005, 2006; Seifert et al., 2006; Wright and Hargreaves, 2006; Hu et al., 2008; Stavrakakis et al., 2008; Evola and Popov, 2009; van Hooff and Blocken, 2010a, 2010b; Norton et al., 2010; Kobayashi et al., 2010; Nikas et al., 2010; Ramponi and Blocken, 2012).

* Corresponding author. Tel.: +31 40 247 2138; fax: +31 40 243 8595.
E-mail address: b.j.e.blocken@tue.nl (B. Blocken).

0167-6105/$ - see front matter
doi:10.1016/j.jweia.2012.02.005

In CFD simulations, accuracy and reliability are main concerns. It is widely recognised that CFD simulations can be very sensitive to the large number of computational parameters that have to be set by the user. Therefore, CFD verification and validation studies are imperative, as well as detailed sensitivity studies that can provide guidance in the selection of computational parameters for future CFD studies. Recently, an extensive sensitivity study was performed by Meroney (2009). Later, the present authors have performed a detailed review of the literature, followed by an extensive sensitivity study for a generic isolated building (Ramponi and Blocken, 2012). The impact of a wide range of computational parameters was investigated, including the size of the computational domain, the resolution of the computational grid, the inlet turbulent kinetic energy profile of the atmospheric boundary layer, the turbulence model, the order of the discretisation schemes and the iterative convergence criteria. It should be noted that this study only focused on the Reynolds-Averaged Navier–Stokes (RANS) approach. This was motivated by the fact that, although Large Eddy Simulation (LES) is intrinsically more accurate than RANS, the RANS approach is still most often used. Indeed, a recent review of CFD cross-ventilation studies by the authors (Ramponi and Blocken, 2012) indicated that out of 39 studies analysed, 32 were based on the RANS approach, 3 on Large Eddy Simulation (LES) and 1 on Detached Eddy Simulation (DES), while among the remaining 3 studies, 2 studies applied both RANS and LES and 1 applied RANS, LES and DES.

The previous sensitivity study by the authors however only focused on a single building configuration, and it did not focus in detail on the effects of physical and numerical diffusion on the simulation results. Numerical diffusion refers to the fact that the simulated airflow exhibits a higher diffusivity than the real airflow. It is not a real phenomenon, but its effect on the flow is the same as that of increasing the real (physical) diffusivity. Numerical diffusion arises from truncation errors due to the discretisation of the governing flow equations. The amount of numerical diffusion is directly related to the resolution of the computational grid. Spurious numerical diffusion is also known to occur when first-order discretisation schemes are used. The effect of numerical diffusion will be largest when the real diffusion is small, i.e. when the flow is dominated by convection. It is interesting to note that the importance of limiting numerical diffusion is emphasised by the Journal of Fluids Engineering Editorial Policy (ASME, 2011), incited by contributions by Roache et al. (1986) and Freitas (1993), which demands at least formally second-order accurate spatial discretisation. For this reason, also the best practice guidelines for CFD in general (Casey and Wintergerste, 2000) and CFD in environmental wind engineering (Franke et al., 2007; Tominaga et al., 2008) prescribe the use of second-order accurate discretisation schemes.

The present paper provides a validation study of cross-ventilation flow for four different isolated building configurations. The simulation results are compared with Particle Image Velocity (PIV) wind tunnel measurements by Karava et al. (2011). The present paper also provides an evaluation of the effects of physical and numerical diffusion on cross-ventilation flow in these four building configurations. Coupled CFD simulations of outdoor wind flow and indoor airflow are performed with the 3D steady RANS approach and the SST k–ω turbulence model (Menter, 1994). This turbulence model was chosen because of its superior performance compared to other RANS models for cross-ventilation of a simple isolated building, as shown in the sensitivity study by Ramponi and Blocken (2012). The amount of physical diffusion is varied by changing the inlet profiles of turbulent kinetic energy within a realistic range. The amount of numerical diffusion is varied by changing the grid resolution and by applying both first-and second-order discretisation schemes.

2. Wind tunnel experiments

Wind tunnel measurements with Particle Image Velocimetry (PIV) were performed to analyse cross-ventilation flow of simplified building models (Karava et al., 2011). The experiments were performed in the open-circuit Boundary Layer Wind Tunnel of Concordia University in Montreal (Stathopoulos, 1984). The wind tunnel is 12 m long and has a cross-section of 1.8×1.8 m^2 but the measurements were performed in a small extension of the wind tunnel added downstream of the turntable. The building models, at a scale of 1:200, were built from a 2 mm cast transparent polymethylmethacrylate (PMMA) sheet and had dimensions of $W \times D \times H = 100 \times 100 \times 80$ mm^3 ($20 \times 20 \times 16$ m^3 in full scale) (Fig. 1). Different configurations were obtained by changing the mutual position of the openings at the bottom (h=20 mm), centre (h=40 mm) and top (h=60 mm) of the windward and leeward walls of the building model. The area of openings was also varied to test different wall porosities (w.p.), i.e. different ratios between the areas of the openings and the areas of the facade walls. A fixed opening height of 18 mm (3.6 m in full scale) was used, while the opening width was varied between 22 mm (4.4 m in full-scale; w.p.=5%), 46 mm (9.2 m in full-scale; w.p.=10%) and 88 mm (17.6 m in full-scale; w.p.=20%). In this paper, the configurations with the openings at the centre (configuration 1) and the bottom (configuration 2) of the opposite walls are considered, with wall porosities of 5% (Cases 1.05 and 2.05) and 10% (Case 1.10 and 2.10), as shown in Fig. 1.

The measurements were performed with the model placed in the extension of the wind tunnel with the openings perpendicular to the streamwise direction. An upstream roughness profile corresponding to open terrain (z_0=0.005 m in full scale) was obtained by placing extruded polystyrene (XPS) cubes far upstream and a carpet less far upstream of the wind-tunnel turntable (Karava, 2008). The incident vertical profiles of mean wind speed and streamwise turbulence intensity were measured with a hot-film probe at the building position. A reference mean wind speed U_{ref}=6.97 m/s and a streamwise turbulence intensity of 10% were reported at building height (H=80 mm), while the turbulence intensity was about 17% near the ground level (12 mm) and about 5% at gradient height (738 mm). Fig. 2 shows the measured mean wind speed and turbulence intensity profile, as well as the fitted logarithmic law, which will be used as inlet boundary conditions in the simulations in this paper. The values in the log law are z_0=0.025 mm (reduced scale) and u^*=0.363 m/s. The PIV measurements were conducted in the vertical centreplane for both configurations 1 and 2, as indicated in Fig. 1. Further information about the measurements can be found in Karava (2008), Karava et al. (2011) and Karava and Stathopoulos (2011).

3. CFD simulations: computational settings and parameters

3.1. Computational domain and grid

The CFD simulations were performed at model scale. The dimensions of the computational domain were determined based on the existing best practice guidelines (Franke et al., 2007; Tominaga et al., 2008), except for the upstream length (i.e. distance between inlet plane and windward building facade), which was taken equal to 3H instead of 5H, in order to limit the development of unintended streamwise gradients (Blocken et al., 2007a, b). A test simulation has shown that using 3H is justified, because the extent of the upstream disturbance of the flow pattern by the building is less than 3H. The resulting dimensions of the domain were width × depth × height = $0.9 \times 1.54 \times 0.48$ m^3