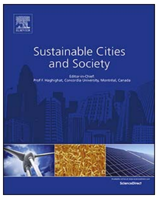
Sustainable Cities and Society 37 (2018) 492–500

Contents lists available at ScienceDirect

Sustainable Cities and Society

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



Effects of minor changes in the mean inlet wind direction on urban flow simulations

D.J. Wise[a,*], V.B.L. Boppana[a], K.W. Li[b], H.J. Poh[a]

[a] Institute of High Performance Computing, 1 Fusionopolis Way, #16-16 Connexis North, Singapore 138632, Singapore
[b] Building & Research Institute, Housing and Development Board, 480 Lor 6 Toa Payoh, HDB Hub, Singapore 310480, Singapore

ARTICLE INFO

Keywords:
Computational fluid dynamics (CFD)
Simulation validation
Ventilation
Wind flow

ABSTRACT

There has been increased prevalence in the use of computational fluid dynamics (CFD) for the assessment and planning of the urban environment in recent years, spurred not only by the decreasing costs of computational resources, but also by increasing demand for enhanced living conditions, especially in densely populated and land-scarce places like Singapore. In spite of this, there are still many unknowns as to how the simulation parameters affect the prediction accuracy. In the present work, an assessment is carried out to investigate how small modifications to the mean inlet wind direction affect the simulation results. Field measurements performed within a town in Singapore are compared to three-dimensional (3D) steady Reynolds-averaged Navier–Stokes (RANS) simulations for seven inlet wind directions within a 30° sector, using the realizable k-ϵ model. The driving conditions for the simulations, i.e. the overlying wind conditions, are derived from nearby meteorological station data, which is also used in the sampling of the field measurements. The simulation results compare favourably with the field measurements for the wind speed, while the veracity of the computed wind directions is highly dependent on the immediate surroundings of the sensors. Additionally, major changes in the simulated flow patterns are observed even for small changes in the mean inlet wind direction. A single simulation result cannot be considered representative of a specific mean inlet wind direction – this is evidenced by high variation in the field measurements even after stringent sampling criteria. The results have implication particularly in the field of urban planning, and highlight the care that should be taken when CFD results are used as part of the decision-making process.

1. Introduction

Computational fluid dynamics (CFD) studies are frequently used in the assessment of the wind environment in urban areas. Simulations can provide insights into wind loading (Montazeri & Blocken, 2013), ventilation (Ramponi, Blocken, Laura, & Janssen, 2015), the pedestrian wind environment (Blocken, Janssen, & van Hooff, 2012; Tominaga et al., 2004), pollutant dispersion (Gousseau, Blocken, Stathopoulos, & Van Heijst, 2011; Tominaga & Stathopoulos, 2011), and thermal comfort (Dimoudi et al., 2014; Tominaga, Sato, & Sadohara, 2015). Due to the variety of commercially available software for performing these simulations, as well as the ever-decreasing costs of the necessary computational resources, CFD simulations are becoming widely used as part of the design and tender process in the construction of new buildings (Fletcher, Mayer, Eghlimi, & Wee, 2001). In order to standardize as well as impose quality control on the simulation process, several guidelines have been developed.

The first set of recommendations for the use of CFD within the field of urban wind engineering was that of Franke et al. (2004), who provided guidelines specifically for the prediction of mean velocity and turbulence intensity within the built environment. Therein are outlined recommendations on the turbulence model, computational domain size, level of detail in the building geometry, boundary conditions, convergence levels and necessary grid convergence tests. In addition to Franke et al. (2004), there are further recommendations on the standardization of urban CFD, excellent examples of which can be found in Blocken (2015), and in works from the Architectural Institute of Japan (AIJ), for example within Tominaga et al. (2008) and Tamura, Nozawa, and Kondo (2008). Within the present work however, we will be predominantly following the recommendations presented by Franke et al., as these are still followed by many industrial practitioners of urban CFD.

One aspect that is only addressed briefly by Franke et al. and which is the focus of the present paper is the inlet flow direction considered in

---

[*] Corresponding author.

E-mail addresses: daniel-wise@ihpc.a-star.edu.sg (D.J. Wise), boppanavbl@ihpc.a-star.edu.sg (V.B.L. Boppana), Kelvin_WH_LI@hdb.gov.sg (K.W. Li), pohhj@ihpc.a-star.edu.sg (H.J. Poh).

https://doi.org/10.1016/j.scs.2017.11.041
Received 10 August 2017; Received in revised form 1 November 2017; Accepted 28 November 2017

the simulation. Therein it is recommended to use nearby metereological stations to determine a reference wind speed at a reference height, such that the logarithmic profile at the inlet can be fit to these values. This is a valid proposal for industrial CFD; as the most common methodology is still to use the Reynolds Averaged Navier–Stokes (RANS) approach and therefore considers only a single mean value for the inlet wind direction. In the case where the overlying wind conditions vary seasonally – as is the case for Singapore where during the two monsoon seasons the dominant wind directions are either South West or North East – multiple wind directions should be simulated.

The question as to how many directions are necessary is addressed by further sets of guidelines. Work by Yoshie et al. (2007) as part of the AIJ project for CFD prediction of wind environment suggests 16 possible wind directions should be considered, while it was also recommended by Ng (2009) that for Air Ventilation Assessment (AVA) of a high-density city such as Hong Kong, 16 wind directions are investigated. It should be noted here that Ng (2009) states a preference for the use of wind tunnel assessment rather than CFD, citing reliability issues, however this is a topic which we will not address here. In both of these works however, the inlet wind directions that should be considered are spread evenly over 360°, meaning that each simulation or wind tunnel experiment is assumed to be representative of a 22.5° sector.

The concept of AVA is of particular interest to urban planning in Singapore, and so will be elaborated on here. The urban heat island (UHI) effect – i.e. the increase in temperature in an urban environment compared to neighbouring rural areas – is a major issue being faced by cities worldwide. It is not limited to a certain continent or stage of economic development, and has been verified to occur in cities as diverse as London (Jones & Lister, 2009), Montreal (Wang & Akbari, 2016), Surabaya (Kurniati & Nitivattananon, 2016), and Venice (Peron, De Maria, Spinazzè, & Mazzali, 2015). There are several contributions to literature documenting the UHI effect in Singapore (Chow & Roth, 2006; Jusuf, Wong, Hagen, Anggoro, & Hong, 2007), and promoting solutions for its mitigation to improve the comfort of residents (Priyadarsini, Cheong, & Wong, 2004; Wong et al., 2003). Additionally, Singapore experiences relatively low wind speeds, with the yearly average being less than 1 m/s (Liping & Wong, 2007). As thermal comfort improves with air movement (Parkinson & de Dear, 2016), it is important in the planning stages of urban development to consider how the wind might flow through the buildings. Guidelines for improving ventilation in the built environment are given by Ng (2009), albeit not in the context of heat island mitigation. Therein it is stated that "designs and developments should focus on not blocking the incoming wind, as well as minimizing the stagnant zones at pedestrian level". Using this suggestion urban planners can observe wind patterns generated by wind tunnel experiments or CFD, and make adjustments on their designs in order to improve both ventilation and thermal comfort.

In some very recent studies (García-Sánchez, Philips, & Gorlé, 2014; Margheri & Sagaut, 2016), the impact of uncertainties in the inlet parameters on the resulting flow field are investigated using 'uncertainty quantification' techniques. This is still an emerging and sophisticated approach for CFD in the built environment, and it is likely to take a few more years before the building industry considers this approach.

Based on the field measurements obtained during the Joint Urban 2003 field campaign, Klipp (2007) discusses the dependence of atmospheric boundary layer turbulence parameters on the wind direction. This is crucial as the wind direction determines the upwind surface characteristics which is encapsulated in the aerodynamic roughness length z_0, and the surface friction velocity u_* – these are some of the essential inputs in computations. The works of Toparlar et al. (2015) and Blocken, van der Hout, Dekker, and Weiler (2015) provide good examples of considering various values of z_0 depending on the inlet wind direction.

Within the present work we attempt to address via CFD simulations whether considering a single simulated wind direction as representative of a 22.5° sector is a valid assumption. While unsteady RANS simulations are possible and useful, we instead choose to perform steady RANS simulations. This is because they are widely used by architects and planners to make informed decisions during the production of a master plan, or while proposing modifications to a current plan. As such, henceforth when we refer to the inlet wind direction we are referring to the mean inlet wind direction as within RANS simulations the fluctuations due to turbulence are not considered. The simulations are conducted on a typical residential district in Singapore. Seven inlet wind directions are considered within a 30° sector, and the results are compared to field measurements. The simulation results are also analyzed with respect to the flow patterns, i.e. generated wind corridors and recirculation zones, and with respect to the ventilation quality of the urban area. The simulation strategy and a brief description of the field measurements are presented in Section 2, and comparison between the simulation results and field measurements, and analysis of the simulated flow patterns are given in Section 3.

2. Methodology

2.1. Geometry, computational domain and mesh

The building and terrain geometries were provided by the Housing Development Board of Singapore (HDB) in 3DS file format. These models were converted to a usable format by Right Dimension Pte. Ltd.: the buildings were converted to STL file format, while the terrain was converted to a primitive digital elevation model (DEM) at a resolution of 1 m in the horizontal directions and 0.25 m in the vertical direction. An aerial view of the simulated area is denoted by the red outline in Fig. 1a, within this 1 km × 1 km region the buildings and terrain are explicitly resolved. As the outer limit of the terrain DEM was not homogeneous and varied from 10 m to 20 m in elevation, it was necessary to artificially extend it such that at the outer limit of the explicitly resolved terrain there would be uniform elevation. This was done by setting a border approximately 200 m from the simulation area to zero elevation, and recreating the terrain within the border limits using biharmonic inpainting (Shen & Chan, 2002). This method ensures that the synthesized terrain is second-order smooth, therefore there are no sharp changes in the elevation (i.e. no sudden steps) which could create erroneous flow patterns. The intent of this method was to reduce the effects of any terrain discontinuities on the simulation results. As the region-of-interest consists of a densely built up area, and as the flow in this region is dominated by these building length-scales, it is unlikely that the terrain recreation process affects the simulation results. However the method chosen limits any non-physical flow patterns that might have occurred if there were sharp changes in the terrain surrounding the buildings.

The geometry predominantly consists of residential buildings with multi-storey car parks and is shown in Fig. 1b. It also consists of a temple to the north-east of P5 and three school buildings – (i) north-west of P6, (ii) south-east of P8 and (iii) north-east of P8. The heights of the buildings in this computational domain range from approximately 10 m to 70 m. Most of the residential buildings vary in height from 50 to 70 m, and the multi-storey car parks located amidst the residential blocks have heights from 20 to 30 m. Note that there are buildings in the computational domain that do not appear in the aerial photo (highlighted by the green rectangles in Fig. 1a and b), this is due to the age of the aerial photo. These buildings have been present throughout the full period of the field data collection. The cyan dashed circles in Fig. 1a and b indicate the area used for the ventilation assessment in Section 3.3. The computational domain size is 3.7 km × 3.7 km × 0.56 km. In terms of the tallest building height h, the horizontal extents of the computational domain from the outer edge of the buildings span approximately 20 h, while the vertical extent is 8 h. The larger horizontal extents are chosen so that the same