Journal of Wind Engineering & Industrial Aerodynamics 172 (2018) 85–95

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

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

Numerical simulation of downburst wind flow over real topography

El-Sayed Abd-Elaal [a,b,*], Julie E. Mills [a], Xing Ma [a]

[a] School of Natural and Built Environments, University of South Australia, Australia
[b] Department of Structural Engineering, Faculty of Engineering, Mansoura University, Egypt

ARTICLE INFO

Keywords:
Downburst
Real topography
Numerical simulation
Wind speeds
Speedup factors

ABSTRACT

Downburst winds cause significant threats for many structural systems. Most failure reports of transmission line systems point to downburst events rather than other types of wind events. These events produce the highest wind speeds at low elevations and the shape of terrain topology is one of the main parameters that significantly changes the distribution of the surface wind speeds. The main objective of this study is to investigate the profiles of downburst wind speeds as they pass over real topography, and then to provide guidance for the design of transmission line systems in similar terrains. Numerical simulations of downburst wind flow over two samples of real topography were conducted and the consequent changes in horizontal and vertical downburst wind speeds were investigated.

1. Introduction

Downburst winds damage many buildings every year and are the main cause of transmission line failures in many areas of the world. For example, in the last ten years, several hundred accidental failures of power lines and more than a thousand rooves have been damaged throughout Australia due to severe wind events such as downbursts and tornadoes (Bureau of Meteorology, 2017). Downbursts are the most common cause of severe winds in several regions around the world (Boss, 2010; Chay et al., 2006) and they are the cause of the highest wind speeds at low height in Australia (Holmes, 2002). Therefore, the impact of topographic features on resultant downburst wind speeds is an important factor that should be considered during the design of different surface level structural systems.

Many researchers have examined the effects of topographic features on boundary layer winds, e.g. Jackson and Hunt, 1975; Bowen and Lindley, 1977; Pearse et al., 1981; Taylor et al., 1987; Glanville and Kwok, 1997; Holmes et al., 1997; Uchida and Ohya, 1999; Bowen, 2003; Burlando et al., 2007, and O'Sullivan, 2012. However, few researchers have investigated the effect of topology for localized wind events. Selvam and Holmes (1992) undertook early investigations into the effects of topographic features on downburst winds. They investigated the changes of downburst wind speeds over a single hill of slope gradient = 0.25 and concluded that the resultant changes are generally much less than for boundary layer (synoptic) winds. Letchford and Illidge (1998, 1999) measured speedup factors for downburst winds on various topographic features and different slope gradients. They concluded that the crest speedup factors increase with the embankment gradient, and decrease as the embankment is placed further from the impact point.

Wood et al. (2001) investigated the effects of the locations of topographic features and concluded that there was a slight decrease in topographic speedup factors with increasing distance between the testing surface and the jet outlet. They also indicated that the given topographic speedup factors for embankments in the Australian Wind Code, AS1170.2–1989 (Australian Standards, 1989) were conservative except at heights near ground level above the embankment crest. Wood et al.'s (2001) exception and the outcomes of Letchford and Illidge (1998, 1999) point to the need for investigating the near ground downburst flow over more types of topographic features.

Otsuka (2006) investigated more complicated features using a simple model of the moist atmosphere with warm rain. Mason et al. (2007) conducted numerical and experimental simulations for stationary downbursts over three topographical features, namely an escarpment, a triangular hill and a bell-shaped hill using an impinging jet model. They investigated the appropriate turbulence models for numerical simulation of impinging jet steady state flow simulation. Then, Mason et al. (2010) conducted numerical simulations for stationary downburst events over two topographical features, namely an escarpment and a bell-shaped hill, using a non-hydrostatic sub-cloud model. They also conducted a parametric study to investigate the influence of topography location, topography size and downburst size on downburst flow structure. Wakes et al. (2010) concluded that using simplified geometry in the topography

* Corresponding author. School of Natural and Built Environments, University of South Australia, Australia.
E-mail address: abdee001@mymail.unisa.edu.au (E.-S. Abd-Elaal).

https://doi.org/10.1016/j.jweia.2017.10.026
Received 24 August 2016; Received in revised form 29 October 2017; Accepted 29 October 2017

description is insufficient for accurate simulation, given that it is the topography that causes a significant effect on the wind flows.

This work investigates the behaviour of a downburst outflow over real topography. 3D numerical simulations for unsteady state downburst events have been conducted over two different samples of real topography, then the effects of surface topology on the simulated wind speeds have been inspected and an evaluation of the relevant provisions of design guidelines has been presented. Investigating the assumption of averaging the slopes for undulating terrain over a horizontal length of 500 m as suggested in the Guide to AS/NZS1170.2:2002 (Holmes et al., 2012) was one of main questions addressed in this study.

2. Downburst model

2.1. Model dimensions

Many researchers have demonstrated that impinging jet models are appropriate for representing the flow structure of downburst winds (Kim and Hangan, 2007; Xu and Hangan, 2008; Zhang et al., 2013). The commercial Computational Fluid Dynamics (CFD) software, ANSYS CFX 15.0 (ANSYS CFX Reference Guide, 2013) has been employed in this study for simulating downburst flow.

The model's dimensions have been suggested in proportion to scaled approximate dimensions of real downburst events. Fujita (1981) concluded that the diameter of microbursts ranges between 400 m and 4.0 km and Wilson et al. (1984) suggested 1,000 m as an approximate diameter for the downbursts. For the current study the diameter of the downburst was chosen to be 750 m based on the lower end of Fujita's (1981) and near to Wilson et al.'s (1984) suggestions. The small diameter was preferred to reduce the computational cost.

Hjelmfelt (1988) concluded that the height of the cloud base for downburst events is between 2.3 km and 3.3 km. Accordingly, the jet height was selected to equal 3.5D where D is the downburst diameter. Sengupta and Sarkar (2008) inspected different configurations for the positions of the jet nozzle and the related domain shapes for numerical simulation of downbursts. They concluded that the variation in the resultant downburst wind speeds with the different domain configurations and shapes was less than 2.8% and hence this was not critically important. The shape of the adopted domain is one of Sengupta and Sarkar's (2008) domain shapes and very similar to the one previously used by Abd-Elaal et al. (2013). The radial distance from the jet centre line to the outlet side is 7.0 D. Hence the simulated area per model extended to approximately 7D = 5.25 km from its centre and two numerical models were established.

Because of the high computational cost of modelling the suggested real topology (see Section 2.2 and 3.3), a small part of a 3D computational domain equal to one eighth of a cylinder was used instead of the whole cylindrical domain. Using a part of a cylindrical domain instead of the full cylindrical domain was validated by Li et al. (2012) and Abd-Elaal et al. (2013). Fig. 1 shows the shape and dimensions of the adopted domain.

Kim and Hangan (2007) and Abd-Elaal et al. (2013) investigated the scale dependency (Reynolds number dependency) for the numerical simulation of steady state and unsteady state downburst flow, respectively. Their results showed a significant divergence in the resultant wind speed near the ground. It is suggested that this divergence is due to the large change in the non-dimensional distance y^+ between the two scales of models, where $y^+ \approx 105$ for a large model, which is too far from the thickness of the viscous sub-layer as recommended by Menter (2002). In addition, flow simulation using high Reynolds number can fail due to the equilibrium assumption of turbulence in wall functions (Chen, 1995). Xu and Hangan (2008) concluded that the Reynolds number is significant for $R_e < 2.7 \times 10^4$. In this study, a small scale model (scale 1 to 1000, $R_e = 2.1 \times 10^5$) has been used.

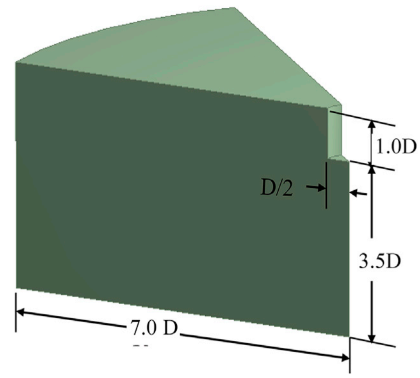

Fig. 1. Numerical geometry and dimension for downburst domain.

2.2. Simulation time period

Kim et al. (2007) analysed a full-scale downburst event and suggested 29 m/s for the inlet jet velocity for full scale CFD simulation. Abd-Elaal et al. (2013) also analysed several full-scale events and suggested that the time periods of impinging inlet jet flow for full scale downburst events are in a range from 4 min to 7 min if compared to an equivalent downburst with a diameter equal to 750 m and with inlet jet velocity equal to 30 m/s. They also concluded that, the different profiles and time periods of decaying inlet speed during simulation had only a minimal effect on the results.

Accordingly, the suggested parameters for a full-scale downburst wind model could be D = 750 m, H = 3.5D and $V_{jet} = 30$ m/s, $\Delta t = 7$ min and $R_e = 1.26 \times 10^9$, where D is the jet diameter, H is the jet height, V_{jet} is the inlet jet velocity and Δt is the time period of impinging inlet jet flow. These parameters have then been scaled by Shehata et al.'s (2005) procedure to D = 0.75 m, H = 3.5D and $V_{jet} = 5$ m/s, $\Delta t = 2.52$ s and $R_e = 2.1 \times 10^5$ and utilized in this study.

2.3. Grid arrangements and boundary conditions

Most of the researchers who have studied wind flow over real topography have used body-fitted coordinates which are characterized by their ability to represent the ground boundary surfaces with high accuracy (Bitsuamlak et al., 2004). The O-Grid mesh has been used in the presented model as it is one of the best grid types for fitting complex topologies. ANSYS ICEM CFD was employed to establish the mesh because of its advanced O-Grid tools, which make it easier to accomplish complicated geometry and generate many vertices to adjust the surface blocking to ideal locations on the real topography terrain.

The domain was divided into a total grid point count of approximately 4.0 million. Grids have been utilized along the radial direction to ensure that the flow direction coincided with the mesh lines. Stretching was employed in both the radial and vertical directions to focus the resolution near the ground and the impinging jet centre line.

In the vertical direction, the grid spacing was stretched from $\Delta z = 0.002$ m above the ground to 0.1 m at the top of the model. The minimum value of Δx was 0.003 m at the inlet of the impinging jet, stretching to 0.13 m at the outlet side. A free slip condition was imposed for the wall nozzle surface, a pressure outlet condition was imposed for the upper and lateral faces and a symmetrical condition was imposed for the side faces. The ground plane was treated as a rough wall surface with surface roughness z_0 equal to 0.002 cm (equivalent to 0.02 m at full scale), which is equivalent to open terrain or category 2 in the Australian wind load standard AS/NZS 1170.2:2011 (Standards Australia, 2011).

2.4. Choice of the turbulence model

As the current study investigates the profiles of downburst wind