J. Wind Eng. Ind. Aerodyn. 135 (2014) 182–190

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

Circumferential analysis of a simulated three-dimensional downburst-producing thunderstorm outflow

Leigh G. Orf[a], Chris Oreskovic[b], Eric Savory[b,*], Erica Kantor[b]

[a] Department of Earth and Atmospheric Sciences, Central Michigan University, USA
[b] Department of Mechanical and Materials Engineering, University of Western Ontario, London, Ontario, Canada N6A 5B9

ARTICLE INFO

Keywords:
Downburst
CM1 cloud model
Thunderstorm
Numerical modeling

ABSTRACT

A high-resolution simulation of a downburst-producing thunderstorm has been conducted using a three-dimensional cloud model. The highly asymmetric near-ground outflow within the downburst has been analyzed within an axi-symmetric framework, by undertaking circumferential spatial averaging of the wind speed components, in order to explore whether the salient features of the outflow can be represented using the simpler models employed in wind engineering. The results indicate that this approach yields, at a given time during the event, outflow profiles that preserve the regions of the highest wind speed but do not preserve any flow structure. It is suggested that the peak radial wind speed around any circumference may be computed from the spatially-averaged mean multiplied by a consistent peak factor.

1. Introduction

Downbursts are intense downdrafts that form within thunderstorms, causing damaging winds near the earth's surface (Fujita, 1985). These winds are induced by thermodynamic and kinematic processes involving rain, snow and graupel (small hail) within the thunderstorm. Thermodynamic cooling induced by evaporation, melting and sublimation causes pockets of negative buoyancy, forcing a downdraft. Heavy rain and falling graupel also induce drag, further enhancing downdrafts. Observational and numerical studies of downbursts and downburst-producing thunderstorms have previously been conducted to better understand the environmental conditions conducive to downburst formation and the structure of the downburst winds (e.g. Hjelmfelt, 1987; Proctor, 1988; Orf et al., 1996).

Because of the damage to structures caused by downbursts, especially to electricity transmission towers and lines (Dempsey and White, 1996), over the past decade there has been considerable research into the wind flow near the ground associated with these events. The wind engineering approach to modeling the near-ground wind profiles has evolved from simple empirical models based on a steady wall jet, such as that of Holmes and Oliver (2000), which was then applied to a transmission line tower (Savory et al., 2001), through to the unsteady, transient, circular impinging jet model, produced both experimentally (e.g. Mason et al., 2005; Sengupta and Sarkar, 2008; McConville et al., 2009) and using computational fluid dynamics (CFD) simulations (e.g. the Unsteady Reynolds-Averaged Navier Stokes (URANS) model of Kim and Hangan (2007)), which showed that the vertical profiles of the radial wind speed for transient impinging jets were very different from those of a steady jet for the same surface roughness and, more recently, the Detached Eddy Simulation of Abd-Elaal et al. (2013). The attraction of the transient impinging jet model is that it allows for a simple linear scaling of events, both in terms of their intensity (by linear scaling of the time-history of the jet nozzle exit velocity, V_j) and size (by scaling the wind field with the jet nozzle exit diameter, D_j). This has allowed parametric studies of the impact of different events on member loads and failure mechanisms of electricity transmission lines and towers by coupling the downburst wind fields with a finite element structural analysis model of the transmission line system (Shehata et al., 2005; Shehata and El Damatty, 2007, 2008; Darwish et al., 2010). One of the drawbacks of the experimental simulation of downbursts using an impinging jet is that the height of the outflow region, of interest to wind engineers, is only a very small percentage of the nozzle diameter, with the height to the peak radial wind speed, for example, being 3–5% of the nozzle diameter (Kim and Hangan, 2007). To help overcome this limitation a part-simulation of the outflow region only, using a 2-D transient slot jet has been developed and incorporated into an existing large boundary layer wind tunnel (Lin et al., 2007; Lin and Savory, 2010), permitting studies of the effect of downburst outflows on an aeroelastic model of a transmission tower and line (Lin et al., 2012).

* Corresponding author. Tel.: +1 519 661 2111x88256.
E-mail address: esavory@eng.uwo.ca (E. Savory).

http://dx.doi.org/10.1016/j.jweia.2014.07.004
0167-6105/

The major drawback of the impinging jet model is that it contains none of the physics of real events. Pragmatically, one might argue that this does not matter because such models appear to produce a flow structure and wind profiles that resemble field observations of downbursts. However, the impinging jet model does not allow us to say anything about the conditions of formation of these events and, hence, anything about the likelihood of occurrence of a given size and intensity of event when a thunderstorm occurs. Such information is essential for developing structural design guidelines, for transmission lines, for example. To the first order, it is clear that real events do not display linearity between any initial downdraft column velocity and the resulting near-ground radial wind speeds. Downbursts are non-linear events in which the potential energy of a dense (cold) fluid mass is converted into kinetic energy (velocity squared) with the process augmented by the drag of precipitation and phase changes of that precipitation. Hence, as shown by the Large Eddy Simulations (LES) of Vermeire et al. (2011a), a more realistic model is that of a cooling source in which negatively-buoyant, colder air falls within a warmer atmosphere, with the resultant vorticity being generated baroclinically. Others have also adopted this approach using URANS models (e.g. Mason et al., 2009, 2010; Zhang et al., 2013). Replicating the cooling source approach experimentally is a challenge, due to the Froude number scaling required, with the release of a parcel of dense liquid into a larger mass of less dense liquid being successfully implemented (e.g. Alahyari and Longmire, 1994, 1995), albeit at very small model scale. Very recently, a hybrid impinging jet /density perturbation experimental modeling approach has been proposed (Demarco et al., 2013), with the results confirming the numerical model results of Vermeire et al. (2011a) in that the outflow vortex structure is strongly dependent upon the type of initial forcing; negative buoyancy or jet impulse.

Unfortunately, even the cooling source model has limitations in that it does not include the forcing due to precipitation and the simulations are normally conducted within neutrally stratified ambient surroundings. Cooling source simulations (e.g. Orf et al., 1996; Orf and Anderson, 1999; Mason et al., 2009; Vermeire et al., 2011a, 2011b) typically impose an elevated source of negative buoyancy whose characteristics follow a simple analytical function, with cooling intensity imposed as a function of the cosine squared of the distance from the center of an elongated spheroidally-shaped "blob" of dry air that is symmetric about a vertical axis. This approach attempts to emulate the effect of thermodynamic cooling found in a shaft of hydrometeors descending beneath the cloud base of an air-mass thunderstorm where rain, snow and graupel cool the air via the processes of evaporation, melting, and sublimation (Straka and Anderson, 1993). The cooling-source approach, however, does not include any actual water substance at all and, hence, lacks all the physics involving water substance that occur in clouds, including the effects of hydrometeor drag. Hydrometeor drag dominates downdraft initiation and maintenance when heavy rain occurs in environments with high relative humidity where little, if any, thermodynamic cooling can occur. Downbursts that are initiated in this fashion are called wet downbursts due to the occurrence of heavy rain at the Earth's surface (Fujita, 1985) and cannot be emulated using the cooling source approach. Rather, the nature of the downward momentum in wet downbursts under these environmental conditions is a function of hydrometeor concentration, classification, size distribution and terminal velocity, as well as the interaction between different hydrometeor classes and between hydrometeors and their surrounding environment (e.g. Morrison et al., 2009).

In a cloud model, the processes involving solid and liquid water substance are emulated via a so-called microphysics parameterization (e.g. Lin et al., 1983). Due to the addition of several new prognostic variables and the complex mathematical nature of the highly parameterized microphysical source and sink functions, microphysics routines add a large computational load to a numerical simulation. Full cloud model simulations also require a significantly larger computational domain in order to capture the entire cloud and its surrounding environment. It is only recently that high performance computing has enabled such full-cloud simulations to be conducted at the high resolutions traditionally employed by simpler wind engineering models of downbursts.

All mid-latitude thunderstorms occur in environments containing vertical wind shear that is a function of the larger-scale synoptic conditions in which the thunderstorm occurs. To the best of our knowledge, all cooling source numerical simulations of downbursts in the published literature have occurred in either quiescent environments (e.g. Orf et al., 1996; Mason et al., 2009; Vermeire et al., 2011a,b) or environments with simple wind shear profiles (e.g. Orf and Anderson, 1999; Mason et al., 2010). This is yet another limitation of the simpler models typically employed in simulations of downbursts. In addition to wind shear and translation of the downburst, the simulations summarized in Mason et al. (2010) encompass the effects of tilting of the vertical axis of the downburst column and of topography in the form of escarpments. As the forward tilt angle increases the outflows become deeper, whilst the effects of shear and translation are not simply a superposition of wind speed vector components since the outflow morphology changes with those parameters. In considering all of the different test cases studied in their work they note that "all results intrinsically assume each downburst type has the potential to produce design strength winds. This is an assumption that cannot be justified at this time and requires further research." That statement remains true and, indeed, all of the well-documented physical processes that occur in real thunderstorms listed above are either ignored or treated in a highly analytical way in impinging jet and cooling source simulations. Perhaps, however, the most critical problem with these approaches is that they assume that downbursts are axi-symmetric. On that basis, the question that arises is what is the relationship of any maximum wind speed from such a modeled event (which is assumed to occur anywhere around the circumference of the outflow region at a given radial location) to that occurring in a real event where the radial wind speed, at any given time, may vary significantly around that circumference? Hence, one is not only interested in the residual turbulence, after the coherent part of the velocity associated with the dominant outflow vortex has been subtracted (Holmes et al., 2008; Lin and Savory, 2010), but also the spatial variation of the radial outflow wind speed circumferentially around the event. Such information is important if one wishes to develop a reliable axi-symmetric modeling framework for use in providing design wind speeds.

In the absence of comprehensive field data from downbursts, Orf et al. (2012) described the nature of downbursts produced by a very high resolution cloud model numerical simulation utilizing the CM1 cloud model (Bryan and Fritsch, 2002) and applied simple circumferential analysis of the near-surface outflow during two times corresponding to the strongest horizontal winds, at a single radial location. The present paper extends the scope of the analysis, including comparison with data from simpler numerical models, with, as will be shown in the ensuing discussion, the results suggesting that in terms of peak radial wind speed magnitudes there may be a consistent relationship between the circumferentially-averaged (i.e. nominally axi-symmetric) wind speeds at any given time and the peak wind speed around that circumference at that time. The following section summarizes the numerical modeling approach and the method of data analysis and this is followed by a discussion of the results, conclusions and the proposed future direction of the research program.