Available online at www.sciencedirect.com

SCIENCE DIRECT®

Engineering Structures 28 (2006) 240–254

ENGINEERING STRUCTURES

www.elsevier.com/locate/engstruct

Numerical and analytical simulation of downburst wind loads

M.T. Chay[a,*], F. Albermani[a], R. Wilson[b]

[a] Department of Civil Engineering, University of Queensland, St Lucia Queensland 4072, Australia
[b] Department of Mathematics, University of Queensland, St Lucia Queensland 4072, Australia

Received 13 January 2005; received in revised form 18 July 2005; accepted 27 July 2005
Available online 5 October 2005

Abstract

Researchers and designers currently have a number of methods for numerically simulating either the non-turbulent or turbulent characteristics of downbursts. Examples of non-turbulent downburst winds simulated using a commercially available Computational Fluid Dynamics software package are discussed, as well as a simple analytical model. The significance of the translational velocity of the storm, and the variation in intensity as the event matures and decays is discussed. An ARMA method of adding turbulence to the non-turbulent wind speed is proposed. The various aspects of the model are then integrated to create a method suitable for generating wind speed time histories for the dynamic analysis of lattice structures subjected to downburst winds.

Keywords: Thunderstorm; Downburst; Wind; Simulation; Turbulence; Computational fluid dynamics; Autoregressive moving average process

1. Introduction

Design practices and standards call for wind loads on structures to be evaluated on the basis of an atmospheric boundary layer profile, despite increasing recognition of thunderstorm downbursts as the cause of design wind speeds in many regions of the world. Certainly, downbursts pose a great risk to long span structures, such as transmission lines, which undergo regular failures during thunderstorms [24].

Researchers have proposed many methods of modelling certain aspects of downburst winds. However, there has been little emphasis on developing a comprehensive model of a downburst that is suitable for the generation of wind loads in a time domain structural dynamic analysis.

The wind of an atmospheric boundary layer can be described as the sum of a mean speed ($\bar{U}(z)$), which is a function of height (z), and a fluctuating process $u'(z,t)$, which is a function of height and time (t).

$$U(z,t) = \bar{U}(z) + u'(z,t). \tag{1}$$

Similarly, the wind speed occurring at any point in space (x, y, z) and time within a downburst can be thought of as the sum of two vector components:

$$U(x,y,z,t) = \bar{U}(x,y,z,t) + u'(x,y,z,t) \tag{2}$$

where $U(x,y,z,t)$ is the total wind velocity; $\bar{U}(x,y,z,t)$ is the non-turbulent wind velocity; and $u'(x,y,z,t)$ is the turbulent fluctuation. This form is similar to that of the boundary layer wind, except that now the non-turbulent component is a function of time and location with respect to the storm, and the turbulent component is also affected by the object's relative location to the storm.

This paper discusses methods of modelling several aspects of a downburst and suggests a way in which they may be integrated to produce a model that is able to simulate correlated wind speed time histories at several locations during a storm.

2. Non-turbulent downburst models

A number of non-turbulent wind speed models of downbursts are available to engineers. Two methods are presented here: Computational Fluid Dynamics (CFD), and

* Corresponding author. Fax: +61 7 3365 4599.
E-mail address: m.chay@uq.edu.au (M.T. Chay).

0141-0296/$ - see front matter © 2005 Elsevier Ltd. All rights reserved.
doi:10.1016/j.engstruct.2005.07.007

a modified version of an analytical model originally created for wind shear estimation.

2.1. Computational fluid dynamics (CFD)

2.1.1. Validation and choice of turbulence model

CFD modelling techniques have been used previously to investigate a number of characteristics of near ground downburst winds. Selvam and Holmes [28], and later Wood et al. [34] used a 2-D model to investigate the wind speed increases as downburst winds flowed over a hill. Hangan et al. [13] used a Reynolds stress model to investigate downburst gust front characteristics.

Downbursts are typically simulated as impinging jets, a philosophy which has been adopted in this study. Impinging jet flows are commonplace in today's society, particularly in a variety of manufacturing applications. However, they are particularly difficult to model using CFD, as their flow field is quite complex. As such, the impinging jet has been the subject of much scrutiny, and has been widely used as a test case for turbulence models.

During this study, the commercially available CFD package FLUENT 6.0 [11] was used to simulate 30 downburst scenarios with varied diameters and downdraft speeds. The intent in this case was to generate a steady-state model (i.e. not varying with time) of a non-turbulent downburst wind field, for which time dependency could be later added empirically. The steady-state characteristics of most downburst simulators are often the most studied traits, and generally used for assessing the suitability of an apparatus for simulating such phenomena. While this is a rather simple view of downburst (the variability of downburst strength and size is discussed later in Section 2.3 of this paper), once simulated, the steady-state flow field presents a simple downburst non-turbulent wind speed 'template' that can be manipulated in a number of computationally efficient ways, which are discussed later, to provide simulated downburst wind speed time histories.

Simulations were performed in three spatial dimensions using a hybrid-tetrahedral mesh. Element size varied throughout the domains used. The smallest elements were located in the impingement zone, and progressively increased in size towards the limits of the domain. A velocity inlet with constant velocity and turbulence intensity was used as the jet outlet and the impingement surface was a 'no slip' wall. Pressure outlets with a zero gauge pressure border the simulation domain. Flow at the velocity inlet was initiated (the velocity inlet was 'turned on') at time $t = 0$. The simulations were run using an unsteady solver, which was computationally more stable than a steady solver, until the flow achieved a steady state.

Unfortunately, there was no suitable full-scale data available for proper validation of the model outputs. In order to validate the CFD simulations, the stationary jet tests (i.e. when the simulated downburst had no translational motion) of the Texas Tech University (TTU) downburst wind tunnel

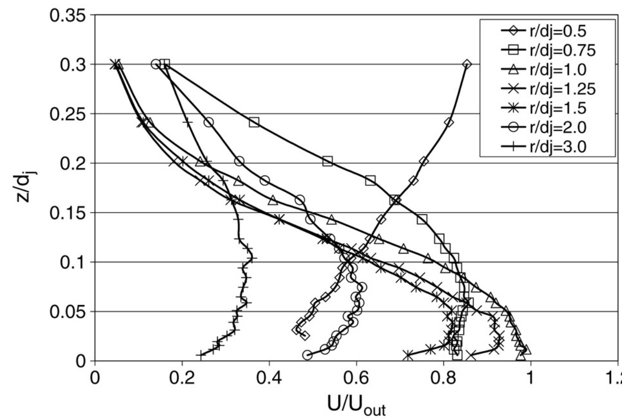

Fig. 1. Near-surface wind speed profiles created by the Texas Tech University downburst wind tunnel for a non-translating downburst [6].

were used as the test case. The wind tunnel utilises a circular jet with an outlet diameter (d_j) of 510 mm blowing with an outlet speed (U_{out}) approximately 11 m/s, impinging on a flat surface 860 mm away from the jet outlet. Full details of this apparatus are available in [4]. The wind field characteristics of this fan forced physical simulation are well documented, and such simulations are accepted as having downburst-like qualities. The philosophy of this approach is that if the CFD model could reproduce the scaled down downburst-like flow of the TTU stationary jet tests, then results would be valid when a full-scale flow is modelled. Fig. 1 shows the TTU stationary jet outflow wind speeds (U) as a ratio of U_{out} at a number of heights from the testing surface (z) at several radial distances in the horizontal plane from the centre of the jet (r), otherwise known as the 'centre of divergence'. Dimensions are shown as a ratio of d_j.

Several of the turbulence models available in FLUENT 6.0 were investigated for the TTU simulations. The models were run using an unsteady simulation from 0 to 4 s in 0.05 s increments, with 20 iterations per time step. Jet outlet speed (U_{out}) was 11 m/s with 4% turbulence and a hydraulic diameter equal to the jet diameter ($d_j = 510$ mm) [2].

The k–ε turbulence models are the most commonly used models, and as such are the best understood and validated of the CFD turbulence simulation methods. They are semi-empirical models based on turbulent kinetic energy (k) and its dissipation (ε).

When modelled in FLUENT 6.0, the exact equation describing ε is approximated. The contribution of fluctuating pressure to dissipation is also neglected, as it is considered negligible in most flows. However, in the stagnation region of an impinging jet, which occurs at small distances from the centre of divergence and close to the impingement surface (approximately $r/d_j < 0.5$ and $z/d_j < 0.5$), it plays an important role in the redistribution of turbulent kinetic energy. There are a number of known shortcomings for the k–ε model when applied to impinging jets. Jung-lei et al. [16] listed two of the major shortcomings as: