Engineering Structures 56 (2013) 779–793

Contents lists available at SciVerse ScienceDirect

Engineering Structures

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

Modeling of microburst outflows using impinging jet and cooling source approaches and their comparison

Yan Zhang, Hui Hu, Partha P. Sarkar *

Department of Aerospace Engineering, Iowa State University, Ames, IA 50011, United States

ARTICLE INFO

Article history:
Received 17 December 2012
Revised 30 April 2013
Accepted 6 June 2013
Available online 9 July 2013

Keywords:
Microburst
Impinging-jet models
Cooling-source model
Laboratory simulation
PIV measurement
Numerical simulation
Field data comparison

ABSTRACT

Microbursts have been simulated and studied using different physical and numerical modeling methods. In the present study, the steady impinging jet model was comprehensively studied by using a 2-foot-diameter (0.61 m) microburst simulator available in the Department of Aerospace Engineering at Iowa State University. Point measurements and Particle Image Velocimetry (PIV) results revealed a detailed picture of the overall flow and distribution of velocity and turbulence in the outflow of the steady impinging jet. Comparisons suggested that the average wind velocity profile of the steady impinging jet matched well with those derived from field data and previous research. FFT of the velocity time-history and instantaneous PIV results implied that the outflow consisted of low-frequency periodic shedding of vortices and the steady impinging jet model could be seen as an ensemble average of a series of simulated microburst events. Due to lack of time-dependent evolutionary information of the steady impinging jet model, a transient impinging jet model was studied to capture the transient features which were then compared with those of the cooling-source model by performing numerical simulations. Transient features of the transient impinging jet model and cooling source model showed several differences mainly related to the different formation and transportation process of the primary vortex. Ground surface pressure distributions were found to be different due to different forcing parameter of the two models. Comparison with the field data suggested that both models resembled the dynamic features of a real microburst outflow. However, results showed that the cooling source model could produce a reasonable instantaneous radial velocity profile at maximum wind condition, while the transient impinging jet model resulted in a large deviation. Finally, merits and demerits of each modeling methods were discussed.

1. Introduction

A microburst is defined as an intense downdraft impacting the ground and forming a damaging outflow with a diameter less than 4 km [1]. Since 1970s, a number of field projects had been conducted to study this natural phenomenon, mainly within the meteorological society [2–5]. Microbursts are dramatically different from the traditional straight-line winds and other wind hazards. They could produce significant wind shear and extreme winds near ground with a wind profile differing from the atmospheric boundary layer. Due to its transient nature, microbursts usually have very short lifespan and large vertical velocity components, which make it difficult to be detected and studied by Doppler radar. Therefore, different engineering models have been developed and used to produce microburst-like flow fields for a variety of research purposes.

Microburst-modeling methods to date can be classified into three categories, i.e. ring-vortex modeling, impinging jet modeling, and cooling source modeling. The first method has mainly focused on revealing the structure and evolution of flow patterns around the primary vortex generated in a microburst. Ivan [6] described a mathematical model of a downburst that resolves the stream function around a ring vortex. It was reported that this model produced results resembling some of the flow patterns, particularly the primary-vortex pattern noted in field data from the JAWS project. Schultz [7] constructed a multiple vortex-ring model by using time-invariant vortex ring filaments from potential flow theory. The velocity distribution around this simulated ring vortex matched the field data of the 1985 DFW microburst reasonably well. Vicroy [8] compared three theoretical models: linear, vortex-ring, and empirical. He found that latter two types provided more favorable results than the linear model.

The impinging jet model has been widely adopted due to its simplicity and ability to produce reasonable outflow-velocity profiles. As early as in 1987, by summarizing field data collected from a series of Colorado microbursts during the JAWS project, Hjelmfelt [4] pointed out that the outflow structures were found to have features resembling those of a laboratory-simulated wall jet. Subsequently, the impinging-jet model was utilized, both numerically

* Corresponding author. Tel.: +1 5152940719.
E-mail address: ppsarkar@iastate.edu (P.P. Sarkar).

0141-0296/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.engstruct.2013.06.003

and experimentally, by a number of researchers for microburst studies. Selvam and Holmes [9] used a two-dimensional k–ε model to simulate impingement of a steady jet of air on a ground plane. A reasonable agreement between numerical results and field data was achieved. Holmes [10] and Letchford and Illidge [11] performed experimental studies using a jet impinging on a wall to investigate topographic effects of a microburst outflow on velocity profiles. Holmes and Oliver [12] empirically combined wall-jet velocity and translational velocity and obtained a good representation of a travelling microburst which was well correlated with a 1983 Andrews AFB microburst. Wood et al. [13] experimentally and numerically studied impinging jets over various terrains. This study found agreement with respect to the established steady outflow at distances beyond 1.5 jet diameters from the impingement center. Choi [14] carried out both field and laboratory studies on a series of Singapore thunderstorms. Terrain sensitivity of microburst outflows was studied by comparing microburst observations at different heights and impinging jet experiments with different H/D ratios. The study produced similar trends, reflecting the impinging jet model's good capability for dealing with such problems. Chay et al. [15] conducted steady simulation and obtained good agreement with downburst wind-tunnel results. A non-turbulent analytical model was also used to study velocity–time history at a single point. Kim and Hangan [16] and Das et al. [17] performed both steady and transient two-dimensional CFD studies using an impinging jet model, producing reasonable radial–velocity profiles and good primary-vortex representation. Sengupta and Sarkar [18] carried out laboratory and 3-D numerical simulations using an impinging jet model. Both numerical and PIV results showed good agreements with full-scale data. To physically capture transient features, Mason et al. [19] deployed a pulsed-jet model to simulate transient microburst phenomenon. The formation and evolution of the primary, successive intermediate, and trailing edge vortices were visualized and recorded. Additionally, Nicholls et al. [20], Chay and Letchford [21], Letchford and Chay [22], and Sengupta et al. [23] performed impinging jet simulations to study the effects of microburst winds on low-rise structures. Generally, the impinging jet model is driven by a momentum-forcing source without any buoyancy effects. Although the steady-state models of impinging jet flow has been validated with field data by comparing wind velocity profiles, the transient features of an impinging jet flow compared to those of a real microburst still remain unknown.

An alternative approach using thermal cooling source was adopted by a few researchers, which puts more emphasis on the negative buoyancy and the dynamic development of the microburst. Experimentally, this method was accomplished by dropping denser fluids into less dense surroundings, which can be found in Lundgren et al. [24], Yao and Lundgren [25], and Alahyari and Longmire [26]. Nevertheless, the scale of physical modeling has remained very limited, making it almost impossible to study the wind loading effects on reasonably-scaled building models. Numerical simulations using cooling source approach involves a cooling source function, which was suggested by Anderson et al. [27]. The atmospheric full-cloud model was simplified to a space- and time-dependent cooling source function without considering the micro-physical process of a real microburst. This model was later used by Orf et al. [28] to study colliding microbursts, and by Orf and Anderson [29] to study travelling microbursts. Mason et al. [30] also investigated topographic effects on simulated downbursts using a sub-cloud model. Comparing the simulation results to their previous impinging jet modeling results, they suggested that little discrepancy was found with respect to the topographic effects despite use of two different modeling methods. Most recently, Vermeire et al. [31] compared the non-dimensional results using cooling source model and transient impinging jet model, and claimed that the impinging jet results deviated significantly from the cooling source results due to its unrealistic forcing parameters. This study used simplified impinging jet and cooling-source models and did not compare the simulation results with the transient characteristics of the field data. More comparisons with field data and data obtained from laboratory and numerical simulations are needed to compare and validate these two models apart from improving the models themselves.

Overall, due to the scarcity of field data and the complexity of this natural phenomenon, it is of critical importance to know which modeling method is the best for microburst study, particularly from an engineering point of view. Despite significant efforts by previous researchers, very little research has been found that compares the merits and demerits of different microburst models. In the present study, a steady impinging jet model was investigated by taking point and PIV measurements. Although the time-averaged characteristics of a microburst have been studied previously, its transient behavior and hence its dynamic features have not been fully explored. To complement the experimental study of a steady-impinging jet model, the transient behavior of an impinging jet model was studied numerically and compared with a simplified cooling source model. All results were compared to field data collected in the NIMROD and JAWS projects. Finally, the merits and demerits of these modeling methods were analyzed and concluded to provide references for use in future studies.

2. Experimental setup

The microburst was physically generated by a steady impinging jet flow simulator in the WiST (Wind Simulation and Testing) Laboratory at Iowa State University, shown in Fig. 1. The jet flow is produced constantly by a fan on the top and impinges on a wooden plate to form a steady wall-jet flow field. The diameter of the nozzle is about 0.6 m (2 feet). The distance between the nozzle exit and the plate representing the ground plane is adjustable from 1 to about 2.3 times the diameter (D) of the nozzle (0.75–7.5D in nature). The fan on the top of the simulator is driven by a step motor (RELIANCE ELECTRIC Duty-Master, Model number P2167403L). A honeycomb and several screens are placed at the exit of the nozzle to produce a uniform velocity across the exit and reduce the turbulence of the issuing jet. The axial velocity of the jet was measured at one nozzle diameter underneath the nozzle exit at different fan speeds, and the distribution across the jet was found to be sufficiently uniform, as shown in Fig. 2. The mean jet velocity under the nozzle exit was $V_{jet} \approx 6.9$ m/s.

Velocity measurements were first performed at different r/D locations (i.e. $r/D = 1, 1.5, 2, 2.5$) using three-component cobra-probe (TFI Pvt. Ltd.), where r is the radial distance from the center. Using this multi-hole probe, three components and the overall magnitude of the velocity vector can be measured at the same time. At each r/D location, measurements were taken at 38 points ranging from 0.25 in. to 7 in. above the ground plane. For each point, the data was collected at a frequency of 1250 Hz for 10 s. The measurement error was within ±0.5 m/s according to the specified accuracy of the cobra-probe. However, the probe could only resolve velocity information for the incoming flow within ±45° of the probe's axis. Therefore, for the shear layer of the wall jet flow, which is dominated by large-scale vortex structures, the accuracy of statistical results within the shear layers is significantly reduced due to reduced quantity of valid data gathered by the probe. PIV (Particle Image Velocimetry) technique was used (schematic is shown in Fig. 3) to capture whole-field information of the near-ground wall jet flow. The coordinate system indicating three velocity components was also shown in Fig. 3. The flow was seeded with 1–5 μm oil droplets and illumination was provided by a