Experimental and Numerical Investigations on the Flow Characteristics of Microburst-like Winds

Yan Zhang\(^1\), Partha P. Sarkar\(^2\) and Hui Hu\(^3\)\(^{(\star)}\)

Iowa state university, Ames, Iowa, 50010

In the past few decades, microbursts have been widely studied using different physical and numerical models. In this study, the most widely adopted model--steady impinging jet was investigated comprehensively by conducting point measurements and PIV measurements on the flow field. Numerical simulations were also performed to compare the transient features of the impinging jet model and the cooling source models. Experimental results suggest the steady impinging jet, which can be treated as a statistical average of a series of simulated microburst events, could provide a reasonable radial velocity profile at the critical location \(r_{\text{max}}\). Transient behaviours of the numerically simulated impinging jet model and cooling source model were different due to different driving mechanisms. Comparisons between the numerical results and the field data showed that the cooling source model could produce a reasonable instantaneous radial velocity profile at the critical location, while the transient impinging jet model resulted in a large deviation. However, the cooling source model constantly overpredicted the velocity after the primary vortex. Conclusions of pros and cons of each modeling methods will be given at the end of this paper.

Nomenclature

- \(H\) = distance from jet exit to the ground plate
- \(D(D_{\text{jet}})\) = jet diameter
- \(r\) = radial distance from the impingement center
- \(z\) = vertical distance above the ground
- \(L_0\) = length scale
- \(V_0\) = velocity scale
- \(T_0\) = time scale
- \(b\) = height of half maximum velocity
- \(T.K.E.\) = turbulent kinetic energy
- \(U_{\text{max}}\) = spatial and temporal maximum velocity magnitude
- \(r_{\text{max}}\) = radial distance from the impingement center to where the maximum velocity occurs

I. Introduction

In nature, a microburst is recognized as an intense downdraft of air, which could give rise to a damaging divergent outflow with a diameter less than 4 kilometers (Fujita, 1985). Although a microburst is a common wind hazard due to intensified atmospheric air circulation, its characteristics are dramatically distinct from the strong straight-line winds and other widely concerned wind hazards, i.e. tornadoes and gust fronts. First, unlike the atmospheric boundary-layer flow, microbursts can produce a jet-like outflow profile diverging from the center with its maximum velocity at less than 50 meters above ground level. The extreme winds and large velocity gradients near the ground would bring more significant negative effects on low-rise civil structures compared to normal boundary-layer winds. Second, different from tornado-like winds, microbursts produce almost no tangential velocity components and behaves like purely straight-line winds in outburst regions. However, in contrast to the straight-line winds, large vertical components of velocity could be observed in both the core regions and the leading edge of the outburst, which is extremely dangerous for the safety of aviating aircrafts as well as civil structures. As a result of these

---

\(^1\) Graduate Student, Department of Aerospace Engineering
\(^2\) Professor, Department of Aerospace Engineering
\(^3\) Associate Professor, Department of Aerospace Engineering, AIAA Associate Fellow, Email: huhui@iastate.edu

American Institute of Aeronautics and Astronautics
unique characteristics, additional research attentions should be paid to the microburst-induced winds to avoid possible losses in the area of aviation and civil structure construction.

Initiated by a meteorological study of the Eastern 66 crash at New York City's JFK airport in 1976, studies of the characteristics of the microburst and its outburst flow have been widely conducted by meteorologists and researchers in the fields of engineering as well. During 1970s and 1980s, two major projects named Northern Illinois Meteorological Research on Downburst (NIMROD, Chicago, IL) and Joint Airport Weather Studies (JAWS, Denver, CO) were carried out to gather data from the field. These field studies provided precious trustworthy data and depicted a vivid picture of microbursts occurred in the nature. These field researches were documented in Fujita (1979), Wilson et al. (1984), Hjemfelt (1987) and Hjemfelt (1988). Meanwhile a number of field studies at different places were also conducted. Atlas et al. (2003) investigated the physical origin of a microburst happened in Amazonia of South American based on the observation of a set of Doppler radar. Vasilloff and Howard (2008) deployed two types of radar system to capture a severe microburst occurred near Phoenix, Arizona. Nevertheless, the overall amount of the field data is scarce and the Doppler radar has its intrinsic limitations, such as low scanning frequency (3-5 min), low resolution near the surface, etc. All these limitations of the field study make the simulation and modeling essential in order to obtain a deep understanding of the microburst flow and its influences on civil structures.

Microburst modeling methods so far can be classified into three categories, namely ring-vortex modeling, impinging jet modeling and cooling-source modeling. The first modeling method mainly focused on revealing the structure and evolution of the flow patterns around the primary vortex generated in a microburst. Ivan (1985) described a mathematical model of a downburst which resolves the stream function around a ring vortex. It was reported that the mathematical model could resemble some of the flow patterns, particularly the primary vortex pattern, noted in field data from the JAWS project. Schultz (1990) constructed a multiple vortex ring model by using time-invariant vortex ring filaments from potential flow theory. The velocity distribution around the simulated ring vortex was matched reasonably well to the field data of the 1985 DFW microburst. Vicroy (1992) compared three theoretical models: linear, vortex-ring and empirical models and found that latter two model provided more favorable results than linear model.

Jet impingement modeling has been widely adopted due to its simplicity and ability of producing reasonable outflow velocity profiles. As early as in 1987, by summarizing the field data collected from a series of microbursts in Colorado during JAWS project, Hjemfelt (1987) pointed out that the outflow structures were found to resemble major features of the laboratory-simulated wall jet. After then the impinging jet model was utilized for microburst studies by a number of researchers both numerically and experimentally. Selvam and Holmes (1992) used a two-dimensional k-e model to simulate impingement of a steady jet of air on a ground plane. Although the simulation is coarse, a fairly acceptable agreement between the numerical results and field data was achieved. Holmes (1999) and Letchford and Illidge (1999) undertook experimental studies using a jet impinging on a wall to investigate the topographic effects on the velocity profiles of a microburst outflow. Holmes and Oliver (2000) combined a wall jet velocity and the translational velocity empirically and obtained a good representation of travelling microburst which shared well correlation with 1983 Andrews A.F.B microburst. Wood et al. (2001) studied the impinging jet over different terrains in both numerical and experimental methods. Good agreement was found in the established steady outflow beyond 1.5 jet diameters from the impingement center. Choi (2003) carried out both field and experimental studies on a series of thunderstorms in Singapore. Terrain sensitivity of a microburst outflow was studied by comparing the observation of microbursts with different heights and the impinging jet experiments with different H/D ratios. Similar trends were found, indicating a good capability of impinging jet model dealing with such problems. Chay et al (2005) conducted steady simulation and get good result agreement compared to downburst wind tunnel results. A non-turbulent analytical model was also used to study the velocity time history at a single point. Kim and Hangan (2007) and Das et al. (2010) performed both steady and transient two dimensional CFD studies on impinging-jet model. Reasonable radial velocity profiles and good representation of primary vortex were revealed. Sengupta and Sarkar (2008) carried out experimental and 3-D numerical investigations using impinging jet model. Both numerical and PIV results show good agreements with full-scale date. In order to capture the transient feature physically, Mason et al. (2005) deployed a pulsed-jet to simulate the transient microburst phenomenon. The formation and evolution of the primary vortex, successive intermediate and trailing edge vortex were visualized and recorded. Additionally, Nicholls et al. (1993), Chay and Letchford (2002), Letchford and Chay (2002), Sengupta and Sarkar (2008) performed impinging jet simulations to study the effects of microburst winds on different types of low-rise structures.

Generally, the impinging jet model is driven by a momentum forcing source and neglects the buoyancy effects, which remains arguable so far whether or not it could truly reproduce the natural phenomenon. Hence an alternative approach using thermal cooling source was adopted by many other researchers. Experimentally, it was usually
simplified by dropping denser fluids into less dense surrounding. Good representations of primary vortex structures and reasonable velocity profiles could be found in Lundgren et al. (1992), Yao and Lundgren (1996) and Alahyari and Longmire (1995). Nevertheless, the scale of physical modeling is very limited which makes it almost impossible to perform flow-structure integration studies. Numerical investigations using cooling source models were initiated by meteorological simulations. Srivastava (1985) examined the properties of a microburst downdraft in a one-dimensional microphysical model. He found the microburst intensity would increase with increases of some environmental parameters, such as temperature lapse rate, precipitation concentration and humidity. Proctor (1988, 1989) performed two-dimensional axisymmetric simulation on Terminal Area Simulation System (TASS). Primary driving force of a microburst was found to be evaporation cooling. The formation and expansion of a ring vortex and surface friction effects were also investigated. Similar full-cloud meteorological model can also be found in Straka and Anderson (1993), Fu and Guo (2006) etc. For the purpose of engineering studies, the full-cloud model was simplified to a space and time dependent cooling source model by Anderson (1992). A 3-D model system for simulating thunderstorm microburst outflows. This model was later used by Orf et al. (1996) to study the colliding microbursts and Orf and Anderson (1999) to study travelling microburst. Mason et al. (2010) also investigated the topographic effects on simulated downburst using this sub-cloud model. Comparing the simulating results to their previous impinging jet modeling results, it was suggested that little discrepancies was found on the topographic effects despite two different modeling methods were used.

Overall, due to the scarce 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 engineering point of view. Despite of great efforts contributed by previous researchers, very little research was found to compare the pros and cons of different microburst models. In current study, the steady impinging jet model, which was the most widely used, was studied experimentally using both point measurements and PIV techniques. Numerical impinging jet model and cooling source model were then investigated to compare the different transient behaviors between these two modeling methods. In order to make more accurate assessments, all results were compared to the field data collected in NIMROD and JAWS projects. Finally, the advantages and disadvantages of these modeling methods will be analyzed and concluded to provide useful references for the choice of future studies.

II. Model Description

A. Experimental Setup for Steady Impinging Jet Model

The steady impinging jet flow field was physically modeled by a microburst simulator in WiST lab of Iowa state university, shown in Figure 1. The jet flow is produced constantly and impinges on a wooden ground plate to form a steady wall jet flow field. The diameter of the nozzle is about 560 mm (22 inches). The distance between the nozzle exit and the ground plane could be adjusted from 0 to about 2.3 diameters of the nozzle. The fan on the top of the simulator is driven by a step motor (RELIANCE ELECTRIC Duty-Master, Model number P2167403L), which has a RPM of 1765 and working efficiency of 90.2%. A honeycomb and several screens were implemented to the exit of the nozzle to uniform the velocity across the exit and reduce the turbulence of the issuing jet. The velocity distributions across the jet exit at different distances were shown in Figure 2. In addition, the motor frequency can be

Figure 1. Microburst simulator in WiST lab of Iowa State University

Figure 2. Axial velocity distribution underneath the jet exit.
changed smoothly up to 60 Hz. An excellent linearity was found between the jet flow velocity and the motor frequency.

Velocity measurements were first performed under H/D=1 and H/D=2 situations using three component cobra-probe (Turbulent Flow Instrumentation Pty Ltd) at different r/D locations (i.e. r/D=1, 1.5, 2, 2.5). The probe has the ability to measure three components of the velocity vector at the same time. At each r/D locations, the measurements were taken at 38 points from 0.25 inches to 7 inches above the ground plate. For each point, the data was collected at a frequency of 1250 Hz for 10 seconds. The measurement error was within ±0.5 m/s according to the accuracy of the cobra probe. However, the probe could only resolve the velocity information for the oncoming flow within ±45 degrees of the probe x-axis. Therefore, for the free shear layer of the wall jet flow, which was dominated by large scale vortex structures, the quantity of the valid data gathered by the probe decreases dramatically. Therefore, the accuracy of the statistical results within free shear layers is considered to be reduced due to the lack of valid data.

PIV technique was then used to capture the whole-field information of the near ground wall jet flow. The schematic of the PIV system is shown in Figure 3. The flow was seeded with 1-5 μm oil droplets. Illumination was provided by a double-pulsed Nd:YAG laser (NewWave Gemini 200) adjusted on the second harmonic and emitting two laser pulses of 200 mJ at a wavelength of 532 nm with a repetition rate of 10 Hz. The laser beam was shaped to a laser sheet (thickness ~1 mm) by using a set of mirrors, spherical and cylindrical lenses. A high resolution (1365×1024 pixels, Cooke Corp.) charge-coupled device (CCD) camera was used for PIV image acquisition with its axis perpendicular to the laser sheet. The CCD camera and the double-pulsed Nd:YAG lasers were connected to a workstation via a digital delay generator, which controlled the timing of the laser illumination and the image acquisition.

The CCD camera was focused to a measurement window of 207×152 mm size and totally 14 windows were investigated to cover all the interested areas of the outflow region. The layout of the investigation windows are illustrated in Figure 4. In order to ensure the results from different windows match each other reasonably well, 30% overlaps were taken between each window and its vertical neighbor. Instantaneous PIV velocity vectors were obtained by a frame to frame cross-correlation technique involving successive frames of patterns of particle images in an interrogation window with 32×32 pixels. An effective overlap of 50% was employed to satisfy the Nyquist criterion. After the instantaneous velocity vectors were determined, time-averaged quantities such as mean velocity, turbulent velocity fluctuations and normalized turbulent kinetic energy, and Reynolds stress distributions were obtained from a cinema sequence of 360 frames of instantaneous velocity fields for each case. The measurement uncertainty level for the velocity vectors is estimated to be within 2.0%, and that of the turbulent velocity fluctuations and turbulent kinetic energy are about 5.0%. The uncertainty level of the spanwise vorticity data is expected to be within 10.0%. In order to provide a validation test for the following numerical studies, a transient start-up process of the microburst simulator was investigated to capture the main dynamic features of a transient impinging jet. For this case, the measurements in each window were accurately synchronized by starting the fan 5 seconds after triggering the PIV system. 166 frames in a time sequence were taken for each of the investigation window. Theoretically speaking, in order to get statistical information of the transient process (such as turbulent kinetic energy, Reynolds stress), the experiment should be repeated for hundreds of the times for all 14 windows based on a phase-locked synchronization. However, in order to avoid the tremendous workloads, this study only took one instantaneous velocity profile at the maximum velocity time and location in a single simulated microburst event. The information is though sufficient for the purpose of validating the numerical model of this study.
B. Numerical Simulations for Transient Impinging Jet Model and Cooling Source Model

1) Computational setup

An axisymmetric unsteady RANS model was applied to this study using commercially available software FLUENT 12.1 (ANSYS Inc.). Although LES is well known to have the ability to resolve the large scale turbulent structures and simulate the time-dependent turbulent flows, the application of LES requires very fine mesh and sufficiently small time step. Therefore, considering the large geometric scale of the computational domain, LES could be extremely expensive, particularly for this high-Reynolds-number problem. The objective of this numerical simulation, however, is to investigate the differences of macro scale flow features between two modeling methods and compare them with the field data. Therefore, the unsteady RANS model is overall an economic and effective tool for this study. In the unsteady RANS simulation, the ensemble-averaged velocities, denoted by $\langle u \rangle$, are still a function of time. Thus the Reynolds decomposition of velocity can be expressed as $u = \langle u \rangle + u' = u + u'' + u'$, where $u$ is the time-averaged velocity and $u''$ is the resolved unsteadiness of the mean flow. Therefore, although the turbulent component was fully modeled, the unsteady features of the ensemble-averaged flow field are still resolved, which makes URANS an effective tool particularly for solving macro flow problems.

In current study, standard $k-\varepsilon$ viscosity models were used as closure equations. The standard $k-\varepsilon$ model is widely used due to its simplicity, robustness, and reasonable accuracy for a wide range of turbulent flows. The default model parameters used in the simulation were suggested by Launder and Spalding (1972): $C_{1k} = 1.44$, $C_{2k} = 1.92$, $C_{\mu} = 0.09$, $\sigma_k = 1.0$, $\sigma_{\varepsilon} = 1.3$. Second order upwind scheme was used for solving the continuity and momentum equations. T.K.E. and turbulent dissipation rate were both solved using Quadratic Upstream Interpolation for Convective Kinematics (QUICK) scheme. SIMPLE scheme was used for pressure-velocity coupling. For the transient formulation, second order implicit scheme was adopted.

The geometry and boundary conditions are shown in Figure 5. The jet diameter D was set to be 2,500m to simulate the realistic natural phenomenon and H/D ratio was set to be 1. For steady impinging jet, a velocity inlet was used combined with incompressible flow condition. For cooling source model, a specific cooling function covering the inlet region was incorporated by adding a source term into the energy function. The cooling function will be discussed in details later. A pressure inlet and the compressible flow condition were used in order to resolve a density change induced by the cooling function.

All simulations in this study were solved on a structured grid. At the wall boundary, $y+$ was confined within 5 for the first row of cells. The mesh was gradually stretched as it moved away from the ground boundary. A mesh independent study was carried out at the beginning of the simulation. As shown in Figure 6, the velocity profiles (at $r/D=1$ location; at the time step of 470 seconds) tend to converge to the same line as the number of cells increases. Therefore, 1-million-cell grid was chosen here and it is safe to believe that the results are independent of mesh conditions.

2) Cooling function

Cooling source model was simulated by adding a spatial and temporal varying cooling source to the computational domain as shown in Figure 6. This sub-cloud cooling model was suggested by Anderson (1992). This effect is achieved by adding a source term to the energy equation, which is described by:

Figure 5. Computational domain

Figure 6. Mesh independent study
where \( g(t) \) is a time-dependent coefficient which increases from 0 to maximum in the first 120 second and then remain constant from 120 s to 420 s. Finally \( g(t) \) will gradually decrease to 0 between 420 s to 540 s. Maximum value of \( g(t) \) was given -0.1 K/s which is larger than that described in Anderson (1992) in order to get more significant cooling effects. \( R \) is radius of the cooling source determined by position of the geometric center \( x-x_0, \ y-y_0, \) and the distances from the center \( h_x, \ h_y. \) Figure 7 illustrates the entire life cycle of a simulated microburst event, visualized by the evolution of the temperature field of the cooling source model.

![Figure 7. Evolution of the temperature field.](image)

**III. Experimental Results of Steady Impinging Jet model**

**A. Results of Point Measurements**

Figure 8 shows the normalized velocity profiles of the steady impinging jet flow field at different radial locations \( r/D= 1, 1.5, 2, \) and \( 2.5 \) under three \( H/D \) ratios, i.e. \( H/D=1, 1.5, \) and \( 2, \) where \( r/D \) is radial distance from the center normalized by the jet diameter and \( H/D \) is the height to diameter ratio. The velocity was normalized by the mean exit velocity of the jet and the vertical height was normalized by the diameter of the jet. From Figure 8, it could be concluded that the profiles of normalized velocities are quite similar and almost independent of the \( H/D \) except for the discrepancies of the maximum velocity. It is evident that with an increase of \( H/D \) ratio, the maximum velocity of the wall jet decreases at \( r/D=1, 1.5, \) and \( 2. \) However, at the place sufficiently far from the impingement center, for example \( r/D=2, \) all profiles become extremely similar due to the energy dissipation of the wall jet. Hence, it is believed that the normalized flow field far from the impingement center is independent of \( H/D \) ratio.
Fig. 8 Normalized velocity profiles of the steady impinging jet.

Figure 9. Normalized velocity component profiles of the steady impinging jet.
Figure 10. Turbulent intensity profiles of the steady impinging jet.

Figure 11. Turbulent intensity component profiles of the steady impinging jet.
All three components of the velocity profiles under H/D=1 was compared in Figure 9. In the comparison, u component denotes the velocity magnitude in the radial direction of the impinging jet flow; v component denotes the velocity magnitude in the tangential direction of the main flow; w component denotes the velocity magnitude in the vertical direction. In the well-developed steady impinging jet flow, it is clear that u component dominates the flow and v, w components are relatively negligible. At r/D=1, a considerable increase of v and w magnitude could be seen from the ground plate to the flow interface. This phenomenon will be later confirmed again in the PIV measurement results. For other r/D, this trend is unobvious due to the growth and mixing of the wall jet. Since the results of H/D=1.5 and H/D=2 are similar, they are not shown here for simplicity.

Figure 10 shows the overall turbulence intensity at different r/D. It is a common trend that the turbulent intensity remains small within a certain vertical distance, roughly z/D=0.1 and then dramatically increases to a maximum value. This implies that the turbulence level within the outflow is relatively small. However, the free shear layer produces a strong turbulent mixing at the interface between wall jet and the ambient air. Comparing the curves of four radial distances, it could also be found that the turbulent intensity within the outflow also notably increases with an increase of r/D. The slope of the curve becomes milder at a larger r/D, indicating a fully turbulent outflow has been developed.

Figure 11 shows turbulent intensity of three components of velocity and the overall fluctuations of the velocity magnitude. It is revealed clearly that the w component turbulent intensity behaves different from other components. The flow sufficiently near the ground shows little vertical fluctuation, whereas it grows rapidly to a dominant value as z/D ratio increases. However, for large r/D cases, the peak value of w component drops from 50% to less than 40% and eventually follows the same trends of other components.

The significant fluctuation in vertical direction may be closely related to the shedding vortices within the free shear layer. This fluctuation decreases substantially as the vortices dissipate towards larger r/D locations. To confirm this, fast Fourier transformations of the velocity fluctuations were performed at z/D=0.20, which is within the free shear layer according to the previous plots, at r/D=1 and r/D=2. It can be seen in Figure 12 that instead of a complete randomness, a low frequency near 17 Hz dominated the spectrum at r/D=1. This indicates a periodical motion of large scale structure within the shear layer. As the flow moves to r/D=2, the magnitude of this frequency decreased as the large scale structure breaks down to smaller ones. This will be further verified in the discussion of PIV results.

B. PIV results

The ensemble-averaged information for H/D=1 case is shown in Figure 13. The U and V velocity contours were normalized by the averaged jet exit velocity. In the U component contour, a large velocity region, with the maximum magnitude of U/V_{jet}=1.1, can be found near r/D=1, which corresponds well with the point measurement results. In W velocity contour, a positive velocity region could be found around r/D=1, which also agrees well with points measurement results in Figure 9. This indirectly confirms the CFD results of Kim and Hangan (2007) about the existence of the secondary vortex caused by the surface friction. Due to the secondary vortex, the flow nearby is not only accelerated within the narrow channel, but also moved in vertical direction due to the flow separation.

Turbulence kinetic energy and Reynolds shear stress were normalized by the squared jet velocity. For H/D=1 case, it can be seen that the outflow turbulence were mainly generated in two regions: interface between the jet flow and ambient air; the wall boundary. Turbulent flow from these two parts then mixed to form a large turbulent region
in the wall jet flow. It is also clear that the region between \(r/D=1.5\) and \(r/D=2\) contains more turbulent energy than other regions. In Reynolds shear stress contour, turbulence from these two sources can be easily distinguished. Negative regions were caused by a negative velocity gradient along the vertical directions and therefore represent the turbulent flow formed in the free shear flow. In contrast, the red region shows the turbulent flow developed in the wall jet boundary layer.

For a visual comparison, Figure 14 shows the corresponding ensemble-averaged contours for \(H/D=2\) case. In the U and V velocity contours, it is revealed that the large U region and positive W region both shrunk considerably due to the weaker impact force. The maximum velocity was also found more close to the ground plate. For the turbulence kinetic energy in the wall jet flow, the contribution from the free shear layer increased while the other part from the wall boundary decreased substantially. From both the T.K.E. and Reynolds shear stress contours, it can be concluded that the overall turbulence level was reduced due to the increased distance between the jet exit and the ground.

As previously assumed, the large turbulent mixing caused by the free shear layer was actually dominated by periodically shed vortices. This effect could be visually verified in the instantaneous velocity fields of PIV results. Figure 14 presents a single snapshot of the instantaneous flow in the investigated window 2B. Two vortices could be
clearly visualized at the flow interface. These vortices generated by the Kelvin-Helmholtz instability confirm the periodical pattern of velocity fluctuations found in the FFT analysis in the previous part.

C. Summary

The steady impinging jet was studied comprehensively by conducting experiments using the microburst simulator. Generally, H/D ratios did not change the wall jet characteristics too much except that larger velocity regions and larger wall boundary turbulence could be expected with a smaller H/D ratio. Periodically shed vortices were found in both point and PIV measurements. This indicates that the steady impinging jet model could be treated as a combination of a series of simulated microburst events with a time period towards infinity.

The averaged velocity profiles at the maximum velocity locations for both H/D=1 and H/D=2 cases were compared with the field data and the previous numerical and experimental results in Figure 15. It should be noted that although microbursts are transient in nature (around 10 min), the field data collected by Doppler radar is normally taken for a 1-min average. It can be concluded from the plots that a very good agreements were reached between current measurements and the field data, particularly for H/D=2 case. Therefore, even though the steady impinging jet fails to provide adequate dynamic features of the natural microbursts, the statistical similarity of the velocity profiles ensures that it could still be used as an input of structure model tests or other related research areas.

IV. Numerical Results of Transient Impinging Jet model and Cooling Source Model

A. Validation and Scaling

In order to validate the numerical results, the vertical profiles of radial velocities directly across the primary vortex center at the time of maximum velocity were compared between the numerical impinging jet model and a transient start-up process of the physical impinging jet. Although the instantaneous profiles may vary from case to case, this comparison could provide rough evidence that the numerical simulation can produce reasonably good results.

In order to compare the transient features of different numerical modeling results, the flow field variables of two models should be normalized to the common critical parameters. Since the underlying forcing mechanism is intrinsically different in impinging jet model and cooling source model, it is not convincible to directly match the
results using the computational time and length scales.

As is widely known, the most prominent feature of a microburst is the primary vortex ring which could produce the locally extreme wind. Therefore, the time scale $T_0$ here is given to be the computational time at which the maximum velocity was found in each modeling result. Velocity scale is consequently the maximum velocity itself, denoted by $V_0$. Then the length scale could be calculated by $L_0 = V_0 T_0$. And the corresponding Reynolds number is

$$\text{Re} = \frac{V_0 L_0}{v}$$

In current numerical cases, scaling parameters are given in Table 1. The Reynolds numbers with the characteristic length of $L_0$ the two methods are on the same order.

<table>
<thead>
<tr>
<th></th>
<th>$V_0$</th>
<th>$T_0$</th>
<th>$L_0$</th>
<th>$\text{Re}$</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Impinging Jet</strong></td>
<td>$V_{01} = 45.9 , \text{m/s}$</td>
<td>$T_{01} = 470 , \text{s}$</td>
<td>$L_{01} = 21573 , \text{m}$</td>
<td>$6.55 \times 10^{10}$</td>
</tr>
<tr>
<td><strong>Cooling Source</strong></td>
<td>$V_{02} = 67.5 , \text{m/s}$</td>
<td>$T_{02} = 260 , \text{s}$</td>
<td>$L_{02} = 17550 , \text{m}$</td>
<td>$7.84 \times 10^{10}$</td>
</tr>
</tbody>
</table>

**B. Results and discussion**

Figure 17 shows the radial velocity contours of two modeling method at different scaled time. The velocity was normalized by the maximum value of the radial velocity during the computational time. The four contours, though have different time scale, were organized by matching the location of the first vortex core (namely 1. before touching the ground, 2. at $1r_{\text{max}}/D$, 3. at $1.5r_{\text{max}}/D$, and 4. at $2r_{\text{max}}/D$). As shown in A1, the impinging jet produced a pair of negative and positive velocity contour, i.e. a primary vortex, before the flow touched the ground. As the primary vortex touched the ground at $T=1.007T_0$, the outflow was channeled between the primary vortex and the secondary vortex caused by ground friction and hence accelerated. These vortices can be clearly seen in Figure 19. The temporally and spatially maximum velocity was found at this time accompanied with the primary vortex. As the vortex traveled and decayed radially, new vortices were found to be continuously formed at the free shear layer between the jet flow and the ambient air. These following vortices then caused a series of large velocity regions that were comparable with the maximum velocity, as shown in A3, A4.

Due to the absence of the strong and continuous forcing source at the inlet, the radial velocity contours of the cooling source model show great differences from the impinging jet model. As shown in B1, no strong vortex was found before the flow touched the ground. At $T=1.007T_0$, the maximum velocity was formed below the primary vortex. However, due to the different driving force and the gravitational effects, the velocity contour was apparently more compressed towards the ground than that of the impinging jet model. In addition, this gravity-current-like outflow induced no flow separation, i.e. secondary vortex at this time, as shown in Figure 19. As the primary vortex traveled radially, the negative velocity inside the primary vortex was found to increase gradually (B3, B4), while for the impinging jet model, negative velocity almost remained constant (A3, A4). Meanwhile, no following velocity pairs were found in the cooling source model. The primary vortex accompanied with the large velocity region decayed with time and eventually died out after the strength of the cooling source decreased to 0.

The normalized axial velocity contours of two modeling method at different scaled time are presented in Figure 18. While the impinging jet triggered strong vertical interaction at the interface of the jet and ambient air, the descending cooling air did not influence the ambient air significantly before it touched the ground and generate the primary vortex (A1, B1). As the vortex traveled, the vertical component gradually increased as the radial velocity decreased in both cases. Moreover, the axial velocity pair of cooling source was seen to be relatively more dominant, though it is smaller in size and lower in altitude.

Another phenomenon observed from Figure 19 is that the high velocity region under the primary vortex in the impinging jet model seems to be more stratified compared with that of the cooling source model. Additionally, this high velocity regions in cooling source model was right on the leading edge of the outflow, while that in the impinging jet model is embedded way behind the outflow front. This distinct features could be possibly contributed to their intrinsically different underlying physics. For the impinging jet model, the instability and the primary vortex determines the velocity inside the wall jet flow. On the contrary, the outflow in cooling source model moves much
milder and the generation of primary vortex was more probably related to the convective motion of the density-driven flow.

Figure 20 compares the velocity profiles in the radial direction at different scaled time at the height of the maximum velocity, for both impinging jet model and cooling source model. The velocities were normalized by the maximum velocity at \(T=1.00T_0\) and \(T=1.00T_0\) for each case and the radial distance was normalized by the distance from the center to the locations where maximum velocities occurred at that time step. It can be seen in part A that for impinging jet model, the second peak of velocity was gradually formed as the primary vortex traveled downstream, which is a result of the periodically shed vortices at the free shear layer. In the meantime, the velocity

**Figure 17. Contour of Normalized Radial Velocity.**

\(A1-A4\) impinging jet model; \(B1-B4\) cooling source model
between two peaks dropped significantly, which indicates that the primary vortex and the following vortex were separated as the flow decayed radially. Nevertheless, for the cooling source model, a plateau of constant velocity was steadily developed directly behind the velocity peak. No following peaks were observed. These differences again provide reasonable evidence that in cooling source model, the primary vortex is a “self-triggered” structure, whereas it is greatly influenced by the instability at the flow interface in the impinging jet model.

The descending trajectories of the primary vortex core of two models are shown in Figure 21. Due to the strong instability at the interface of the wall jet flow, the primary vortex in impinging jet model descending from high altitude position was found oscillating in axial direction as it expanded radially. However, the primary vortex in

![Normalized Axial Velocity Contours](image)

**Figure 18. Contour of Normalized Axial Velocity**

(A1-A4 impinging jet model; B1-B4 cooling source model)
The cooling source model appeared rather stable as it moved outwards. Because of the gravity force, the vortex was also found to be much closer to the ground. It should be noted that the dashed line for the cooling source model means there is actually no primary vortex formed before it touched ground, though a slightly reversed flow could be seen before $T=T_{02}$. (Shown in Figure 18 B1)

The trajectories of primary vortex cores in radial direction were compared with the field data gathered from JAWS project (Hjelmfelt, 1988) in Figure 22. In order to make this comparison valid, the field data here was re-normalized to ensure that $r/r_{max}=1$ corresponds to the normalized time $T/T_{0}=1$. It should be noted that the field data does not represent the actual vortex core movement, but the expansion of the gust front of the microburst. Hence, it is assumed here the vortex expansion is equivalent or similar to the gust front expansion. From the figure, it is clear that both the impinging jet model and cooling source model resulted in a linearly expanded primary vortex, which is similar to the real microburst events in nature. The slope of each curve represents the relative expansion speed corresponding to initial conditions of each real or simulated microburst event, which for this study is not the same (the cooling source model in this study generated a larger velocity than impinging jet model).

C. Summary

The comparisons of velocity contours and vortex trajectories between the impinging jet model and cooling source model revealed several different characteristics induced by different intrinsic physics of these two models.
While the flow patterns in impinging jet model were dominated by the instability in the free shear layer, the cooling source model caused a relatively smooth outflow which resembles the features of a gravity current. After the primary vortex, the following vortices in the impinging jet model were found to contribute several locally extreme winds which were comparable to the maximum one. However, no following vortices were found in the cooling source model, while a constant velocity plateau was gradually formed behind the primary vortex instead.

The transient expansion of the primary vortex in these two models, though with a different expansion speed, resembles the linear characteristic of the natural events. In order to further compare the modeling results, a time series of the velocity profiles in radial direction were compared in Figure 23. The field data is the time history of a single microburst events occurred during the JAWS project (Hjelmfelt, 1988). Generally, these two models give reasonably good estimations of the radial velocity distributions of a real field event. However, it can be seen the cooling source model constantly over-predict the relative velocity magnitude after the primary vortex. In other words, trailing-edge flow after the primary vortex in a real microburst appears to be weaker than the simulated microburst in the cooling source model.

![Figure 23. Comparison of the Time Series of Velocity Profiles in Radial Direction (Left: impinging jet model; right: cooling source model)](image)

The radial velocity profiles at the vicinity of the maximum velocity time and location were again compared to the field data and previous studies. As mentioned previously, the field data was collected from a transient microburst event, but usually taken as 1-min average by Doppler radar. Hence, the field data, though averaged, will still
resemble the transient features of the real microburst event. For current simulations, it is actually difficult to
determine the exact time and location at which the maximum velocity occurs. Therefore, the vicinity of the
“visually” determined maximum time and location were used to make the comparison. Figure 24 shows the radial
velocity profiles at the maximum velocity time and the vicinity of the maximum velocity location for both two
models. It is evident that the transient impinging jet deviated from the steady jet and the field data considerably,
while the cooling source resulted in a similar instantaneous velocity profile compared with the field data. Similar
results could be found in Figure 24, where the velocity profiles were compared by taking data from the vicinity of
the maximum velocity time.

![Figure 25. Comparison of radial velocity profiles at the vicinity of the maximum velocity time](image)

**V. Concluding Remarks**

The stationary microburst flow field was modeled physically and numerically using different modeling methods.
Experimentally, steady impinging jet flow field was simulated by a microburst simulator, which produced jet flow
impinging on the ground plate. Point measurements of velocity field were carried out at different r/D positions under
different H/D ratios. PIV measurements were performed in order to get the whole field information of the steady
impinging jet. Numerical simulations were performed using different modeling methods. The transient impinging jet
model used constant velocity inlet, while cooling source model incorporated a cooling source term to the energy
equation to generate the negative buoyant forcing.

The results of the physical modeling show that the H/D ratio actually had trivial influences on the velocity
profiles of the outflow, particularly at the position sufficiently far from the impingement center. The turbulent level
within the outflow increased with r/D increased, indicating turbulent mixing intensifies as r/D increases. Periodically
shed vortices were found in both point and PIV measurements. This indicates that the steady imping jet model could
be treated as a statistical average of a series of simulated microburst events with a time period towards infinity. The
averaged velocity profiles at the maximum velocity locations were compared with the field data and the previous
numerical and experimental results. Very good agreements were found between current measurements and the field
data. It has been concluded that even though the steady impinging jet fails to provide adequate dynamic features of
the natural microbursts, the statistical similarity of the velocity profiles ensures that it could still be used as an input
of structure model tests or other related research areas.

The velocity fields of the numerical simulations showed substantial differences between two modeling methods.
While the flow patterns in impinging jet model were dominated by the instability in the free shear layer, the cooling
source model caused a relatively smooth outflow which resembles the features of a gravity current. Due to the strong
continuous forcing source of the impinging jet model, the primary vortex was found to be formed at the interface
between the jet flow and the ambient air, immediately after the flow was initiated. As the primary vortex touched the
ground and expanded radially, following vortices generated due to the strong instability in the free shear flow.
However, for the cooling source model, the primary vortex was found to be formed only after the cooled air descending to the ground and no following vortices were found compared to the impinging jet model. The primary vortex then traveled downstream leaving a velocity plateau behind. The transient expansions of the primary vortex in these two models, though with a different expansion speed, both resembled the linear characteristic of the natural events.

Comparisons between transient velocity profiles of the two modeling methods and the field data were performed. Results indicated that the cooling source model might lead to an over-prediction of the velocity magnitude after the primary vortex. However, in terms of reproducing the radial velocity profiles at the maximum velocity location and time, the impinging jet model resulted in a considerable deviation from the field data, while the cooling source model could provide a more reasonable agreement.

The pros and cons of each modeling methods are summarized as follows:
1) The steady impinging jet model could provide an averaged flow field with a reasonable radial velocity profile at the critical location (maximum velocity location), but is lack of time dependent information.
2) The transient impinging jet model fails to provide a reasonable radial velocity profile at the critical location, but it could give a good simulation of the transient behavior of the radial velocity distribution.
3) The cooling source model could provide a good representation of the instantaneous radial velocity profile at the critical location and is also capable to give a reasonable representation of the transient expansion of the primary vortex, but it over predicts the velocity after the primary vortex.

In conclusion, as the field data is rather scarce, the truth of the real microbursts in nature is far from fully understood. Therefore, from the engineering point of view, the choice of the microburst modeling methods should not be fixed. Future studies related to the microburst modeling could take advantage of the simplicity and accuracy and avoid the fatal drawbacks of each modeling methods.

Acknowledgments

The project is funded by National Science Foundation under award number of CMMI-1000198 with Dr. Kishor Mehta as the program manager. The authors would also like to thank Mr. Bill Richard and Nick Krauel of Iowa State University for their generous help in building test models and conducting experiments.

References


American Institute of Aeronautics and Astronautics


