Advances in wind tunnel simulation, techniques and tools for assessing extreme-wind hazard to structures

Partha P. Sarkar
Professor of Aerospace Engineering, Iowa State University, Ames, IA, USA, ppsarkar@iastate.edu

Abstract

Extreme weather phenomena such as hurricanes, tornadoes, downbursts, and gust fronts produce 36% of combined insured losses from all natural hazards annually in the United States and cause similar losses in many other countries of the world. This paper primarily focuses on the advances in wind tunnel simulation, modeling and data analysis techniques and tools used for assessing wind hazard to structures in straight-line and transient winds. At present, civil structures are designed to resist only straight-line winds associated with a neutrally stable atmospheric boundary layer whereas gust fronts, tornadoes, downbursts and even hurricanes produce transient winds that have distinctly different characteristics from the design wind specified in building codes. Tornadoes are translating vortices with significant tangential and vertical velocity components in the core region. Microbursts are intense downbursts characterized by a strong localized down-flow and an outburst of strong winds near the surface. Thus, the velocity fields in both of these wind phenomena are three dimensional and the structural loading effects they produce are transient in nature. Tornado of even moderate intensity can produce up to three times the design wind loads for structures located in regions outside the hurricane or coastal zone. Advances in wind simulation techniques and understanding of winds in gust front, tornado and microburst near the ground surface and their wind loading effects on buildings and other structures are discussed. Effects of parameters influencing wind loads such as characteristics of the storm, terrain and topography surrounding the structure, geometry and shape of the structure, interference from surrounding structures, and internal pressures are mentioned. Tools like System Identification, CFD and Finite Element Modeling are used in combination with physical simulation in wind tunnels to study the interaction of structures with transient wind to predict structural damage.

Keywords: Wind tunnels, Tornado, Microburst, Gusts, Wind hazard, Transient effects

Introduction

Extreme weather phenomena produce 36% of combined insured losses from all natural hazards annually in the United States and cause similar losses in many other countries of the world. Low-speed boundary-layer wind tunnels generating straight-line winds have played an integral role in the design of wind-sensitive structures for decades. Capable of simulating the lower portion of the earth’s atmospheric boundary layer, these tunnels have enabled the safe design of long-span bridges, low-rise and tall buildings, towers, and a host of other structures. Velocity fields resulting from extreme wind events in non-hurricane regions such as gust fronts, thunderstorms, microbursts and tornados, however, are far from conventional straight-line wind of boundary-layer type. These events produce transient winds that have distinctly different characteristics from the design wind specified in building codes. Tornadoes are translating vortices with significant tangential and vertical velocity components in the core region. Microbursts are intense downbursts characterized by a strong localized down-flow and an outburst of strong winds near the surface. Thus, the velocity fields in both of these wind...
phenomena are three dimensional and the structural loading effects they produce are transient in nature. This paper primarily focuses on the advances in wind tunnel simulation, modeling and data analysis techniques and tools used for assessing wind hazard to structures in straight-line and transient winds. The advances in wind simulation techniques and some unique features of non-synoptic wind such as gust front, tornado and microburst near the ground surface and their wind loading effects on buildings and other structures in comparison with those of straight-line wind are discussed. In addition to known parameters that influence wind loads in straight-line wind such as terrain and topography, geometry and shape of the structure, interference from surrounding structures and internal pressures, the characteristics of the storm also influence the wind loads in transient wind. Tools like System Identification, CFD and Finite Element Modeling are used in combination with physical simulation to study the interaction of structures with transient wind to predict its structural damage as shown here in this paper.

Wind Simulation and Measurement

**Straight-Line Wind Simulation**

Wind tunnel simulation of the earth’s atmospheric boundary layer is a well-established practice. Numerous researchers have contributed to the set of tools now in use for generating wind tunnel boundary layers that are very deep [for e.g., Cermak (1971); Cook (1973); Davenport (1966); Farell and Iyengar (1999)]. Conventional approaches employ a combination of passive devices such as spires, barrier walls, and floor roughness to simulate planetary surface friction of varying classes of terrain. It is assumed that atmospheric velocity variations can be adequately modeled by stationary mean and turbulent flow properties. This assumption means that despite the fact that hurricanes and gust fronts can have non-stationary characteristics, wind hazards are evaluated in stationary flow environments. Wind tunnel turbulence intensities are matched to site values, and wind tunnel integral scales are scaled with the geometric scale of models. While this conventional approach has served (and still serves) research and industrial needs for some time, the following sections show how the relatively new wind simulation facilities can answer questions that cannot be addressed with the past generation of wind tunnels.

The near-ground flow environment in complex and built-up terrain consists of numerous obstacles all inducing flow separation and all generating shear layers that interact with each other. Free stream turbulence modifies the structure of separating and reattaching shear layers. The role of turbulence in these scenarios has been extensively documented in the literature [e.g., Gartshore (1973); Kareem and Cermak (1979); Hillier and Cherry (1981); Bearman and Morel (1983); Saathoff and Melbourne (1997); and others]. Although turbulence scale clearly influences flow structure, the precise mechanism of these influences has not yet been clear.

Small scale content—particularly scales on the order of the thickness of the separated shear layer—has been shown by a number of researchers to have a significant effect on separated shear layer flow structure [Gartshore (1973); Tieleman and Akins (1990)]. A significant difference in small-scale turbulence content can exist between wind tunnel and full-scale flows because wind tunnel Reynolds (Re) numbers can be as much as three orders of magnitude lower than those of atmospheric flows. Tieleman and Akins (1990) reported that wind tunnel simulations with insufficient small-scale turbulence content resulted in poorer comparisons between model and full-scale results. Decreasing Reynolds number disparities between model and prototype flows will increase our confidence in test results. Understanding the physics of
how flows depend on Reynolds number will decrease the uncertainty associated with imperfect turbulence simulation.

The Aerodynamic/Atmospheric Boundary Layer (AABL) Wind and Gust Tunnel at Iowa State University (ISU) was designed specifically with the capability to examine these issues—particularly relating to the smallest and the largest scales of turbulence. This tunnel has a unique active gust generation capability. AABL Wind and Gust Tunnel [Sarkar and Haan (2008), see Figure 1, constructed in 2005] is primarily a closed-circuit tunnel that can be also operated in open-return mode. It is designed to accommodate two test sections (2.44m x 1.83m and 2.44m x 2.21m) with a maximum wind speed capability of 53 m/s. The gust generator is capable of producing gust magnitudes around 27% of the mean flow speed. The wind tunnel is one of the several wind tunnels that are housed in the Wind Simulation and Testing Laboratory (WiST Lab) in the Dept. of Aerospace Eng. at ISU.

Fig. 1 AABL Wind and Gust Tunnel at Iowa State University

**Gust Simulation**

Large-scale turbulent gusts also constitute an important aspect of wind tunnel simulation. Passive turbulence generation techniques (such as bluff obstacles) have been shown to produce only a limited range of possible integral scales [Bienkiewicz et al. (1983)] and are often not large enough to match prototype scales. As a result, active turbulence generation schemes have been developed to produce much larger integral scales [Bienkiewicz et al. (1983); Kobayashi et al. (1994); Cermak et al. (1995)]. To increase the range of turbulent flow scenarios that can be simulated, some researchers have employed arrays of individually-controlled fans as well
These types of devices have not generally been used to simulate the non-stationary gusts that can occur in hurricanes and thunderstorms. Extreme wind loads, however, result primarily from extreme weather events (such as gust fronts, hurricanes, etc.) where non-stationary gusts, transitional flow structures and rapid wind directionality changes may play a significant role. The current state-of-the-art boundary layer wind tunnels are incapable of physically simulating the transient effects of such events.

Anemometry data from hurricanes and thunderstorms have shown that velocity records are non-stationary at times [e.g., Schroeder and Smith (1999), Orwig-Gast and Schroeder (2005)]. Further, while advanced analytical simulation methods for non-stationary wind fluctuations have been developed [Chen and Letchford (2005); Wang and Kareem (2005)], little experimental work in this area has been attempted. For thunderstorm-related gusts, Simiu and Scanlan (1996) summarize ranges of these parameters to involve gust magnitudes of 3 m/s to 30 m/s and gust durations from a few minutes to 20 minutes or more. Hurricane data available online are consistent with such ranges. For example, a Hurricane Andrew [Rappaport (2003)] summary reported maximum sustained winds of 64 m/s with gusts up to 77 m/s (a 20% increase). Hurricane Frances data [Florida Coastal Monitoring Program (2004)] showed that 3-second gusts could be 20% to 50% above the 15 minute average velocities. In a review of a number of 1995 hurricanes, Powell and Houston (1998) reported gusts from 15% to 40% above the maximum sustained wind speeds. Values for the rate of velocity changes were also gathered from published literature. Holmes and Oliver (2000) developed a model of the widely reported Andrews Air Force Base downburst. Their model shows a rate of velocity increase of 0.4 m/s$^2$ during that event. While measuring wind speeds on an instrumented tower during a weak downburst event, Sherman’s (1987) data shows wind speeds changing at a rate of approximately 0.04 m/s$^2$. Data from a similar scenario reported by Orwig-Gast and Schroeder (2005) showed rates up to 0.25 m/s$^2$.

It is important to understand the nature of wind loading on structures and vehicles, in variable gusts that can occur in gust fronts, thunderstorm or hurricane winds. This understanding will help in a better prediction of wind-related risk. There has not been a great deal of research in the area of gust loading because of several reasons. The most important of these reasons is the need for a wind tunnel facility capable of generating a gust comparable to that in nature. The ISU AABL Wind and Gust Tunnel’s gusting system [Haan et al. (2006)] was designed to study just such phenomena and it was designed to generate variable gust shapes. The unique feature of the AABL Wind and Gust Tunnel is that the static pressure remains approximately stable inside the test section during a gust unlike most experiments that were recently carried out in other wind tunnels which use vanes or shutters to create a pulse of velocity. These experiments generate changes in static pressure that have undesirable effects on the experimental data. Given the typical geometric scales of 1/50 to 1/200 for low to medium-rise structures and typical velocity scales of 1/3 to 1/2, a time scale of 0.01 to 0.06 and an acceleration scale of 5.6 to 50 are calculated. In other words, the full-scale rates of velocity changes or acceleration described above (0.04 m/s$^2$ to 0.4 m/s$^2$) would have to occur at rates of 0.2 m/s$^2$ to 20 m/s$^2$ in the wind tunnel. Thus a wind gust of 15-50% of the mean velocity is desirable in as little as 3 seconds with an acceleration of 0.2-20 m/s$^2$. Based on the field parameters and practical limitations, it was decided to accomplish a gust of 15-25% of the mean wind speed in the wind tunnel at a rate of 0.2-10 m/s$^2$ in a maximum period of 5 sec. For example, the system should be able to reach a steady-state mean wind speed of 25 m/s up from 20 m/s (a 25% increase) in 5 seconds (an acceleration of 1 m/s$^2$). For a length scale of 1/50 and a velocity scale of 1/2, this gust would
correspond to a full scale event going from 40 m/s to 50 m/s in 125 seconds (an acceleration of 0.08 m/s²).

The basic design of the gust generator that was chosen to achieve the requirements was a bypass duct. The bypass duct, conceptually similar to the transition facility described in Saric (1992), diverts flow from the main duct (Figure 2). This diversion reduces the flow velocity in the main test section. Computer-controlled dampers dictate the amount of flow diverted and the time scales involved. The gust generator is capable of producing non-stationary gust magnitudes up to 25% of the mean flow speed with time and velocity acceleration scales comparable to a wide range of full scale thunderstorm and hurricane gust events. These gust magnitudes are achieved with minimal changes in static pressure drop (2%) across the fan where conventional techniques of using vanes or airfoils to block the flow require large static pressure changes (up to 40% or more) to achieve similar results. These gusts are also achieved without any significant effect on the test section velocity uniformity or turbulence level.

The different gust shapes (ramp-up, ramp-down, triangular, trapezoidal) that were generated in the AABL Wind and Gust Tunnel are shown in Figure 3. The velocity magnitude reaches approximately the steady state value between 2.5 to 5.0 sec depending on the initial and final velocities and gust shape. Marshall and Krayer’s (1992) curve from Hurricane Bob gives a gust factor of approximately 1.25 for gusts of 100 seconds duration (1-sec to 6-sec in Wind Tunnel). Thus, the gust magnitudes occurring in nature during gust fronts, thunderstorms and hurricanes are well within the capabilities of this gust generation system.

Fig. 2 Diagram of the bypass duct surrounding the portion of the main wind-tunnel duct containing the fan showing the flow when the bypass ducts are open [Haan et al. (2006)].
**Tornado Wind Simulation**

Velocity fields resulting from tornadoes differ significantly from conventional atmospheric boundary-layer type winds. Various types of tornado simulators (or tornado vortex chamber) have been built in the past to create tornado-like vortices in the laboratory and study the vortex dynamics by varying the controlling parameters [Ward (1972); Snow and Lund (1988); Monji and Wang (1989); Church et al. (1979); Haan et al. (2008); Matsui and Tamura (2009); Hashemi-Tari et al. (2010)].

Sarkar and his team at ISU have studied tornado-induced wind flow in the lowest elevations above ground, mainly on smooth terrains, and assessed tornado-induced wind loads on low-rise buildings (residential) using a unique laboratory tornado simulator and sophisticated instrumentation including PIV and high-fidelity pressure/force sensors during the 2004-2012 period [Kardell (2004); Haan et al. (2008); Sengupta et al. (2008); Le at al. (2008); Haan et al. (2010); Yang et al. (2010), (2011a,b); Hu et al. (2011); Thampi et al. (2011); Zhang and Sarkar (2011); Kumar et al. (2012)]. The ISU Tornado Simulator [see Haan et al. (2008) for description, as conceived and designed by Sarkar and his team, built in 2004] that was used for the study generates tornado-like vortices in the range of 0.46 to 1.12m in diameter with maximum tangential velocities ranging from 6.9 to 14.5 m/s. It generates vortices that can translate
Horizontally at speeds of up to 0.61 m/s (2 ft/sec) that makes it different from all the earlier Ward-type simulators [Ward (1972)].

The laboratory simulator at ISU simulates a tornado-like vortex by producing a strong region of updraft, surrounded by a spinning tube of air that descends toward the ground plane. Updraft is generated with a 1.83m (6 ft)-diameter fan mounted inside a circular duct that is concentric with respect to a down-flow duct (see Figure 4). The fan duct and down-flow duct are suspended from an overhead crane so that they can move along a 10.4m (34 ft) long by 6.1m (20 ft) wide ground plane. The tornado that forms moves along the ground plane as the entire fan/downdraft-producing mechanism translates.

This facility allowed model tests with larger geometric scales of low-rise buildings (say 1:75) that were not possible before for other tornado simulators. Single-celled vortices and two-celled vortices were generated with corresponding swirl ratio (S) ranging from 0.08 to 1.14 (S = \( \pi V_{\text{max}} r_c^2 / Q \) = ratio of angular momentum of flow to inflow rate (Q), \( r_c \) is core radius where maximum tangential wind speed \( V_{\text{max}} \) occurs; results in 80% or larger values if conventional defn. of S is used as for real tornadoes whose S=2-6 based on radius of updraft). The laboratory data from this simulator were validated with field measurements from Spencer 1998 (South Dakota) and Mulhall 1999 (Oklahoma) tornadoes. Both the velocity profiles and ground pressure distribution compared very well between lab data [Haan et al. (2010)] and field data [Lee and Wurman (2002); Wurman and Alexander (2005); Samaras and Lee (2006)]. Tornado-induced wind loads on a gable-roofed low-rise building and a high-rise building were measured and compared with wind load provisions of ASCE7-05 (2005) in Haan et al. (2010) and Sengupta et al. (2008). It was found that peak wind loads (roof uplift) resulting from tornadoes of F2 intensity could be 1.5 to 1.8 times greater and shear force could be 1.5 to 2.1 times greater than those specified in wind load provisions of building codes for buildings in the Midwest. Finite-element analysis (FEA) combined with laboratory measurements of tornado-induced loads was
Partha P. Sarkar

successfully used to predict the building damage in a tornado as observed in a post-damage survey [Thampi et al. (2011); Kumar et al. (2012) as discussed later. Some of these results were found consistent with others’ work [Hangan and Kim (2008), Mishra et al. (2008)].

**Microburst Wind Simulation**

Lundgren et al. (1992) and Alahyari and Longmire (1995) conducted laboratory modeling of microbursts [Fujita (1985)] by using fluids of different densities to study downbursts for the aircraft industry. The diameter of the simulated downburst was 64 mm (2.5 in). Separate efforts by the author (Sarkar), Homes, Letchford, Selvam and Twisdale and their co-workers have shown that the estimation of structural loads in microburst winds needs separate treatment. Selvam and Holmes (1992) used a 2D $k$-$\varepsilon$ turbulence model to study topographic multipliers, and Wood and Kwok (1999) observed static impinging jets (experimental and numerical) to study topographic effects on velocity profiles. Letchford and Illidge (1999) observed a static impinging jet to study velocity and turbulence characteristics at various distances from the center of the jet (experimental). Holmes (1999) used a wind-tunnel jet to study velocity characteristics. Chay and Letchford (2002) measured pressure distributions on a cube in a laboratory simulated thunderstorm downburst—a moving jet of 0.51 m (1.67 feet) diameter was used. Mason et al. (2005) used a pulsed jet downburst to examine the interaction of a moving ring vortex with a structure. Holmes and Oliver (2000) 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. (2001)]. Sarkar conducted numerical and physical simulation of microburst winds using an impinging jet [Sengupta and Sarkar (2008)]. Their initial work was conducted with a small-scale microburst simulator (D=0.20 m, 0.67 ft) used to test concepts for the design of the large simulator. Velocities were measured with hot-wire, pressure-rake and PIV systems. Sengupta and Sarkar (2008) and Letchford et al. (1999, 2002) have shown that traveling microbursts can produce mean and fluctuating winds and, therefore, mean and fluctuating loads that are quite different from those generated by straight-line winds. Sarkar and his co-workers [Zhang et al. (2013a), (2013b)] have recently compared different laboratory models of a microburst including a steady impinging jet model using a laboratory simulator (Figure 5, D=0.61 m) and a transient impinging jet model and a numerical cooling-source model. The transient impinging jet model and the cooling source model were studied to capture the transient features. The 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.

**Dynamic (Aeroelastic) Simulation Facility**

The Bill James Open-Return Wind Tunnel, which is located in the WiST Lab at ISU, accommodates a test section for aeroelastic tests on section models. This wind tunnel has a test
section of 0.915m (3.0ft) width by 0.762m (2.5ft) height and its maximum operational wind velocity is 61 m/s (200 ft/s). A NACA0012 airfoil section model is shown in Figure 6a inside the test section as suspended from a three-degree-of-freedom suspension system as developed by Sarkar et al. (2004). This suspension system enables vertical, horizontal and torsional motions of the models in both free and forced vibration. In a typical free vibration experiment, the model is released from an initially displaced position that is enabled by using multiple strings attached to the model and pulled by electromagnets [Chowdhury and Sarkar (2004)]. Decayed time histories of displacements of the model at a given wind speed are recorded and used in extracting frequency-dependent flutter derivatives for flutter analysis. For forced vibration tests, for example in a two-degree-of-freedom setup, sinusoidal vertical and torsional motions of the model with constant amplitudes and frequencies are realized by a driving mechanism connected to the model suspension system with four aluminum rods as shown in Figure 6a. The entire mechanism is driven by two motors which are placed above the test section, as seen in Figure 7. One motor is used to drive vertical motion, while the other is used to drive torsional motion of the model. By changing the rotating speed of the two motors independently using two separate controllers, the frequency of vibration in each of the two degrees of freedom can be changed independently. Time histories of surface pressure on the model as it vibrates in the given wind flow are recorded by Cao and Sarkar (2012). Dynamic lift and moment, obtained by integrating the surface pressures, are used along with the displacement of the model in the analysis for extracting parameters of aeroelastic force models for flutter analysis in frequency and time domains.

**Measurement - Instrumentation**

Measurements in transient flows in wind tunnels have become possible within the past decade with the emergence of modern instrumentation and data acquisition that is sophisticated, accurate and fast. Hot-wire and hot-film anemometers for velocity measurement are no longer instruments of choice for non-stationary and three-dimensional flows like in tornadoes and microburst. Multi-hole pressure probes (Cobra, Omni) are used instead.
Fig. 6(a) A section model [NACA0012 airfoil] and the suspension system,
(b) A close-up view of the system for torsional motion at one end

Fig. 7 Driving mechanism for the forced-vibration system
Cobra probe (4-hole pressure probe, TFI Pvt. Ltd.®) has the ability to measure all three velocity components simultaneously with high accuracy (uncertainty ±0.5m/s) and high sampling rate (> 1 kHz) at each measurement point for flow that is confined within a ±45° cone. Omni probe (18-hole pressure probe) has the ability to measure all three velocity components simultaneously for flow confined within a ±165° obtuse cone. While Cobra probe can be used to measure transient 3D-flow that has all three non-zero mean components with large fluctuations, Omni probe can be primarily used to measure mean components of transient flow where flow reversal may occur. The Cobra probe is a standalone probe with in-built pressure sensors, analog to digital conversion and customized data acquisition system. However, the Omni probe works with external pressure transducers that most wind tunnel labs have access to. Figure 8 shows both the probes.

Pressure measurement is routinely used in wind tunnels for assessing point pressures on models and overall forces that are obtained by integrating the pressures. High-frequency pressure transducers are used for this purpose. These transducers come in different capacities, with varied accuracies and in multiple brands. An individual transducer can have a single sensor or multiple sensors or channels (e.g. 16, 32, 64). They can be analog or digital. For example, Scanivalve® Corporation has a line of ZOC (Zero, Calibrate and Operate) series of transducers that are analog and the DSA (Digital Sensor Array) series of transducers that are digital. A ZOC33/64Px (64-ch, ±10in H2O, ±0.15% FS accuracy, 625 Hz/ch sampling frequency) and DSA3217 (16-ch, ±10in H2O, ±0.05% FS accuracy, 500 Hz/ch sampling frequency) pressure scanners are shown in Figure 8. The DSA3217 is a network-ready Ethernet pressure scanning module. These transducers have in-built temperature compensation and don’t need separate data acquisition software.

Force measurement is used in many wind tunnel tests depending on their purpose. For example, displacements of aeroelastic models can be calculated from force measurements with single-component transducers that are attached to elastic springs used in the model’s suspension. Base balances are used to measure the overall wind loads on civil structures although in most cases these can be obtained by integrating the surface pressures. Pressure integration is preferred sometimes to direct force measurement because of the excellent frequency response of the pressure transducers. Force transducers are available as a single-component (force or torque) force balance or six-component force balance (e.g. JR3 seen in Figure 8) that are compact. The JR3 load cell is capable of measuring forces in three directions and the bending moment or torque about each axis with a measurement uncertainty of ±0.25% of the full range.

With the rapidly development of the modern optical techniques and digital image processing techniques, laser-based optical diagnostics is providing researchers and engineers with the capability for remote, non-intrusive, in-situ, spatially and temporally precise measurements of important flow parameters like flow velocities and pressures. This means that, optical diagnostics provides the researchers and engineers not only with the more accurate measurements but also with completely new capabilities. Spatially precise and instantaneous measurements at high rates permit the flow field to be frozen and tracking with high frequency response. Measurements at many locations simultaneously along a line, over a plane or in a volume permits spatial correlation to be obtained providing new phenomenological insight into fundamental behaviors of complex flow phenomena. Particle Imaging Velocimetry (PIV) and Pressure Sensitive Paint (PSP) are two laser-based measurement techniques that can be used to measure flow velocity and surface pressure distributions. Compared with other traditional velocity and pressure measurement techniques, PIV and PSP techniques are non-intrusive...
measurement techniques because these are optical techniques. Papers and textbooks about PIV [Adrian 1991, Grant (1997), Raffel et al. (1998)] and PSP [McLachlan et al. (1995), Liu and Sullivan (2004), Gregory et al. (2008)] are available for review of these techniques. Hui et al. (2010) describes the PIV and PSP procedures as follows. PIV relies on seeding the flow with tiny particles and observing the motion of the tracer particles to derive fluid velocity. In PIV, a sheet of laser light is usually used to illuminate the region of interest. The tracer particles scatter the laser light as they move through it. Photographic film or charge-coupled device (CCD) cameras are used to record the positions of the tracer particles. The positions of the tracer particles are recorded at two different times, separated by a prescribed time interval. The displacements of individual particles, or more often groups of particles, are determined by a well-developed computer-intensive procedure. The displacements over a known time interval provide the estimate of the particle velocity vectors. The velocity of the working fluid is deduced...
based on the assumption that the tracer particles move with the same velocity as the local working fluids. As shown in Figure 8, a typical setup of a PIV system [Zhang and Sarkar (2011)] that was used to visualize a laboratory-simulated tornado flow consists of several sub-systems such as a CCD camera or two (stereoscopic), a dual-head Nd Yag Laser with a control unit, optical lenses/splitter and a computer. It involves particle seeding, flow field illumination, particle image acquisition and PIV image processing. PSP is a relatively new measurement technique for surface pressure distribution measurements in wind tunnel testing [McLachlan et al. (1995), Liu and Sullivan (2004), Gregory et al. (2008)]. PSP has been quickly gaining recognitions as an important experimental tool for optically and quantitatively measuring partial pressure or concentration of oxygen near model surface based on luminescence quenching. In PSP measurements, luminescent molecules are suspended in a polymer binder to create the paint. Upon the excitation of photons at an appropriate wavelength, the molecules can emit photoluminescence light at a longer wavelength. The intensity of the light emission from excited molecules in the paint is related to the partial pressure or concentration of the oxygen on the painted surface.

Other Unique Wind Tunnel Facilities

There are a few unique wind tunnel facilities in North America that have recently emerged. These facilities can address larger length scales, real wind conditions, higher Reynolds number, and wind-rain conditions. Two such facilities are described here. The Wall of Wind (WOW) facility at Florida International University (FIU) is shown in Figure 9 (courtesy of Dr. A.G. Chowdhury). This facility and its capabilities are described in Irwin et al. (2013). It consists of twelve 1.83m (6ft) diameter fans arranged in two tiers, with 6 fans at each tier, that can produce up to 70m/s (157mph) or hurricane-level wind. The fans are arranged in a circular arc configuration and powered by a VF drive system with a total of 5.97MW (8,000 hp) where each tier can be controlled separately. They blow into a contraction section that is then followed by a 6.1m (20ft) wide by 4.3m (14ft) high flow conditioning section. In the flow conditioning section spires and floor roughness are used to tailor the mean velocity profile, turbulence intensity and integral scale of the turbulence to desired values. The test specimen is located on a 4.9m (16ft) diameter turntable downwind of the end of the flow conditioning section. The WOW facility, a new tool, is now available for creating close to real wind conditions, with and without rain. By simulating the small-scale turbulence that interacts with shear layers and vortices, the important aerodynamic effects of wind are included and large scale turbulence can be handled by treating it as equivalent to slow changes in oncoming flow speed and direction.

The Wind Engineering, Energy and Environment (WindEEE) Dome at University of Western Ontario is the world’s first hexagonal wind tunnel that was inaugurated recently in October 2013. As described at http://www.eng.uwo.ca/windeee/facilities.html, its large scale structure (25m-diameter inner dome and 40m-diameter outer return dome) will allow wind simulations over extended areas and complex terrain. WindEEE will allow manipulation of inflow and boundary conditions to reproduce large turbulence scales under controlled conditions. Mounted on the peripheral walls and at the top of the dome, an array of specialized fans will be activated using a sophisticated control strategy to provide time-varying and spatially-varying flow fields in the test section. A large variety of wind fields such as boundary layers, portions of hurricanes, tornadoes, downbursts, low level currents or gust fronts will be physically simulated.
Techniques

Wind Tunnel Models

While full models of buildings are commonly used in wind tunnels, full models of long-span bridges are not so common because of lack of wind tunnels with wide test sections and prohibitively high cost associated with building these models. Section models of long-span bridges or tall poles or any other slender structure like an airplane wing are more commonly used because they (a) can be accommodated in relatively small tunnels, (b) can be easily built and therefore are cheap, (c) can be easily modified and hence useful during the conceptual design of the structure. Section model of a structure faithfully represents the cross section of the structure over a finite length. Section model limits the flow to 2D and hence can’t be used to study the combined effects of terrain/topography and wind directionality. Analytical methods need to be used to predict the prototype response using aerodynamic properties of the section model. Models can be classified as rigid or aeroelastic depending on the purpose of the test. Rigid models are used either for measuring surface pressures or overall forces or both while aeroelastic models are used for measuring the dynamic response (displacement or acceleration) along with surface pressures. The material of choice for a model depends on the user and the purpose of the test. Balsa wood because of its superior surface finish is a common choice for models. However, plexiglass which also provides a smooth surface is a cheaper alternative to Balsa wood. Rapid prototype (or RP) printers are now commonly used to build models with complex design, for example buildings with architectural features. RP technology allows the models to be light, with smooth finish and even with variable stiffness. One advantage of a RP printed model is that it can have surface pressure taps and embedded channels connecting the taps to outside ports as part of the model, reducing the requirement of drilling taps and connecting them with flexible tubes. One such example is shown in Figure 11, where thin panels of a photovoltaic or solar cell needed to have surface pressure measurement on both sides. Styrofoam cutter and 3D router can be also used to carve out models of certain shapes. Examples of such models is a section model of an airfoil (Figure 10) and buildings used to test interference effects of surrounding structures (Figure 13).

Shape, thickness and size of end plates used in a section model and its aspect ratio (ratio of length to depth) are important parameters for getting accurate results. Thin aluminum end
tes are preferable. The size of the end plate should be such that there is enough distance between the model and the edge of the end plate for the flow to separate and the wake to develop. This distance should be proportional to the thickness or depth of the model and dependent on the range of angle of attack used. If measuring force on the section model then it is preferable to physically detach the end plate from the model, especially if it has a small aspect ratio, that the forces on the end plate does not contribute to the total force measured. The size of the wind tunnel limits the aspect ratio of the model to be tested which is recommended to be greater than equal to 10. To maximize the aspect ratio, the model could be aligned along the wind tunnel tension, width or height, whichever is larger. The geometric scale of the model and hence dimensions of its cross section are usually chosen based on the recommended blockage criterion of maximum projected area of the model in any rotated configuration on a plane normal to wind to the cross sectional area of the wind tunnel) of 5% or less.

Figure 10 shows a typical full model of a low-rise building and Figure 12 shows an elastoplastic-section model of a bridge used in forced-vibration tests. Both models were made out of plexiglass with pressure ports drilled on the surface.

Windborne Debris and Internal Pressure

Windborne debris is considered a major source of damage during strong wind storms such as hurricanes and tornadoes. After wind-induced failure, building components can become airborne as missiles can cause significant damage to the surrounding structures. The shape, trajectory and velocity of the debris are important parameters to determine its impact on a given structure. Tachikawa et al. (1983) has done extensive research in the area of plate-type debris, attributing the Tachikawa number (a non-dimensional ratio of the aerodynamic forces to gravitational forces for trajectories of flat plates), an understanding of the autorotational component of plate flight, and demonstrating that the mode of flight (autorotational, 3D rotational, or translational) can account for some of the spread of full scale observations of flight distances. Wills et al. (2002) characterized wind-borne debris by shape and aerodynamic properties into three groups: compact, plate-like, and rod-like. Holmes et al. (2006) and Baker (2007) applied model equations to predict horizontal flight speeds in uniform 2-D flow for applications to impact testing.

Kordi et al. (2009) showed that the buoyancy parameter, rotational drag, and initial conditions significantly affect the flight path of flat plates in a uniform 2-D flow. There has been little investigation of predicting wind-borne debris flight in tornado or hurricane wind. Hayama (2011) used a numerically generated tornado simulator to create various tornado vortices and model the trajectory of a simplified debris “compact” object.

ISU’s Tornado Simulator was used to validate the numerically simulated trajectory of two types of wind-borne debris (a compact object and an elongated object). While the details of the experimental procedure to map the trajectory and numerical method to predict it can be found in Crawford (2012), a brief description of the experimental procedure and results follow. The ordinates of the trajectory in the experiment were captured using two video cameras and principles of stereo-photogrammetry (see Figure 14).
Fig. 10 (a) Section model (aeroelastic) of NACA012 airfoil, (b) Full model (aeroelastic) of a twin cable-stayed bridge (courtesy: Y. Fujino), (c) Full model of a sports light, (d) Prototype sports light in tunnel, (e) Full model of a sports-light array, and (f) Full model of the TTU building with pressure taps (courtesy: CSU).
Fig. 11  (a) CAD drawing of a solar cell showing pressure taps, (b) Rapid prototype of the solar cell model connected to flexible tubing, (c) CAD drawing showing embedded channels that connects pressure taps on both sides of the solar cell to flexible tubing.

Fig. 12  Section model of a box-girder bridge with the pressure taps on the top and bottom surfaces along its centerline connected to a pressure transducer and a close-up view [right] showing the pressure tubes inside the model.
The experimental trajectories were compared to a numerical simulation model that used the aerodynamic properties of the selected debris shape such as force and moment coefficients either measured in a wind tunnel for various yaw and pitch angles or taken from the literature. The wind-borne debris models that were used for validation were (a) a sphere that is representative of compact objects, and (b) a circular cylinder with an aspect ratio of 3:1 (length to diameter) that is representative of a slightly elongated object such as a vehicle or a timber/steel beam used in construction. The spherical object was suspended by a string of a given length (l) attached to the center of the simulator and allowed to reach equilibrium in the vertical plane (i.e. rotating at a constant $r$ and $h$) during a tornado simulation. Equations of dynamic equilibrium were used to balance the forces acting on the sphere such as centripetal force, gravitational force, tensile force from the string, pressure force and aerodynamic drag force. The predicted and the measured values of the radial distance “r” of the rotating sphere in dynamic equilibrium were compared. The results of this controlled-flight test show excellent match (within ~2-7%) between the numerically predicted values and measured values. In a separate set
of tests, the spherical objects of different diameters (25.4 and 38.1 mm) and masses (0.19 and 0.77 g) were released at a pre-determined height from the ground plane and radial distance from the tornado vortex (within the tornado core). Locations of the experimental trajectories of the free-flight tests were calculated for times corresponding to the video frames used. The locations of the numerically simulated trajectories were calculated at very small time steps (0.0035 sec), but only the locations at those time steps corresponding to the experimental time steps were used in the comparison. Velocities of the experimental trajectories were found by taking the distance traveled between two frames and multiplying by the frame rate, in this case 30 fps. Comparison of numerically simulated 3D trajectory based on the analytical model versus experimental trajectory for the spherical debris is shown in Figure 15 until the time when either trajectory impacted the ground. The comparison between the two results is quite favorable.

The windborne debris can result in the opening of a structure and thereby influence the internal pressure which plays a major role in the net pressure acting on various structural components in a wind storm. The internal pressure can change the failure mode of the structure subject to wind loads and hence important to consider. Internal pressure coefficient can be positive or negative in a straight-line wind and specified in building codes for use in combination with external pressure coefficients to assess the net wind loads. Internal pressure is not measured in every wind tunnel test (for straight-line wind) because it can be assessed for common structures based on the past work. However, internal pressure which in a straight-line wind reaches a steady-state value over a given time after a breach in the structural envelope is quite dynamic in transient wind. Therefore, it needs to be measured and analytical/numerical model of the internal pressure simulation in transient wind based on openings and porosity in the structure needs to be formulated.

![Fig. 15 Comparison of experimental vs. numerically simulated trajectory of a spherical debris (25.4mm dia., 0.19g) for initial conditions (r0 = 305mm, h0 = 279.5mm)]
To measure internal pressure, the internal volume of a laboratory model (Figure 16a) has to be scaled to maintain the similarity of the dynamic response of the volume at model scale to that in full scale [Holmes (1978)]. The internal volume scale ($\lambda_{vol}$) is calculated as follows:

\[
\lambda_{vol} = \left( \frac{\lambda_t}{\lambda_v} \right)^{\frac{3}{2}}
\]

To study the effect of internal pressure in a tornado, tests were conducted on a low-rise building at 1/75 geometric scale with three different model opening configurations [Thampi (2010)]. These configurations were: (1) sealed building (closed doors and windows; not fully sealed because of porosity in the cladding), (2) open building (with all 2 doors and 11 windows open), and (3) dominant opening (with only one open door). The time series of the pressure coefficients, as the tornado translated at a speed of 0.30 m/s, was obtained. It was found more useful for this work to observe the pressure as a function of the distance $x$, normalized w.r.t. core radius $r_c$ instead of observing it as a function of time. The model contained 5 taps inside the building at different locations of the building to capture the internal pressure in different building chambers. Figure 16b shows the internal pressure coefficient, $C_{pi}$, at one location of the building as a function of opening when the tornado is located at $x=0$ which is closest to the building. Opening is defined here as the percentage ratio of the total opening area to the total surface area of the building. The porosity in the material properties was ignored in the calculation. Thus, zero percent opening represents a sealed building (Configuration 1) and 3 percent opening represents an open building (Configuration 2). The negative maximum internal pressure coefficient seen in Figure 16b corresponds to the dominant opening case (only one open door, Configuration 3).

Figure 17 shows a time history of external ($C_{pe}$), internal ($C_{pi}$) and net ($C_{pnet} = C_{pe} - C_{pi}$) pressure coefficients with the passage of a tornado over a building (located at $x=0$ and $y = 1.42r_c$ offset from the centerline of the tornado path) with dominant opening (one door open).
Important Scales

It is known that two scales (defined as ratio of model to prototype parameter) that play an important role in ABL wind tunnel simulation are the length (or geometric) scale and the velocity scale. These scales determine the other scales like the time and frequency scales and the non-dimensional parameters, such as Reynolds number, Froude number, Strouhal number, etc., that are of importance to the wind flow simulation. With advances in transient flow simulation, where three dimensionality became dominant, other scales were added to this list. For example, to put the tornado simulation in context, the flow parameters that control the dynamics of tornado vortices include the radial Reynolds number, the swirl ratio (defined earlier) and the aspect ratio (height to radius of inflow). Core radius and maximum tangential or azimuthal wind speed that determine the swirl ratio are of importance when comparing simulated tornadoes with the real ones. The swirl ratio (S) is a measure of the amount of rotational energy in the vortex relative to the convective energy in the vortex and correlates well with vortex structure [Church et al. (1979)]. With sufficiently high swirl ratios, the vortex breaks down into multiple vortices rotating around the main vortex. Davies-Jones (1973) explained that vortex flows with large swirl ratios break down into multiple vortices because they cannot generate sufficient radial convergence to counteract the large outward centrifugal force. Both of these studies (covering experimental and numerical work) reported that vortex simulations with S below 0.5 produced “one-celled” vortices. They also reported that the vortex transitions into a “two-celled” vortex above S of 0.5 and that multiple vortices appear when S is greater than 1.0. Lee and Wurman (2005) made the first attempt to estimate the swirl ratio in a real tornado [the Mulhall tornado of 1999] using velocity data derived from Doppler radar. Axisymmetric radial and azimuthal velocities were obtained from the radar and then used to integrate the continuity equation to obtain the vertical velocity component. With the accompanying increase in uncertainty, they reported swirl ratios ranging from 2 to 6. Mulhall tornado observations included both two-celled structure and multiple vortices. Additional scales need to be considered in transient events, like a separate velocity scale to model the translation speed of the tornado or microburst. It was argued by the author and his co-workers that the time scale for translation in tornado and microburst
Limitations of Wind Tunnels

Wind tunnels have certain limitations that users need to understand when comparing data with full-scale measurements. It is after all modeling a complex flow phenomenon in an idealized environment. The simulated flow inside a wind tunnel is confined within a physical boundary unlike the natural flow. The wind tunnel model is scaled down version of the prototype and hence point pressures are no longer point pressures in full-scale, rather are area-averaged pressures. There is deficit in small-scale eddies and shortfall in lateral and vertical turbulence. Only mechanical turbulence can be produced in wind tunnels whereas in most cases the temperature and pressure driven convective effects of the atmosphere cannot be produced. Simulation of rain or snow in combination with wind cannot be produced in closed test sections as sometimes are required. Larger facilities like the WOW at FIU and the wind tunnel facility can address some of these issues but cost of testing could be prohibitive.

Tools

CFD Simulation

CFD and numerical simulations play a complementary role to physical simulations. With reference to tornado simulation, for example, CFD simulations of tornadic flow have been performed as a tool to improve understanding of tornado dynamics, small-scale flow characteristics, and possible genesis mechanisms [e.g., Lewellen and Lewellen (1997); Lewellen et al. (2000a); Le et al. (2008)] because of prior difficulties in collecting small-scale observations within and near a tornado. Many of the numerical models developed were patterned after existing laboratory simulators, Wicker and Wilhelmson (1993) found that the inclusion of surface friction into simulations of tornados (instead of assuming free slip lower boundary conditions) can significantly affect the evolution and intensity of the tornado. More recently, Lewellen et al. (2000a) found in numerical studies that changes in surface layer inflow can affect
substantially the level of intensification and turbulent structure within the corner flow, even if the swirl ratio of the tornado vortex itself does not change. Thus, differences in this inflow, which could arise from variations in surface topography or roughness [Lewellen et al. (2000b)] may determine whether or not a tornado vortex is spawned at all from a supercell, or if the damaging winds impact the surface. In addition, Lewellen et al. (2000b) suggest that the impact of other flow features such as the rear flank downdraft wrapping around and cutting off the low swirl inflow near the surface could play an important role in the tornado structure and evolution. The emphasis in the numerical simulations in the past has generally not been on near-ground tornado flow due to uncertainties in the parameterization of surface-atmosphere interaction. However, such studies have suggested that the peak winds in a tornado may occur in the lowest 20 m above ground, or below the lowest level at which radar observations are available. Le et al. (2008) suggests that numerical models are able to simulate tornado flow, with generally good agreement with radar observations in FLUENT simulations that were initialized with radar data.

For microburst simulations, Mitchell and Hovermale (1977) used a non-hydrostatic, two-dimensional, fully compressible model. Droegemeier and Wilhelmson (1987) used a 2D quasi-inviscid, compressible model without turbulence closure scheme. A damping term was added to dissipate energy. Proctor (1988) employed an axisymmetric TASS 3D time-dependent model using first order closure for sub-grid turbulence. Commercial software packages (FLUENT and STAR-CD) were used by Sengupta and Sarkar (2008) to generate 2D and 3D simulations of the flow field of an impinging jet to model a microburst. The CFD models that were used included Standard k–ε model, RNG (renormalization group) k–ε model, SST k-Omega (k-ω) model, Realizable k–ε model, RSM or Reynolds Stress Model and Large Eddy Simulation (LES) model [Sengupta and Sarkar (2008)]. All the turbulence models captured the vertical- and radial-velocity profiles as well as the trends of the boundary-layer growth and decay of maximum radial velocities with increasing radial distance, but the predicted values from Realizable k–ε and RSM models matched the experimental data better than the other three models. It was also found that turbulence models have certain limitations which LES was able to overcome.

**Numerical Simulations**

Numerical simulation of wind field combined with numerical load simulation on a structure using its aerodynamic properties measured in the wind tunnel is sometimes necessary. When the turbulence is stationary and Gaussian, the conventional spectral representation method [e.g., Shinozuka and Deodatis (1991); Shinozuka and Jan (1972)], which relies on the fact that a stationary random process can be represented by its power spectral density function (PSD) and its probability density function (PDF), is used for spatio-temporal simulation of wind field. Established spectral models [e.g Kaimal] and coherence functions [Davenport (1962)] of the wind turbulence are used in this method. If the probability distribution of the wind turbulence is non-Gaussian, methods based on PSD with spectral correction or spectral distortion [e.g., Gurley et al. (1996)] can be used to facilitate the simulation of wind fields.

**Data Analysis**

Various techniques have been developed in the past to process stationary data that are collected in straight-line wind tunnels for understanding aerodynamics of bluff bodies that are common in wind engineering. For example, the pressure time histories on the building surfaces in the flow separation region are often non-Gaussian and thereby affects the loads on buildings. This feature has been long recognized and statistical methods have been proposed for data
analysis [Peterka and Cermak (1975); Peterka (1982); Stathopoulos (1982); Reed (1993); Gurley and Kareem (1997); Sadek and Simiu (2002); Tieleman et al. (2006)]. In most data analysis, whether point pressures or their integrated effects or forces, the mean, root-mean-square (RMS) and peak values (maxima and minima) are ascertained and presented. However, when dealing with non-Gaussian data, skewness and kurtosis and probability density functions (PDFs) are also calculated. Many researchers [Bienkiewicz et al. (1995); Holmes et al. (1997); Tamura et al. (1999); Chen and Kareem (2005)] have used proper-orthogonal-decomposition (POD) analysis for reduced-order modeling of wind pressure field. Pressure contours (without or with color-coded) on surfaces of models tested are commonly presented. Flow contours were deemed not so important in straight-line wind but are of great importance in transient flows to show both mean and fluctuating nature of the flow (Figure 18). Since the conventional spectral representation method cannot be used to analyze non-stationary pressure or load data from model tests in transient wind simulation (tornado or microburst) enhanced spectral representation method [Liang et al. (2007)] based on the evolutionary power spectral density function (EPSD) instead of the PSD may be used. It is known that non-stationary random processes are also non-ergodic, and therefore the temporal and spectral distribution of pressures and loads can be calculated based on ensemble statistics. Multiple simulations or data runs (10 or more) are needed to conduct numerical analysis and laboratory tests when simulating tornado or microburst wind to assess the ensemble statistics.

**Fig. 18** Normalized velocity and turbulence kinetic energy [Zhang et al. 2013b]

**Finite Element Modeling**

Finite Element Modeling (FEM) is routinely used to analyze civil structures subject to various loads for determining stresses in individual structural components and deformations in the structure. A lot of work has been done to deal with 3-D performances of timber framed buildings. One of the first analytical models was developed by Tuomi and McCutcheon (1974) which assumes linear elastic behavior of nails. Gupta and Kuo (1987) presented a linear building model with shear wall elements using 9 degrees of freedom and seven superelements. This model used a strain energy formulation and analyzed the building tested by Tuomi and McCutcheon (1974). Foschi (1977) developed a FEM which included nonlinear load-deflection properties for fasteners. Frame elements were modeled linearly and sheathing elements were modeled elastic and orthotropic. Kasal (1992) used the Finite Element software ANSYS to develop a three-dimensional model. It consists of linear orthotropic 2-D shell elements and fasteners represented by three 1-D spring elements at each node. The properties of nails, when pulled out and pulled through plywood and OSB boards were studied by Herzog and Yeh. He et al. (2001) developed a 3D model using the FE technique with plate, beam, and nonlinear nail
connections. Kasal et al. (1994) have developed a non-linear model of a complete light-frame wood structure and performed analysis under static loads. They were able to get a good correlation between the theory and experiments which were limited to static loads only. Foliente (1995) also has modeled the wood joint and structural systems under hysteretic behavior of wood under seismic loads. Paevere et al. (2003) has studied the load-sharing and redistribution in a one-story wood framed building which was subjected to lateral loading in static and static-cyclic modes. Starting from very simple linear models, the development has taken place in the analysis and more and more complex elements, such as orthotropic, and from linear to non-linear nail models have been incorporated. Researchers have studied the actual nail pull out and attempted to incorporate these into the Finite Element codes. Dutta et al. (2002) have studied the dynamic response of structures subjected to tornado loads by FEM method where an analytical model of a tornado was used. Kumar et al. (2012) have used ANSYS (2009) for full-scale numerical simulation of gable roof buildings with three different roof angles subject to tornado loads. The nail was modeled as a non-linear element but the wood was assumed to be linear. The tornado-induced wind loads recorded in the laboratory were scaled up and applied to the models to determine the detailed stress distribution in the structure. The deterministic FE model incorporated the damage criteria to assess the damage potential due to tornadic forces. The stress distribution, pattern of failure, the order of failure and the type of failure have been studied as the tornado sweeps past the building at different angles to the building centerline. Interaction of a EF5 tornado with a one-story gable-roofed timber building was studied by Thampi et al. (2011). A partially damaged one-story building, located within the damage path of the Parkersburg EF5 tornado (May 25, 2008), was chosen for analysis using Finite Elements (FE) and comparison of observed damage to those predicted in this study. This study was similar to that of Kumar et al. (2012), except the effects of changes in internal and external pressures on walls and roofs, which occur as a result of partial or total loss of cladding, increase in stiffness due to the presence of internal walls, decrease of stiffness as a result of wall openings and deteriorating structural components during the storm. Experiments were performed to obtain the pressure data on a geometrically scaled model (1:75) of the building placed in the ISU Tornado Simulator. The pressure data were applied on a finite element model of the building and the failure modes of the structural components were identified at different stages. The experimental simulations were repeated with the partially damaged model as predicted by the FE analysis to assess the change in loading and then followed by subsequent FE analysis with the updated data. This sequence was repeated to replicate the observed damage of the example building. Strength tests of different nail connections were performed to find the load-displacement curves for different nail connections to better represent the behavior of the nail in the FE model. The dynamic effects of changing internal and external pressures on the building are taken into account, as the tornado translates by the building and inflicts damage. The methodology predicted the successive stages of structural damage caused to the building by a translating tornado as a result of its interaction with the building components.

System Identification

In design of flexible structure such as a long-span bridge, it is important to identify whether there is aeroelastic instability [flutter] at wind speeds below the design wind speed. Scanlan and Tomko (1971) developed a technique to carry out flutter analysis in frequency domain using experimentally obtained flutter derivatives. This laid the foundation for the development of various efficient system identification (SID) techniques to extract flutter
derivatives from wind tunnel experiments, such as Scanlan (1978) and Sarkar et al. (1994)’s Modified Ibrahim Time Domain [MITD] method, Brownjohn and Jakobsen (2001)’s Covariance Block Hankel Matrix [CBHM] method, and Chowdhury and Sarkar (2003)’s Iterative Least Squares [ILS] method, and many other methods. Application of the frequency domain formulation is restricted from use in the nonlinear and transient domain because these methods are valid for linear models and stationary wind. The time-domain aerodynamic force formulations [Lin and Ariaratnam (1980); Scanlan (1984); Tsiatas and Sarkar (1988); Scanlan (1993); Chen and Kareem (2002); Caracoglia and Jones (2003)] are suitable for finite element modeling, feedback-dependent structural control mechanism, fatigue-life prediction, and above all modeling of transient structural behavior during non-stationary wind phenomena. This has motivated the developing of time-domain formulation of aerodynamic loads that are in parallel to the existing frequency-dependent formulation. Parameters vis-à-vis Rational Functions defining these time-domain formulation can be extracted from wind tunnel tests of section models [Chowdhury and Sarkar (2005); Cao and Sarkar (2013)]. In a section model only the rigid modes of vibration (either 3-degrees of freedom: 2 linear and 1 rotational or 2-degrees of freedom: 1 linear and 1 rotational) [Sarkar et al. (2004)] are modeled using a suitable spring-based suspension system. Surface pressures on the physical model and its motion in terms of displacement and/or acceleration are measured. These are used to extract the required aeroelastic parameters such as Rational Functions or Flutter Derivatives or buffeting parameters such as indicial functions or admittance functions that describe the aerodynamic/aeroelastic loads (fluctuating lift, drag, etc.) on the structure. The full aeroelastic models can then be used to validate the aerodynamic/aeroelastic load formulations based on the section-model based parameters that were extracted. The full aeroelastic model can be also used to validate the analytical formulation to predict its response (displacement, acceleration, stress) in the wind tunnel by taking into account the mechanical properties (stiffness and damping) of the model and using the aerodynamic/aeroelastic load formulation.

**Wind Load Effects**

**Storm Characteristics**

The characteristics of extreme wind events resulting from a windstorm influence the peak wind loads on a structure. For example, in tornadoes, the Swirl ratio of the vortex, its core diameter (size), maximum horizontal wind speed and how fast it moves or translational speed, are known to influence wind loads. In a microburst, its translational speed, the diameter (size) of the downburst and the height of the base cloud to some extent, are known to influence peak loads, and in a gust during a thunderstorm or hurricane, the acceleration of the flow and gust factor (ratio of maximum to minimum wind speed) could amplify the loads.

**Static Pressure**

The distributions of surface static pressure on the ground in a stationary microburst and a stationary tornado vortex are shown in Figure 19. The static pressure increases underneath a microburst while it decreases or drops inside a tornado with respect to surrounding atmospheric pressure away from the storm. The surface pressures on a building surface in these events is a combination of change in static pressure and the aerodynamic-induced surface pressures from the wind.
Terrain and Topography

Wind tunnel tests to study the influence of terrain and topography are not so common. It is important to understand their effects on the wind flow to assess their impact on the structures that are located in special regions surrounded by rough terrain and topography. It is now easier to study these effects as a result of advanced mechanical tools like a 3D router or a foam cutter and techniques like PIV or multi-hole pressure probes or anemometers. Two examples will be described here.

The flow field of a laboratory-simulated tornado was studied by means of 2-D Particle Image Velocimetry or PIV technique with the purpose of investigating the effects of roughness on the near-ground tornado-like vortex structure and its velocity distribution. The results showed the flow regime transition from multi-celled vortices over the smooth surface at a higher swirl ratio to single- or dual-celled vortices with roughness. Significant difference in velocities and turbulence characteristics was observed as a result of increasing ground roughness. A 1:3-scaled or Mini version of the ISU Tornado Simulator was used for this study. A smooth ground plane made out of fully transparent acrylic material was used to obtain the particle images from the bottom view. Three types of roughness elements were made out of foam to simulate the rough ground surface. The Mini Tornado Simulator and one of the Roughness foam elements used (0.5 inch cubes at a center-to-center interval of 2 inch) are shown in Figure 20. The effective roughness length \( z_0 \) for the roughness element was estimated as 0.400 mm. Figure 8 shows a schematic diagram of the PIV experimental setup. The details of the experiment and results can be found in Zhang and Sarkar (2008). Velocities in the tornado wind field were visualized and assessed on three near-ground horizontal planes at different elevations \( z = 11, 26 \) and 53 mm model scale; 3.3, 7.8, and 15.9 m full-scale with 1:300 length scale) and the meridian plane of the vortex (see Figure 20). The simulated tornado vortex was stationary with a Swirl ratio of 0.14 \( (z=11\text{mm}, \text{core radius 75 mm}) \). For both Smooth and Rough grounds, the horizontal velocity increases to its highest value at a certain distance from the vortex center and the magnitudes decrease with height in both cases. The central area with lowest velocity magnitudes gets smaller considerably over the rough surface. With the Rough ground the maximum horizontal velocity
decreased to 85% of that over the Smooth ground in the lowest measured horizontal plane (z=11 mm).

In the second example, a model section of a topography as shown in Figure 21 was used to study the flow field in a simulated tornado [Karstens (2012); Karstens et al. (2012)]. The goal of this work was to prove or disprove the hypothesis of terrain-induced flow channeling and related factors influencing it. An approximate 16 km² area of the Digital Elevation Model (DEM) was downscaled using the aerial extent of observed damage combined with the known range of vortex diameters that can be produced with the ISU Tornado Simulator. It resulted in a scale of about 1:500. The downscaled region was divided into smaller subsets to allow for precise reconstruction of the Earth’s surface from foam using a 3D wire foam cutter. Measurements were obtained in and near the valley channels of interest 25mm above the surface in a regularly spaced grid using the Omni probe. The velocity vectors, corresponding to the tornado vortex nearest to the case study region, show in Figure 21 that the flow is perpendicular to the valleys and flow speed-up on the ridges occur while the valleys have slower wind speeds.

Fig. 20 A Mini-Tornado Simulator at ISU and Roughness element used [top views]; Velocity flow field in a simulated stationary tornado on horizontal planes at three elevations [z=11, 26 and 53 mm, Scale 1:300] at a high Swirl Ratio above a Smooth (left) and a Rough (right) terrain (bottom views); Zhang and Sarkar (2008).
Building Parameters

Building parameters (geometry, orientation, shape, openings, surrounding buildings or structures) are known to influence wind loads on the buildings in straight-line wind. The same effects were found to be true for transient winds, tornado and microburst. Buildings that are located along the centerline of the wind event path experience the maximum wind speed and hence are most vulnerable to damage.

The microburst wind loading effects on a set of low-rise building models and a high rise building model were investigated [Zhang et al. (2013c)]. A microburst simulator at ISU that generates a steady-impinging jet was used (Figure 5). The steady impinging jet replicates a steady microburst-like phenomenon where the wind speed profile represents the one of maximum wind during the evolution of a microburst. The diameter of the nozzle (D) was about 0.61m (2 ft) and the distance between the nozzle exit and the ground plane (H) was set to 2 diameters of the nozzle (H/D=2). The models that were studied include a cube (45mm), a conical-roofed grain bin model (diameter 50mm), two gable-roofed building models (65x65mm plan; 16°, 35° roof angle), and a high-rise building model (45x45x180mm). Figure 22 presents the geometries of low-rise structural models used in this study as fabricated by a rapid prototyping machine. To compare the effects of different shaped roofs, the mean roof height of the grain bin model was kept the same as that of the gable-roofed models (36mm). The geometric scale was estimated approximately 1:700. At this scale, the gable-roofed and cubic buildings considered here approach the size of large industrial buildings or warehouses.

In Figure 23, it can be seen that the difference among all the low-rise models was not significant when they are at or near the center of the microburst. It implies that the static pressure, instead of the flow-structure interaction, has a greater contribution to the overall wind loads in this region. In the outburst region, i.e. r/D≥1.0, the drag coefficient of the cube was the largest among all building models. The 35-degree gable-roofed building model experienced a larger drag than its 16-degree counterpart due to the blockage effect of the steeper roof. Although the roof angle of the grain bin model was not the smallest [30 degree], it actually experienced the smallest drag among all the building models due to the circular cross-section. Figure 23 shows the distribution of wind-induced shear force with height of the high-rise building as a result of its radial (r) distance from the microburst center and a comparison of the shear force distribution at r/D=1/0 where maximum microburst wind occurs and the straight-line wind results obtained by
Lin et al. (2005) and Kim and Kanda (2010) in an ABL wind tunnel. Calculation shows that the bending moment at r/D = 1.0 in the present study is approximately 57% and 62% of that produced by conventional boundary layer winds simulated by Lin et al. (2005) and Kim and Kanda (2010), respectively, given the same wind speed. Since the microburst winds are usually more violent than normal ABL winds, the magnitude of the actual mean and fluctuating force would be expected to be more significant.

**Transient Effects**

Preliminary studies have demonstrated that there are differences between the load and response effects of non-synoptic transient winds and those of synoptic boundary-layer winds.
Previous studies on bridge aerodynamics and fluid dynamics pointed out an overshoot in aerodynamic/hydrodynamic forces due to a sudden change in wind/fluid speed [Shiraishi et al. 1982; Okajima et al. 1997]. Attempts to model the transient aerodynamics using full-scale pressure data on bluff bodies were made by Katsura (1997) and Tamai et al. (2001). The fundamental modeling of non-synoptic wind effects on structures has yet to be developed. The following examples illustrate and emphasize the importance of transient load and response effects.

A flat thin plate was used to understand the evolution of drag force during gusts. The square plate of dimension 211mm (8.3 in) and area 0.045m² (69in²; 1% blockage) was tested in the AABL Wind and Gust Tunnel at ISU in a smooth flow. The evolution of the normalized drag force time history (normalized with instantaneous velocity) on a flat plate is shown in Figure 24 in a “ramp-up” gust with $\Delta V = 5.40$ m/s (Initial $V=19$ m/s) and settling time $\Delta t = 3.29$ sec (1.65 m²/s). Using a length scale of 1/12 and a velocity scale of 1/1.5, this gust would correspond to an acceleration of 0.31 m/s² in full scale which falls in the range of 0.04 to 0.4 m/s² as described earlier. While the overshoot of the mean drag force is not observed here like some researchers, an increase in the fluctuating component of normalized drag at the higher mean velocity compared to the lower one is evident. The overshoot in drag was observed by others when a shutter-like mechanism was used in the wind tunnel to create a step-like sudden increase in velocity, usually with a settling time of less than $\Delta t = 0.1$ sec. It is unknown if the small $\Delta t$ used in the gust generation for creating this overshoot is comparable to gusts found in nature but if it does then this effect needs to be researched further. The overshoot phenomenon in gusts by others is explained using the modified Morison’s equation, which is typically used for denser fluids like water, where two terms represent drag and inertial forces and another term is used as a correction term because Morison’s equation does not fully characterize the drag force during this overshoot period.

In another experiment, a section model of a streamlined bridge deck section with a thin rectangular section and semi-circular fairings on both edges (B/D, B=total width and D=depth of 15:1) was used to test the effect of gust in the AABL Wind and Gust Tunnel at ISU [Cao and Sarkar (2013)]. The primary objective of this experiment was to verify whether the dynamic aeroelastic loads on a flexible structure and its response can be predicted in time domain. The vertical and torsional displacements of the two-degree-of-freedom model were measured by force transducers mounted at the end of the springs suspending the model. Aerodynamic pressures were also recorded in the experiment to assess aeroelastic loads for comparison with those obtained from numerical simulation. Surface pressures were measured on the model along its mid-plane using forty-two pressure taps equally distributed on both the surfaces. To synchronize the displacement data with the pressure data and the wind velocity data, both the pressure transducers and the Cobra Probe were set to work in an external-trigger mode. The section model was subject to a ramp-down gusty wind whose horizontal velocity time history is shown in Figure 25. The dynamic lift and moment coefficients were calculated by integrating the surface pressures measured and compared with the numerical values estimated by using a set of Rational Functions for this bridge section (identified separately) and the measured displacements. The lift coefficient time history in Figure 25 shows that the amplitudes of lift coefficient is predicted well in the first half of the time interval and are over-predicted in the later time period. It also shows that the amplitude of the lift coefficient (normalized by instantaneous mean velocity) grows with time. The same was observed for the moment coefficient.
Measurements of pressures on a cube under simulated downburst winds showed that the acceleration of the diverging flow significantly altered the wind pressures [Letchford and Chay (2002); Mason et al. (2005); Sengupta et al. (2008)], which cannot be explained utilizing quasi-steady theory. Haan et al. (2010) studied the integrated forces on a 1/100 geometrically-scaled model of a gable-roofed building under simulated tornados. The results showed that the tornado-induced loads were not maximum for the faster translation speed. The quasi-steady test, where the tornado was held at specific distances from the model while the forces were measured, produced higher or comparable values of base shear force and bending moment but lower value of torsional moment as compared to the transient unsteady tests. Figure 26 shows the transient loads on a cubic building due to a translating microburst (jet diameter $D$) and a translating tornado (core diameter $D$) where the building is located at $x=0$ [Sengupta et al. (2008)]. It is evident that a slower moving tornado produce larger peak roof uplift than a quasi-steady or a faster moving tornado.

**Fig. 24.** Evolution of aerodynamic drag force on a flat plate for a ramp-up gust

**Conclusion**

This paper discusses some of the latest advances in wind tunnel simulation of extreme wind phenomena that poses wind hazard to civil structures and emphasizes the importance of transient wind phenomena in this context. In the past decade and a half, the following developments have taken place in wind tunnel simulation and modeling: new wind tunnels that can physically simulate winds in gust front, tornadoes and microburst for wind engineering have emerged; larger wind simulation facilities that can address Reynolds number and small-scale turbulence effects, which were limitations of most ABL wind tunnels, have been constructed and serviced; new techniques such as rapid prototype and 3D routers for model building are available; faster and more accurate instrumentation are available for data acquisition; and improved tools have evolved such as better and faster CFD methods to complement laboratory
Advances in Wind Tunnel Simulation, Techniques and Tools for Assessing Extreme-Wind Hazard to Structures

simulations, numerical techniques for non-stationary wind simulation, finite element method that can be iteratively used with laboratory simulations, spectral and probabilistic methods that can be applied to dynamic data measured in wind tunnels to extract the relevant parameters for predicting wind hazard, and system identification algorithms that can directly extract parameters used in aeroelastic force models. Important parameters influencing wind loads such as characteristics of the storm, terrain and topography, geometry of the structure, orientation of the structure, interference from surrounding structures, and internal pressures that are relevant to wind loads in straight-line wind have been also found to be relevant for transient winds such as in tornado and microburst.

Fig. 25. Horizontal velocity time history and evolution of dynamic lift coefficient for ramp-down type of gusty wind [Cao and Sarkar (2013)]

Fig. 26 Normalized roof uplift for a 25.4mm cubic building in a translating microburst (left); and translating tornado showing effect of translation speed - QS: quasi-steady, LS: low-speed, HS: high-speed (right); Sengupta et al. (2008)
Acknowledgement

The author acknowledges the contributions of all his collaborators and students to the work presented here. This work would not have been possible without the financial support from the sponsors NOAA and NSF (USA) during the 2002-2012 period.

References


ANSYS (2009), Academic research 12.1, Help System, ANSYS Inc.

ASCE 7-05 (2005), Minimum design loads for buildings and other structures. ASCE, NY


Garthshore, I.S. (1973), The Effects of Free Stream Turbulence on the Drag of Rectangular Two-Dimensional Prisms, Boundary Layer Wind Tunnel Laboratory, 4-73, University of Western Ontario.


Holmes, J.D. (1978), Mean and Fluctuating Internal Pressures induced by Wind, Wind Engineering Report 5/78.
Department of Civil and Systems Engineering, James Cook University of North Queensland, Australia.


