Physical and numerical modeling of downburst generated gust fronts

Kyle Butler[a], Ahsan Kareem [a]
[a]*NatHaz Modeling Laboratory, University of Notre Dame, Notre Dame, IN, USA*

ABSTRACT: This paper presents three different schemes for modeling and analyzing the effects of gust fronts resulting from downbursts on structures. These schemes include: (i) the utilization of a manually controlled, pivoted flat plate that can be rotated to a high angle of incidence in a traditional low speed wind tunnel facility to mimic the kinematics and dynamics of a gust front and to quantify their effects on a series of prismatic models, (ii) simulation of flow past an inclined flat plate using CFD and (iii) generation of a gust front flow field in a multiple fan wind tunnel facility containing individually controlled fans. In these experimental configurations, the inflow profile is tailored to produce an inverted profile, characteristic of outflows from downbursts where velocity maxima is achieved nearer to the ground surface, while also modeling the transient dynamics of a gust front. Results demonstrate that the desired velocity profile can be reproduced within the conventional wind tunnel framework. Pressure measurements on a series of prismatic building models demonstrate the effect of the transient nature of gust front behavior and the subsequent change in pressures over a typical boundary layer flow, identifying key features for future investigations.

KEYWORDS: downburst; gust front; CFD; flat plate at high incidence

1 INTRODUCTION

Thunderstorms, downbursts and their associated gust fronts constitute various extreme wind events that cause significant damage to life and property, particularly to low or mid-rise buildings, transmission lines, industrial structures and, potentially, long span bridges. Some characteristics of thunderstorms, downbursts and gust fronts are similar, primarily their respective horizontal wind velocity profiles near the ground surface. These winds are transient in nature and are not as easily quantified as is the typical synoptic boundary layer. Additionally, the flow conditions associated with these events potentially introduce major changes in flow-structure interaction that differ significantly from the corresponding boundary layer flows (Kareem 2005). Full scale data regarding the behavior of these events is limited, and the data that does exist only captures the features of particular interest to the observer. For example, the full scale data presented in this investigation regarding downbursts for which the experimental data is compared against data gathered to study the related impacts on aircraft (e.g. Hjelmfelt 1988).

The simulation of downbursts and gust fronts has been attempted through various physical and numerical techniques. Among these are the use of impinging jets (e.g., Wood et al. 2001; Chay and Letchford 2002), large vortex generators (e.g., Sarkar et al. 2006) and wall jets (e.g., Lin and Savory 2006), along with similar physical configurations modeled utilizing various CFD schemes. However, traditional wind tunnel facilities are numerous and offer potential for the simulation and study of the effects from gust fronts with minimal modification to the wind tunnel set-up used to study typical boundary layer flows. Experimental work mapping the boundary layer resulting from wall jets (e.g., Launder and Rodi 1981) has shown that such modifications to existing facilities are possible.

This paper presents two physical modeling techniques and one computational model to simulate downburst generated gust front wind speed profiles, in an effort to capture both time-averaged and transient features. The first technique is the use of a manually controlled, pivoted flat plate adjustable to a high incidence into a bounded flow field, representing a 2D simulation of a gust front flow profile. The nature of the resulting flow is similar, in some respects, to a wall jet. This experiment is simulated both physically in a wind tunnel, and numerically using a packaged CFD program. The second technique involves the use of an actively controlled series of fans, which can be controlled either individually or collectively, to actively tailor the desired flow profile. This is achieved through the development of a transient flow field simulator at the NatHaz Modeling Laboratory, as well as Miyazaki University in collaborative efforts with the Center of Excellence (COE) on Wind Effects in Urban Areas at the Tokyo Polytechnic University. Both techniques provide an effective method for generating the desired flow profile and a suitable environment for testing the effects on scale models.

2 EXPERIMENTAL SETUP

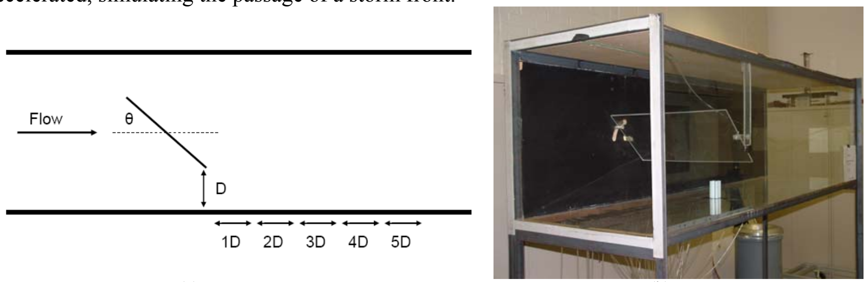
2.1 *Gust front generation using a flat plate at high incidence*

The concept of suddenly introducing a flat plate at a high incidence to the oncoming flow to generate a gust-front-like profile, similar to that of an impinging jet or a wall jet, is to force the flow to accelerate near the defined ground surface. Similar to the dynamics of a wall jet, the flow exiting the vicinity of the flat plate evolves into the desired profile shape as a result of the convective nature of the existing flow conditions.

Much of the research regarding the flow around flat plates at high incidence (e.g., Stahl and Mahmood 1985) has covered angles of attack up to 30º, with some studies up to 50º. Though the aim of previous work was not specifically to asses the resulting changes to the boundary layer flow, they have provided insight into the behavior of the flow when encountering such an obstruction. Additionally, the flat plate experiment considered here resembles a larger scale wall jet experiment, where the aim is to add momentum to the lower portion of the boundary layer flow. Details of various experiments investigating the nature of wall jets are presented in Launder and Rodi (1981).

In the experiment conducted in the Hessert Aerospace Research Laboratory at the University of Notre Dame, a flat plate was positioned within a section of a low speed wind tunnel. The plate was able to rotate through a full 90º, with θ=0º corresponding to the flat plate horizontal to the approaching flow and θ=90º corresponding to the flat plate perpendicular to the oncoming flow, imposing up to 50% blockage of the initial flow region. The subsequent change induced by the flat plate at a high incidence was that the flow is forced to quickly accelerate in the regions above and below the plate, recreating a gust front type flow condition in those regions. The angle of incidence of the flat plate was changed throughout the course of the experiment, to vary the outflow opening and outflow wind velocity, mimicking the transient conditions of an approaching thunderstorm or downburst. Figure 1a shows the general setup of the flat pate in a conventional modular wind tunnel section, with important dimensional parameters indicated (where D is the height of the opening and θ is the plate angle), and Figure 1b shows the actual assembly. Considering the nature and framework of this experiment, new scaling and non-dimensionalization schemes will be required to experimentally match the simulated gust front with full scale data.

Experiments using the flat plate gust front simulator have involved assessing pressures and transient behaviors on various prismatic models within the simulated gust front flow regime. The models were placed downstream of the flat plate, within the region of accelerated flow, while the flat plate was manipulated to simulate storm events. Three models were constructed to asses the affect of the location of maximum horizontal outflow. The location of the model was changed to investigate the response at various proximities to the passing storm front. Pressure taps on the surface of the model recorded pressure changes as the flow was accelerated and decelerated, simulating the passage of a storm front.



(a) (b)

Figure 1: (a) Flat plate experiment nomenclature and (b) experimental setup.

2.2 *Gust front generation using CFD*

Considering that computers have increased in both computational speed and storage capacity in recent years, computational fluid dynamics (CFD) is becoming a more attractive alternative to simulating wind effects. Recent work utilizing the commercial CFD package FLUENT has demonstrated its ability to model particular aspects of the flow field and simulate effects on structures (Chay et al. 2006, Kim et al. 2005). The high incidence flat plate experiment was modeled in FLUENT to visually identify features in the flow field that are not easily discernable from wind tunnel experiments, and to assess its usefulness in examining the transient nature of the simulated events. The CFD model provides an effective method for estimating the extent of accelerated flow in the region of interest, and the location of other flow features that may need to be enhanced or retarded during the course of experimental development.

2.3 *Gust front generation using a multi-fan simulator*

The active generation of a gust front flow profile is performed using a multi-fan wind tunnel simulator. The aim of this experiment is to generate a relatively larger scale gust front wind speed profile for further experimentation. The recently completed prototype apparatus involves a set of four, independently controlled fans.

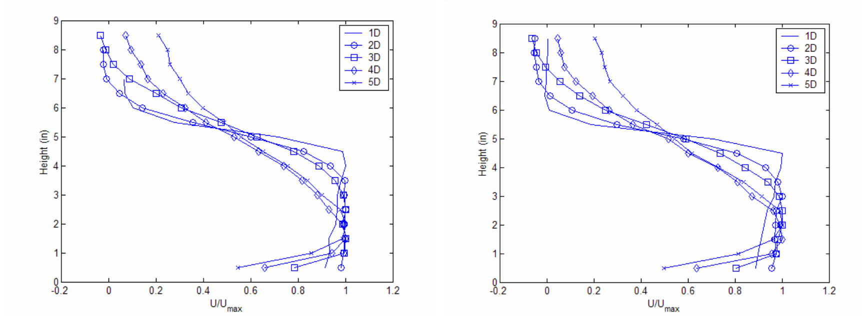
This particular type of wind tunnel provides for fine tuning of the gust front profile. Parameters that can be tuned involve significant control of transient features and control of turbulence intensity, compared to the case of a flat plate at high incidence. A similar tunnel at Miyazaki University employs a larger battery of fans to generate desired wind parameters (Cao et al. 2002). The NatHaz Laboratory, in collaboration with the Center of Excellence (COE) on Wind Effects in Urban Areas at the Tokyo Polytechnic University, is investigating the role of transient aerodynamics in the formation of loads on basic surface mounted prisms. Experimental results based on this facility will be compared against current models for validation and will be used to identify behaviors of structures under transient wind events.

3 RESULTS

3.1 *Profile development using a flat plate at high incidence*

Preliminary results from ongoing experiments with the flat plate at high incidence are shown in Figure 2 through 4. The simulated gust front profile, compared with published full scale data (Hjelmfelt 1988), shows good agreement based on current scaling and non-dimensionalization methods. Additionally, numerical results from the CFD computer simulation, using the FLUENT computer program, demonstrate agreement with the experimentally simulated gust front profile.

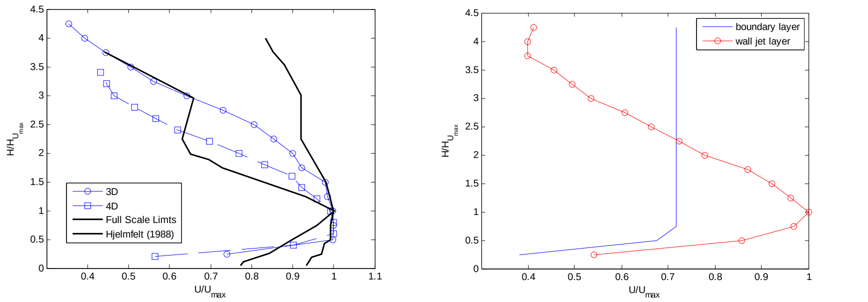
The development of the simulated gust front profile was mapped within the vicinity of accelerated flow in the lower portion of the wind tunnel section. The boundary layer profile was measured at various locations downstream of the outlet, from 1 to 5 outlet heights, where the outlet is defined as the point at the edge of the plate. Additionally, the plate angle was varied between 30º and 50º to quantify the effect of varying plate angle on the development of the gust front profile. Figure 3a shows the flow development of the profile with the plate angle established to be 45º and Figure 3b shows similar development with the plate angle at 50. Figure 3a shows flow development with the flat plate angle at 30º compared to full scaled data when scaled appropriately.



(a) (b)

Figure 2: Profile development beyond a flat plate at (a) 45º incidence and (b) 50º incidence.

The results show that the generated wall jet resembles a gust front profile, with an accelerated region nearest to the lower portion of the boundary layer. The flow nearest the flat plate outlet, 1 outlet height, does not exhibit the nature of the gust front flow profile, as the flow is fairly uniform in its profile. As the flow develops downstream of the flat plate outlet, the profile begins to resemble that of the gust front due to the convective nature of the free stream. Based on the results for the incident angles of the flat plate tested, it is estimated that the flow achieves the desired profile beyond 3 outlet heights. This behavior of flow development appears to be independent of the incident angle. The only dependent features of the flow are the velocity of the outflow profile and the rate of decrease of flow with increasing distance from the test surface.



(a) (b)

Figure 3: (a) Profile development beyond a flat plate at 30º incidence and (b) boundary layer profile comparison during simulated storm event (4D with the flap at 30º incidence).

The evolution of the boundary layer generated before and after the introduction of the flat plate was also evaluated, demonstrating the change in flow conditions from a traditional boundary layer flow to the simulated gust front profile. Because of the nature of the experimental setup, the gust front is not generated impulsively; rather it is generated through modifying the existing boundary layer. The estimated initial flow condition within the wind tunnel is similar to that of an atmospheric boundary layer profile. When the flat plate is suddenly set to some angle, the wall jet profile develops in time. Figure 3b shows preliminary experimental results in mapping the changing nature of the boundary layer condition within the wind tunnel.

3.2 *Transient pressure measurements on model prisms*

The passage of a simulated storm was generated by controlling the movement of the flat plate from a position horizontal to the flow to some preset angle. Several prismatic building models were immersed within the accelerated flow region of the tunnel, and the proximity of the model to the simulated storm front was adjusted during the experiment to coincide with the developed region of the simulated gust front. The goal was to place the building at the location where the profile best resembled that given by full scale experimentation. Three models were constructed for this experiment to estimate the effects on the pressures on a structure when the roof level is below, at, and above the location of maximum outflow. The were three prismatic models tested, one with aspect ratio 1:1 and two with aspect ratio 2:1, varying in height to adjust the location of the roof level with respect to the height of maximum outflow. Figure 4a-c shows experimental results of the estimated pressure coefficients around each of the three models, where the pressure coefficient is defined as $C_p^*(t)=(p(t)-p_\infty)/(1/2\rho U(t)^2)$. The results reveal a snapshot of the pressure change concomitantly with changes in flow conditions, revealing the transient nature of the pressure field.

Similar to the formation of a horseshoe vortex at the base of a prism in a boundary layer, due to shear in the flow, it is anticipated that a vertical structure would emerge in the region above the maxima of velocity profile due to reductions in velocity resulting in conditions conducive to the formation of another horseshoe type vortex, where the vortical filament wraps around the top portion of the prism, as sketched in a flow caricature in Figure 4d. Its exact influence on the flow field will depend on several flow features with respect to prism height. Therefore, one can anticipate changes in the aerodynamics of a prism in a gust front flow as compared to a boundary layer which manifests from the difference in both flow kinematics and dynamics associated with the transient nature of the flow. Results in Figure 4a-c provide a glimpse of the changing trends in pressure distribution when compared to boundary layer flows. The current investigation in different simulated flow platforms would lead to unveiling the potentially discernable differences in pressure and force coefficients in gust fronts.

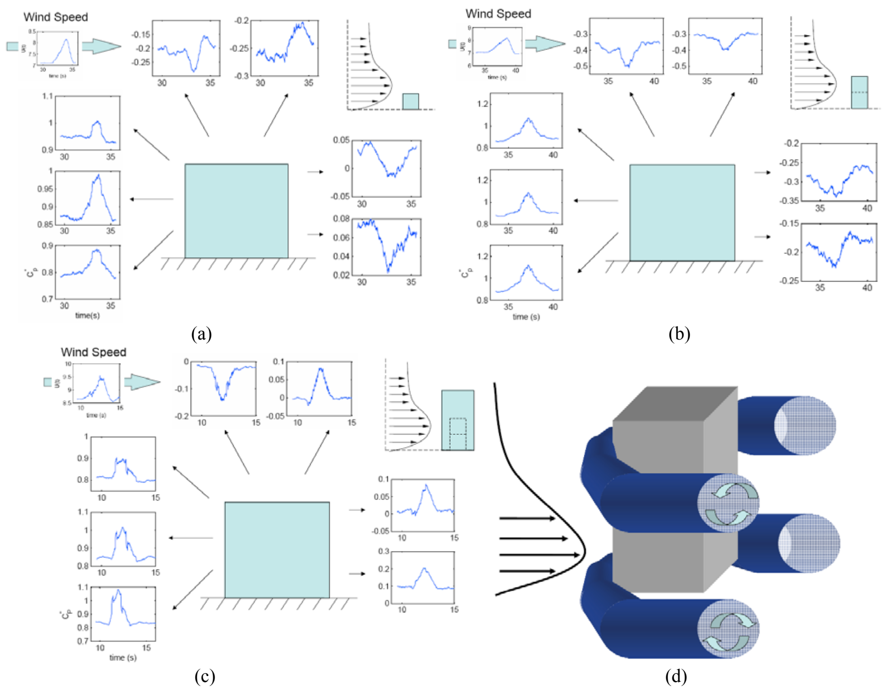


Figure 4: Pressure coefficients on a prismatic model of aspect ratio (a) 1:1 (b) 2:1 and (c) 2:1x2 during a simulated gust front event, and (d) a caricature of the flow development around a prismatic model.

3.3 *Gust front modeling using CFD*

The use of computational fluid dynamics (CFD) has provided a simpler means of testing and verifying the effects of flows on structures. Recent advancements in the development of CFD codes has allowed for the implementation of transient motions of solid bodies within dynamically changing computational grids. Dynamically changing computational grids involves the on-the-fly re-meshing of the computational domain as a solid body moves in time (FLUENT 2006.). The use of dynamically changing computational grids is currently under investigation and validation studies due to its recent development. Figure 5 shows experimental results of the flat plate experiment utilizing the dynamic meshing feature of FLUENT along with a snapshot of the profile in time compared with full scale data. The commercial CFD package, FLUENT, is utilized to simulate a flat plate rotating from a horizontal position to a set incline. The viability of this application is currently under investigation, particularly investigating its ability to recreate the transient changes in the boundary layer flow. The wind tunnel studies have provided a benchmark for this study.

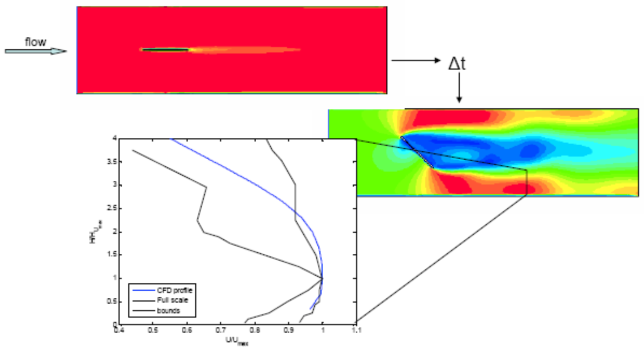
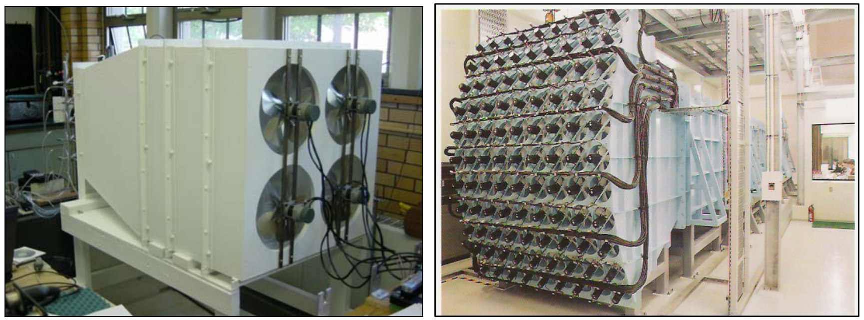


Figure 5: CFD model of the flat plate experiment utilizing a dynamic mesh to model the plate movement and an examination of the wind speed profile compared with full scale data.



(a) (b)

Figure 6: (a) The prototype transient flow field simulator and (b) multiple fan wind tunnel at Miyazaki University.

3.4 *Preliminary results using the multi-fan simulator*

The multi-fan simulation facility is another method under investigation to provide experimental insight into the nature of transient flow effects on structures. Ongoing experimentation with this facility is garnering new insights into the nature of modified flow fields. The NatHaz Laboratory is currently collaborating with the Center of Excellence on Wind Effects in Urban Areas at the Tokyo Polytechnic University, investigating the nature of transient aerodynamics on the formation of loads on basic surface mounted prismatic shapes. Figure 6a shows the wind tunnel in its operational setup. This facility has been involved in numerous tests to establish its usefulness in reproducing the atmospheric boundary layer and turbulence characteristics based on prescribed power spectral densities (e.g. Ozono et al. 2006). A suite of experiments using a finite height prism has recently been completed and their results will be discussed during the presentation.

4 CONCLUSION

The aim of this work is to analyze various methods of generating wind flows related to extreme wind events, examining the many features observed to be characteristic of gust fronts, producing these on a laboratory scale and adding to the growing body of literature. Two such approaches involve the use of a flat plate at a high incidence to the approach flow, which can be actively controlled to simulate transient events, and a multiple fan wind tunnel apparatus. In the case of the flat plate wall jet, the boundary layer profile was mapped to examine the changing conditions within the wind tunnel test section. Ongoing experiments utilizing a constructed facility containing independently controlled fans show promising results in examining transient behaviors of flows. Both of these simulation methods provide a reasonable approach to the modeling of gust front winds, especially in providing a reasonable scale for conducting test arrangements with model buildings within traditional frameworks. Additionally, transient features can also be modeled within these frameworks. Experimental results of simulated gust front profiles demonstrate good agreement with full scale data and other experimental approaches presented in the literature, while the characteristics of the flow field around prismatic building models subjected to a simulated gust front demonstrate the transient nature of the pressure coefficients. Continued work seeks to provide an acceptable method for testing the effects of gust front winds on simulated buildings and environments, both physically and numerically, with the goal of utilizing this information to aid in the reassessment of new codes and standards for gust front conditions.

5 ACKNOWLEDGEMENTS

The authors gratefully acknowledge the support provided in part by NSF Grant CMS 03-24331, as well as Prof. Yukio Tamura, Director COE, TPU, and Prof. S. Cao of TPU for their collaboration, assistance and support for experiments at Miyazaki University.

6 REFERENCES

1 A. Kareem, Bluff body aerodynamics and aeroelasticity: a wind effects perspective, 4th European-Africa Conference on Wind Engineering, (2005)
2 M. R. Hjelmfelt, Structure and life cycle of microburst outflows observed in Colorado, Journal of Applied Meteorology 27 (1988) 900-927.
3 G. S. Wood, K. C. S. Kwok, N. A. Motteram, D. F. Fletcher, Physical and numerical modeling of thunderstorm downbursts, J. Wind Eng. Ind. Aerodyn. 89 (2001) 535-552.
4 M. T. Chay, C. W. Letchford, Pressure distributions on a cube in a simulated thunderstorm downburst. Part B: moving downburst observations, J. Wind Eng. Ind. Aerodyn. 90 (2002) 733-753.
5 P. P. Sarkar, F. L. Haan, V. Balaramudu, A. Sengupta, Laborators Simulation of tornado and microburst to assess wind loads on buildings, ASCE 2006 Structures Congress (2006) St. Louis, MO.
6 W.E. Lin, E. Savory, Large-scale quasi-steady modeling of a downburst outflow using a slot jet, Wind and Structures 9 (2006) 419-440.
7 B.E. Launder, W. Rodi, The turbulent wall jet, Prog. Aerospace Sci., 19 (1981) 81-128.
8 W. H. Stahl, M. Mahmood, Some aspects of the flow past a square flat plate at high incidence, Z. Flugwiss. Weltraumforsch. 9 (1985) 134-142.
9 S. Cao, A. Nishi, H. Kikugawa, Y. Matusda, Reproduction of wind velocity history in a multiple fan wind tunnel, J. Wind Eng. Ind. Aerodyn. 90 (2002) 1719-1729.
10 Fluent 6.3 Documentation, Fluent, Inc. 2006
11 S. Ozono, A. Nishi, H. Miyagi, Turbulence generated by a wind tunnel of multi-fan type in uniformly active and quasi-grid modes, J. Wind Eng. Ind. Aerodyn. 94 (2006) 255-240.