



ESCARPMENT STUDY IN A VIRTUAL FLOW ENVIRONMENT A COMPARATIVE ANALYSIS OF A SINGLE BUILDING TYPE MODELED IN VARYING TOPOLOGICAL SITUATIONS

Michael W. Kuenstle, AIA

School of Architecture, University of Florida, USA
kuenstle@ufl.edu

Abstract

This paper documents the progress of research to investigate the integration of 3-dimensional computational modeling techniques into wind mitigation analysis and design for building structures located in high wind prone areas. Some of the basic mechanics and theoretical concepts of fluid flow and wind pressure as well as their translation into design criteria for structural analysis and design are reviewed, followed by a discussion of a detailed Computational Fluid Dynamics (CFD) application case study for a simulated "3-second gust" hurricane force wind flow over a low rectangular building located in a coastal region of south Florida. The case study project models the wind flow behavior and pressure distribution over the building structure when situated in three varying conditions within a single terrain exposure category. The simulations include three-dimensional modeling of the building type constructed (1) on-grade in a flat coastal area, (2) above grade with the building elevated on structural columns, and (3) on-grade downwind of an escarpment. The techniques and parameters for development of the simulations are discussed and some preliminary interpretations of the results are evaluated by comparing their predictions to existing experimental and analytical data, with special attention paid to the numerical methods outlined in the American Society of Civil Engineers, *Minimum Design Loads for Buildings and Other Structures*, ASCE 7-98.

1. Introduction

The analysis and prediction of atmospheric wind and its interaction with building structures continues to be a subject of intense research. This can be attributed to many factors which, in part, involve the emergence of a plethora of new building material science technologies and systems dedicated to improving our understanding of the diverse and complex web of environmental concerns related to the design and construction of our built environments. Among this broad range of environmental concerns, the study of the behavior of building structures subjected to hurricane force winds is of great importance to architects, engineers and city planners. In the United States alone, for example, the Federal Emergency Management Agency (FEMA) reported that in 1997 there were more than 45 million permanent residents located along the U.S. hurricane-prone coastline stretching from the gulf coast of Texas to the Carolinas. Recent data from a FEMA survey conducted in 2001 also suggests that despite a heightened awareness of the potential hazards associated with tropical storm phenomena, namely damage to life and property caused by storm surge and high winds, the population in these vulnerable areas continue grow at a rapid pace.

Since the enormous and costly devastation caused by hurricanes Andrew and Iniki in 1992, architects, engineers and the construction industry have learned important lessons regarding improved mitigation techniques for wind resistant construction. Field observations of damaged structures and building components, advanced research in the field of wind engineering, increased accuracy and reliability of meteorological data, and subsequent continued upgrades to building codes and standards including the American Society of Civil Engineers (ASCE 7-98), *Minimum Design Loads for Buildings and Other Structures*, have all played a key role in increasing the safety level of building structures sensitive to wind. Today for example, all compliant new construction in high

wind prone areas implement structural systems that incorporate the strategic use main wind force resisting frame systems constituted by diaphragms, shear walls, collectors, rigid and braced frames; all in combination with a detailed focus on component connections to transfer wind loads to the building foundation. Yet in structural terms, the fact still remains that even in the analysis of a very simple building, the dynamic loads associated with a probabilistic event such as a tropical windstorm can be sudden, complex, and unpredictable. Moreover, it is a well known fact that these loads are often characterized by rapid changes in magnitude, direction and distribution over a given structure making the design criteria for the expected structural behavior difficult to recognize (Schodek, 2001).

As a possible method for improving the understanding of wind phenomena and its dynamic interactions with the built environment, the following case study project was initiated to explore the potential application of Computational Fluid Dynamics (CFD) technology to simulate the behavior of wind flow and pressure distribution patterns developed over building structures when subjected to hurricane force winds. The simulation studies generated with the project have evolved within the framework of the commercial CFD code, Phoenix 3.4 with the implementation of a "standard" k- ϵ turbulence model. While there is great interest and value in the flow visualizations generated with this emergent and promising technology, it has also been a key objective of the study to schematically develop well-posed problems based on established governing principles. In this particular application it was also determined at the commencement of the project that geometrically precise and numerically homogeneous schemes would allow for more convincing comparisons of the simulation results with existing data and methods. Also it is worth noting that while the ASCE 7-98 presently makes no direct reference to methods for determining



the wind load effects on building structures using the CFD method, the nomenclature and symbolic notation used in this study is consistent with that found in the ASCE 7-98 which will be more familiar to those involved in the building design disciplines. Finally, in an attempt to establish a theoretical foundation for the initial case study applications, some documentation of the preliminary groundwork developed for the study is provided to assist in defining the general perspective and scope of the investigation.

2. Statics, Fluid Mechanics and Wind Engineering

In an effort to promote a better understanding of wind as a dynamic load source, an outline of a few basic principles regarding the behavior of wind as developed in fluid mechanics and applied in statics for building structures can be appreciated. First, in the analysis and design of structural systems and components for buildings, engineers and architects are primarily concerned with two general classifications of loads acting on a structure, static loads and dynamic loads. In engineering practice, static loads and their resultant stresses and strains are, for the most part, considered highly predictable in character and can be computed with a great degree of confidence. Dynamic loads, on the other hand, are load sources generated by probabilistic events and involve motion in the delivery of an energy load to the building structure (Ambrose, 1995). The two primary conditions under which buildings are subject to dynamic loading are during seismic and windstorm events. In the later condition, which is the focus of this discussion, the dynamic loads associated with wind flow, as previously mentioned, can be hard to pin down. Examples of this elusive behavior can be observed in a wide range of situations varying from a brief gust of high wind associated with a seasonal weather storm to a turbulent flow of wind caused by vortex shedding of an adjacent building. Adding to the complexity of predicting flow behavior at the atmospheric boundary layer where buildings are situated is the potential influence and interaction of the many variables that must be taken into consideration in wind load analysis. These include wind speed, direction and probability of occurrence, adjacent topographic features (natural and man-made), building height and building geometry. All must be considered.

Further detailed definitions of wind phenomena, its physical properties and behavioral attributes can be found in fluid mechanics. From text on fluid mechanics, for example, we know that the fundamental behavior of a fluid regime follows the laws of conservation for mass, momentum, and energy as well as the basic principles of Newtonian physics extended from solid mechanics (Albertson, 1960). In this context, wind is fundamentally defined as a moving fluid. The fluid in question can be more precisely defined by the specific physical properties of air with a given mass density, temperature, viscosity, and its rate of flow at a determined velocity through some known domain with assigned physical boundaries. Additionally, as the fluid stream of air interacts with each of its physical boundary elements, some of the stream flow is deflected producing a force, referred to as dynamic pressure, which is applied to the surface of the boundary element.

The point of application of the dynamic pressure (force) acts normal (perpendicular) to the surface and its direction can be either toward the surface or away from it (Albertson, 1960). The magnitude of the dynamic pressure generated from the fluid flow is derived from the potential energy of the kinetic energy ($E=ma$) of the fluid, in this case moving air, as summarized by application of the well-documented Bernoulli equation for fluid flow, which yields the expression :

$$q = 1/2 \rho V^2 \quad (\text{Eq. 1})$$

where q is the resultant dynamic pressure of the potential energy, ρ is the mass density of the fluid, and V is the velocity vector of the fluid flow. We also know from fluid mechanics that when a fluid stream flow parallels a boundary element, the surface of the boundary element will retard the flow of the fluid due to friction caused by shear stresses developed between the fluid media and the adjacent surface (Albertson, 1960). The amount of deceleration (drag) to the flow stream near the boundary is directly related to the roughness of the boundary's surface and the viscosity of the fluid. In this study the boundary surface in question is defined by the topographic features at the ground plane and the viscosity of air is defined by a dimensionless "Reynolds" number which relates the internal stresses to the viscous forces inherent in the fluid. Figure 2 graphically illustrates the profile of this behavior which can be computed by application of the power-law scheme as is commonly used in engineering practice for approximating specified atmospheric boundary layer conditions (Ward, 1999) summarized as

$$V = V_{ref} \left(\frac{Z}{Z_{ref}} \right)^{1/\alpha} \quad (\text{Eq. 2})$$

where V_{ref} is the reference velocity, Z_{ref} is the distance from the boundary corresponding to the reference velocity, Z is the distance from the boundary corresponding to velocity V , and α is the power law exponent related to the roughness for a given exposure condition. The α exponents used throughout this study are those found in the ASCE 7-98, which were adopted from A. G. Davenport's pioneering research involving wind flow behavior in the atmospheric boundary layer. Figure 2.

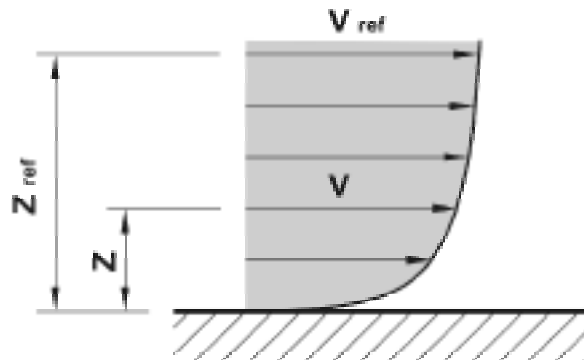


Fig 2 - kuenstle_Atmospheric wind velocity profile

The Davenport exponents are given for four different general exposure categories; A. large city centers, B. suburban residential areas, C. open terrain with scattered obstructions, and D. shorelines of inland waterways. The analytical method delineated in the ASCE 7-98 uses exposure category C for hurricane prone regions.

Finally and perhaps the most important principle from fluid mechanics related to the study of any flow regime is the principle that links fluid pressure with velocity along 2D flow streams. Numerically expressed as a derivative of the Bernoulli equation, and likewise referred to as the Bernoulli effect, the principle fundamentally states that there is a very simple and direct relationship between the fluid pressure and velocity at one point and the fluid pressure and velocity measured at another point along a 2D stream flow - specifically that the pressure plus the kinetic energy of the fluid at the first point equals the pressure plus the kinetic energy of the fluid at the second point (Ward, 1998). In



other words, due to the laws of conservation of energy, as the velocity increases or decreases along its flow path, its corresponding pressure will decrease or increase to create a form of fluid equilibrium. This relationship can be summarized by the following expression;

$$P_1 + 1/2 \rho V_1^2 = P_2 + 1/2 \rho V_2^2 \quad (\text{Eq. 3})$$

and finds many applications in fluid dynamics. A significant attribute of CFD codes is their ability to extend these principles into a 3D domain in which the time averaged Navier-Stokes equations can be solved using computer algorithms for flows over bluff bodies. Figure 3.

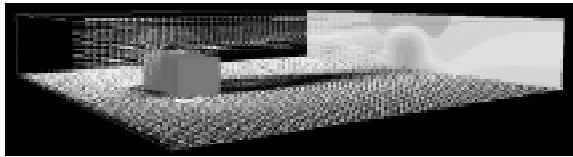


Fig 3 - kuenstle_3D Flow Simulation over a bluff body implementing a time averaged Navier-Stokes equation.

3. The Computational Fluid Dynamics Approach and Flow Visualization

Wind engineering research is a diverse topic with much attention in modern fluid mechanics. It is traditionally conducted using a variety of established methods including numerical analysis, full-scale measurements, and wind tunnel experiments. Over the past few decades, however, the analysis of fluid motion and flow structure using CFD simulation modeling techniques have been developed and applied with considerable success in some areas within the discipline. Working with CFD simulations, one is able to construct a visual window onto the dynamic, viscous, and bifurcating world of fluid media interactions. The visual simulation of this phenomenon is developed and approximated through space and time based numerical solutions of conservation equations in terms of fluid velocity and pressure for flows within a specified fluid flow regime. Some notable example applications can be found throughout the automotive and aerospace design industries where CFD technology is already fully integrated into design, testing, and manufacturing processes. More recently, important work with CFD has emerged in other fields such as biomedical and environmental research as well as marine related design work. With the rapid development of commercially available CFD codes in combination with access to more powerful computers, CFD applications are beginning to surface as a new virtual design tool in the building design disciplines and their related material science industries as the trend in design moves increasingly toward simulation rather than physical model experiments. Figure 4.

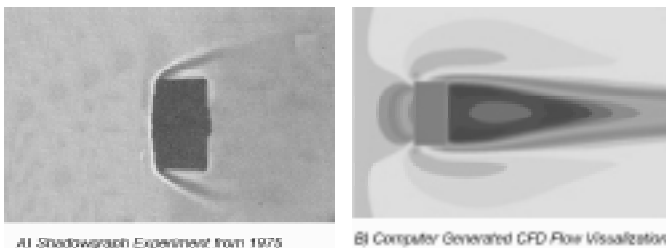


Fig 4 - kuenstle_Flow Visualization Techniques

In wind research on bluff bodies (which includes most building geometries), the application of CFD technology involves three primary elements: pre-processing, calculations, and post-processing (output for visualization). In brief, pre-processing includes some CAD model building and mesh generation techniques, a review of wind speed data, boundary and exposure conditions, and the determination of the physical and numerical properties for the flow regime. Among these components, mesh generation for the discretization of the flow domain is an important feature of the CFD process because it has a direct effect on both the speed and accuracy of the numerical solution. The mesh essentially divides the domain into a discrete number of cells or control volumes for which the partial differential equations can be solved by application of a numerical algorithm in an iterative process. Figure 5.

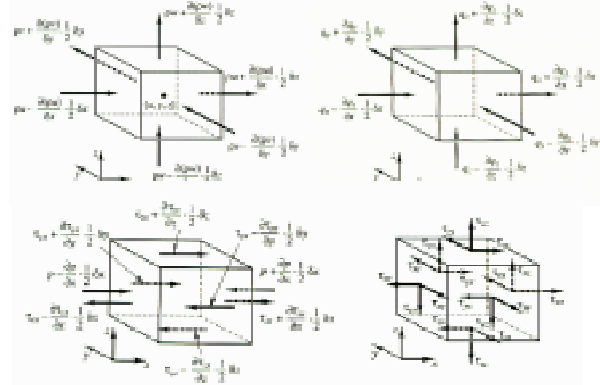


Fig 5 - kuenstle_Diagram of Governing Equations for a Fluid Flow, fluid elements or control volumes from Versteeg and Malalasekera, 1995

Concerning the calculations, the fidelity of the CFD solution for turbulent flows is dictated by turbulence modeling, especially when it comes to flows around buildings and other structures due to the complex features of the flow behavior (Kim 1999) as alluded to previously. Among the many turbulence models in existence, the “standard” $k-\epsilon$ turbulence model (Launder 1974) developed almost three decades ago is the most widely used and validated (Versteeg, 1995). While *ad-hoc* modifications to the “standard” $k-\epsilon$ model continue to be developed for special applications, the accepted use of the “standard” $k-\epsilon$ model is nearly universal within the CFD community. While a full description of the numerical model is far beyond the scope of this paper, it can be readily accessed in the referenced literature along with other examples of converged and stable solutions for schematically similar problems.

The final element of the CFD process involves the translation of the numerical data into graphic representations for flow visualization. The output data can take the form of 2D and 3D vector, surface, and contour plots, streamlines, and animations: some examples of which are included with this paper.

4. Case Study Project

The parameters of interest for the case study project include developing trial studies for the integration of 3-dimensional computational methods for simulating wind flow phenomena and building structure interaction. The dynamic forces acting on building structures due to wind loading must be determined for the appropriate design of the structure’s main wind-force resisting system and components and cladding. The study presented with this paper documents a CFD application for a simulated “3-second



gust” hurricane force wind flow over a low rectangular building measuring 12m x 6m x 5m and located in a coastal region of south Florida. Figure 6.

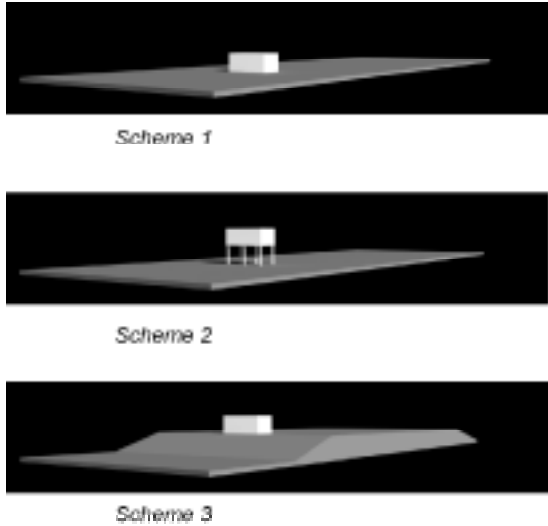


Fig 6 - kuenstle_Building Schemes

The case study project models the wind flow behavior and pressure distribution over the building structure when situated in three varying conditions within a single terrain exposure category. The simulations include three-dimensional modeling of the building type constructed (1) on-grade in a flat coastal area, (2) above grade with the building elevated 5m on structural columns, and (3) on-grade downwind of a 5m high escarpment. A basic premise of the study was the assumption that building structures located in hurricane prone regions are vulnerable to both storm surge and high winds and that common strategies to mitigate storm surge, i.e. raising the building to higher elevations as in scheme 2 and 3, subject the building structure to higher wind loads. Figure 7.

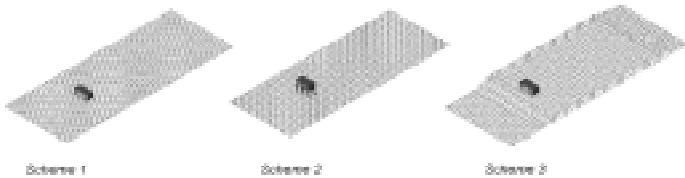


Fig 7 - kuenstle_Mesh generation

As an example CFD study for the schematic development of the parameters for the computational domain, the “Shah and Ferziger” solution for a fully developed turbulent flow over a wall mounted cube (Ferziger, 1999) was reviewed as a starting point and advanced with a previous study (Kuenstle 2001). The final domain parameters and placement of the building structure within the domain were determined after several trial study applications with the CFD solver. In an attempt to generate a more economical study, the overall domain parameters for the three schemes were kept constant. The building models and mesh generation were developed in Form-Z using a structured mesh, and then exported as stereo lithography (.stl) files for integration into the flow domain. The simulations are each single-phase flow, implementing the “standard” k- ϵ turbulence model, and are converged after 10,000 iterations. The attributes of the boundary conditions for the buildings were determined within the Phoenix software using a “solid with smooth wall friction” function. The number of cells for the three

schemes varies between 280,000 and 290,000. To determine the inlet velocity, a “3-second gust wind speed” of 63 m/s (140 mph) was selected from the “Basic Wind Speed” map, figure 6-1b of ASCE 7-98 which corresponds with the southeast Atlantic coastal region of Florida. A wind velocity profile was determined using the power-law scheme (Eq. 2), with the α exponent 9.5 for category C as specified in the ASCE 7-98, then input into the software to study the development of the flow and its behavior with the domain boundary prior to incorporating the building into the simulation model. A fully developed wind velocity profile was achieved as demonstrated. Figure 8.



Fig 8 - kuenstle_Wind Velocity Profile.

5. Observations

The primary sensitive issue that emerged from the trial results relating flow development, domain geometry, and mesh size to convergence of the governing equations involved a fine tuning of the placement of the building structure relative to the velocity inlet and outlet. Full development of the velocity profile was required windward of the building and could only be determined through preliminary testing. Some documented guidance by Versteeg and Malalasekera was relied upon for location of the outlet, “as the velocity profile downstream of an object can greatly affect the accuracy of the numerical results” (Versteeg, 1995). Additionally, as the original premise for the study is based on the concept of a “gust wind,” it was determined that the flow had to envelope the entire structure (Ward, 1998). Figure 9.

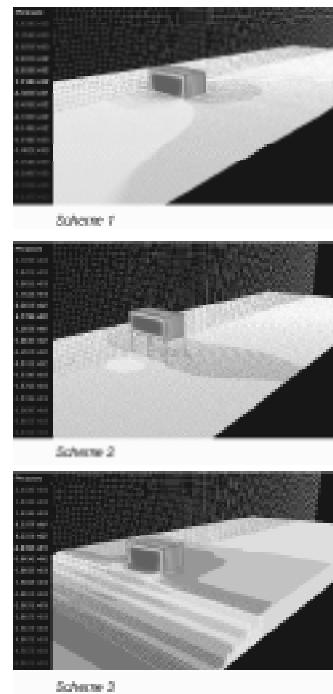


Fig 9 - kuenstle_Pressure Distribution.



The CFD software computed pressure values and their distribution over the building structure for the three schemes are illustrated in (fig. 9). The highest positive pressures (inward forces) occur on the windward face with negative pressure (outward forces) occurring on the side, roof and leeward surfaces. The greater negative pressure generated at the leading edge of the side and roof surfaces (and the underside surface of scheme 2) demonstrate the expected behavior that is consistent with well documented results of physical model data and wind tunnel testing. A comparative study of the pressure values for both the positive and negative forces acting on the building structure confirms the initial assumption that raising the building to higher elevations subjects the building structure to higher wind loads. The maximum wind loads are experienced with scheme 3 where the building is subjected to increased wind flow over the escarpment.

For verification of the simulation results the ASCE 7-98 (Eq. 5) provides standard formulas and tabled coefficients relating to height, exposure, terrain, and building geometry for calculating design velocity pressures. In each of the calculations below the dynamic velocity pressure is derived from the kinetic energy of moving wind, as discussed previously, and is converted into an equivalent static load derived from the Bernoulli equation (Eq. 1) and Newton's law of a mechanical force ($F=ma$) yielding the following expression

$$q = .613 V^2 \quad (\text{N/m}^2) \quad q = .00256 V^2 \quad (\text{lb/ft}^2) \quad (\text{Eq. 4})$$

where (Eq.1) is modified to compensate for the units which relate the mass density of air (1.22 kg/m³ at 15°C or .07651 lb/ft³ at 59°F) to force (N/m² or lb/ft²) by means of Newton's second law for which acceleration is $g = 980.7 \text{ cm/sec}^2$ (32.2 ft/sec²), (ASCE 7-98). For calculating the design pressure on the windward surface the ASCE 7-98 provides the following expression

$$p = q G C_p \quad \text{ASCE 7-98 (Eq. 5)}$$

where p is the design pressure, q is the equivalent static load coefficient (Eq. 4) modified by the building height and terrain factors and the wind velocity, G is a tabled gust factor and C_p is an external pressure coefficient. Application of the ASCE 7-98 method for the windward surface of scheme 1 yields the following:

$$q = .613 (.865) (63 \text{ m/s})^2 = 2104.5 \text{ N/m}^2 \quad (43.95 \text{ lb/ft}^2), \text{ then}$$

$$p = 2104.5 (.85) (.8) = 1431.1 \text{ N/m}^2 \quad (29.8 \text{ lb/ft}^2)$$

For scheme 2:

$$q = .613 (1.0) (63 \text{ m/s})^2 = 2432.9 \text{ N/m}^2 \quad (50.8 \text{ lb/ft}^2), \text{ then}$$

$$p = 2432.9 (.85) (.8) = 1654 \text{ N/m}^2 \quad (34.5 \text{ lb/ft}^2)$$

For scheme 3, which incorporates a topographic factor for the wind flow over an escarpment:

$$q = .613 (.865) (2.25) (63 \text{ m/s})^2 = 4735.2 \text{ N/m}^2 \quad (98.8 \text{ lb/ft}^2), \text{ then}$$

$$p = 4735.2 (.85) (.8) = 3219.9 \text{ N/m}^2 \quad (67.2 \text{ lb/ft}^2)$$

The CFD pressure results for the windward surface, scheme 1, indicate a maximum pressure of 1638 N/m² (34.2 lb/ft²) with an averaged reading over 216 cells of 1365 N/m² (28.5 lb/ft²) which for the trial study is consistent with the ASCE 7-98 calculated design pressure of 1431.1 N/m² (29.8 lb/ft²).

The CFD pressure results for the windward surface, scheme 2, indicate a maximum pressure of 2189 N/m² (45.7 lb/ft²) with an averaged reading over 216 cells of 1459 N/m² (30.4 lb/ft²) which is also in close agreement with the ASCE 7-98 calculated design pressure of 1654 N/m² (34.5 lb/ft²).

The CFD pressure results for the windward surface, scheme 3, indicate a maximum pressure of 2434 N/m² (50.8 lb/ft²) with an averaged reading over 216 cells of 2028 N/m² (42.3 lb/ft²) which is significantly less (almost 25 lb/ft²) than the ASCE 7-98 calculated design pressure of 3219.9 N/m² (67.2 lb/ft²).

While the simulation model shows an increase in the wind load, which is consistent with the expected behavior for the wind speed-up effect over the escarpment, the discrepancy in the pressure values warrants further investigation for a fine-tuning of the model and a closer inspection of the ASCE 7-98 data and method. Similar calculations for the above can be made for both the leeward and sidewalls as well as for the roof (not demonstrated here).

6. Conclusion

The initial trial studies developed with the project demonstrate that the CFD models were able to establish a clear relationship between the simulated wind phenomena and its interaction with the building structure.

While the immediate potential of CFD modeling for use in wind engineering continues to exist primarily in its extraordinary graphic capabilities for visualizing complex flow phenomena, the current study suggest that as research and validation of CFD applications in building design continue to be developed and critically reviewed, the simulation model can provide engineers and architects with an important virtual tool to assist in the mitigation of wind damage to buildings.

Bibliography

- Albertson, Maurice L., James R. Barton and Daryl B. Simons. Fluid Mechanics for Engineers. Englewood Cliffs, NJ: Prentice Hall, 1960.
- Ambrose, James and Dimitry Vergun. Simplified Building Design For Wind and Earthquake Forces. New York, NY: John Wiley & Sons, Inc. 1995.
- American Society of Civil Engineers (ASCE). ASCE 7-98, Minimum Design Loads for Buildings and Other Structures. Reston, VA: 2000.
- Ferziger, Joel H. and Milovan Peric. Computational Methods for Fluid Dynamics. Berlin, Germany: Springer-Verlag, 1999.
- Kim, Sung-Eun and Ferit Boysan. "Application of CFD to Environmental Flows," The Journal of Wind Engineering and Industrial Aerodynamics, 81 (1999): 145-158.
- Kuenstle, Michael. "Computational Fluid Dynamics Applications: Wind Engineering for the Design of Building Structures in Wind Hazard Prone Areas," Proceedings of the 5th Iberoamerican Congress of Digital Graphics, Sigradi 2001, Concepcion, Chile. (2001): 67-70.
- Schodek, Daniel L. Structures, 4th ed. Upper Saddle River, NJ: Prentice Hall, 2001.
- Versteeg, H. K. and W. Malalasekera. An Introduction to Computational Fluid Dynamics. Essex, England: Prentice Hall - Pearson Education Limited, 1995.
- Ward, Del and Stan Crawly. Wind Forces. Washington, DC: National Council of Architectural Registration Boards, 1998.

