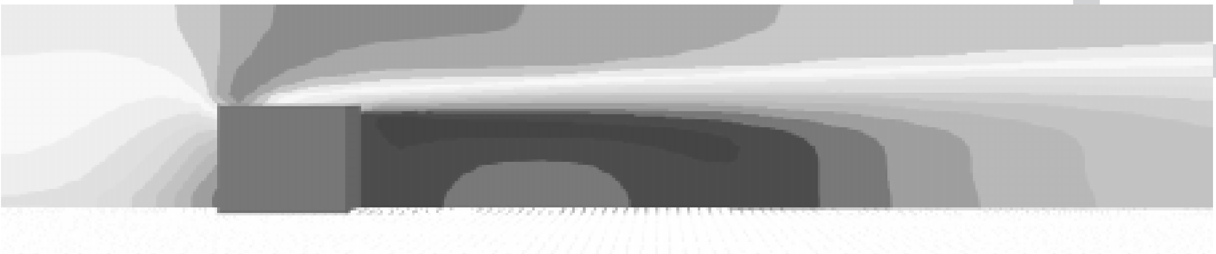
COMPUTATIONAL FLUID DYNAMIC APPLICATIONS
IN WIND ENGINEERING FOR THE DESIGN OF BUILDING STRUCTURES IN WIND HAZARD PRONE AREAS



Abstract

This paper documents an initial study investigating the integration of Computational Fluid Dynamics (CFD) simulation modeling into wind mitigation design for building structures located in wind hazard prone areas. Some of the basic principles 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 CFD application case study for a simulated hurricane force wind flow over a low rectangular building using the k-epsilon turbulence model. The techniques and parameters for development of the simulation are discussed and some preliminary interpretations of the results are evaluated by comparing its predictions against existing experimental and analytical data, with special attention paid to the American Society of Civil Engineers, Minimum Design Loads for Buildings and Other Structures, ACSE 7-98 *and the* Uniform Building Code*.*

Michael W. Kuenstle, AIA School of Architecture, University of Florida, USA
kuenstle@ufl.edu

Synthesis image:
Simulated hurricane force wind over a low rectangular building

The analysis of fluid motion and flow structure using computer simulation modeling, referred to as Computational Fluid Dynamics (CFD), has been used in the combustion, automotive and aerospace design industries with considerable success over the past few decades. 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. Currently, with the rapid development of commercially available CFD codes in combination with access to more powerful computers, CFD applications are becoming more common in other disciplines as the trend in design analysis moves increasingly toward simulation rather than physical model experiments.

As a new tool for building design professionals, the application of CFD software to simulate wind flow conditions on building structures exhibits great potential for improving the understanding of wind phenomena and its dynamic interactions with the built environment.

The current study for a simulated hurricane force wind flow over a low rectangular building using the k-epsilon turbulence model has evolved within the framework of the commercial CFD code, Phoenics 3.4. To establish a theoretical foundation for the initial case study CFD application, some preliminary groundwork and discussion is required to assist in defining the general perspective and scope of the investigation. While there is great interest and value in the graphic simulation of the wind velocity and pressure distribution patterns generated with the study, it is also desirable to schematically develop a well-posed problem based on established governing principles. Most important, a geometrically precise and numerically homogeneous scheme allows for a more convincing comparison of the simulation results with existing data and methods.

In the analysis and design of structural systems 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 building structures 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 can often be sudden, complex, and unpredictable, as in a turbulent flow of wind caused by vortex shedding of an adjacent building or simply as in a brief gust of high wind associated with a seasonal weather storm. It is during these conditions that the resultant forces of the dynamic load source are often characterized by rapid changes in magnitude, direction and distribution over a given structure making the design criteria for the expected behavior and deformations complex and difficult to predict (Schodek, 2001).

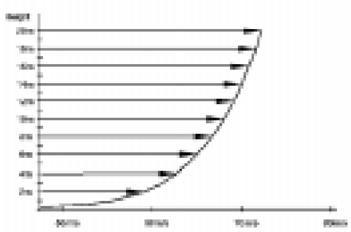
To gain a better understanding of this wind phenomenon as a dynamic load source and to aid in the translation of the CFD simulation results, some brief discussion of a few basic principles regarding the behavior of wind as developed in fluid mechanics can be appreciated. In fluid mechanics, it is stated 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 has the specific physical properties of air with a given mass density, temperature, and viscosity, and flows at a determined velocity through some known domain with assigned physical boundaries.

Furthermore, as the stream of air interacts with each of its 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 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:

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

where q is the resultant dynamic pressure of the potential energy, **p** is the mass density of the fluid, and V is the velocity vector of the fluid flow (Schodek, 2001).

Additionally, 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 surface (Albertson, 1960). The amount of deceleration to the flow stream near the boundary is directly related to the roughness of the boundary's surface.



Wind Velocity Profile(V.. =68m/s)

(Fig. 1) 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:

$$V = V_{ref}\ (Z / Z_{ref})^{1/a} \qquad \text{(Eq. 2)}$$

where V_{ref} is the reference velocity, **Z** is distance from the boundary corresponding to velocity V, Z_{ref} is distance from the boundary corresponding to velocity V_{ref} and **a** is the roughness coefficient for a given exposure condition.

The third and perhaps most important principle related to the study of any flow regime is the principle that links fluid pressure with velocity along a 2D flow stream. 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 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 for mass and momentum, 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 is summarized by the following expression;

$$q_1 + 1/2\ p\ V_1^2 = q_2 + 1/2\ p\ V_2^2 \quad \text{(Eq. 3)}$$

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 are solved (Fig. 2).

As an example CFD study for the schematic development of 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. The final domain parameters and placement of the building structure were determined after several trial study applications with the CFD solver (fig. 3). The simulation is a single-phase flow, implementing the k-e turbulence model, has 216,000 cells and is converged after 10,000 iterations (fig. 4). The attributes of the boundary conditions for the building were determined within the software.

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. The recorded wind speed was converted from exposure category C to exposure category D using a derivation of the power-law expression (Eq. 2) (Ward 1998) summarized as

$$V_z = \left[V_{ref} \left(V_{ref\ g} / V_{ref\ z} \right)^{1/a} \right] \left(Z / Z_g \right)^{1/a} \qquad \text{(Eq. 4)}$$

where V_{ref} is the reference velocity from the wind speed map, $V_{ref\ g}$ is the gradient height for the reference velocity exposure, $V_{ref\ z}$ is the height above the ground surface for which the reference velocity was recorded, **Z** is the height above the ground surface for velocity V_z, and **Zg** is the gradient height for the corresponding exposure category for V_z. The profile exponent **a** and corresponding gradient heights were determined from table 6-4 of *ASCE 7-98*, as suggested by A. G. Davenport (Ward, 1998) where **a** = 9.5 for category C and **a** = 11.5 for category D.

V_z = [63 m/s (274.32 m / 10 m) $^{1/9.5}$] (10 m / 213.36 m) $^{1/11.5}$ = 68 m/s (153 mph)

After the initial wind speed adjustment, a wind velocity profile was determined (Fig. 1) using the power-law scheme (Eq. 2), then input into the software to study the development of the flow and it’s behavior with the domain boundary prior to incorporating the building into the simulation (Fig. 5).

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

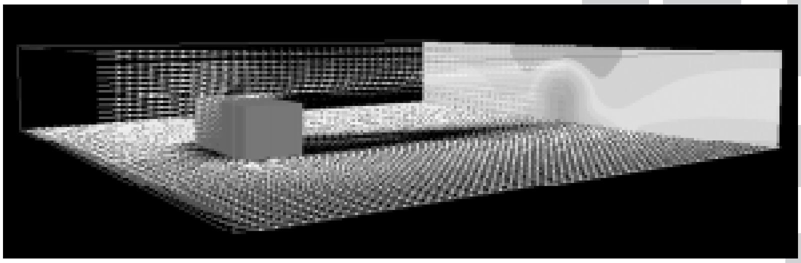


Fig 2: 3D Flow Simulation

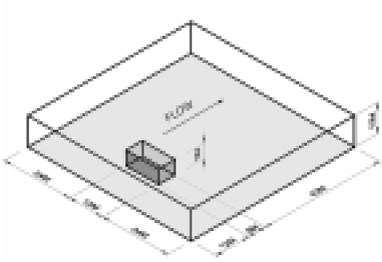


Fig 3: Domain Schematic

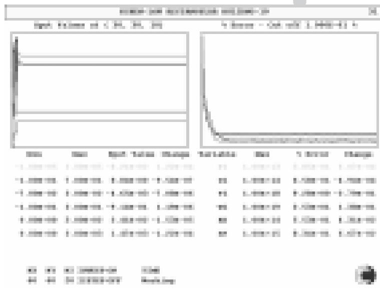


Fig 4: numerical solution for convergence

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).

For verification of the simulation results both the *ASCE 7-98* (Eq. 6) and the *Uniform Building Code* (Eq. 7) provide similar standard formulas and tabled coefficients relating to height, exposure, and building geometry for calculating design velocity pressures. In each of the methods the dynamic velocity pressures are derived from the kinetic energy of moving wind, as discussed previously, and are converted into an equivalent static load developed form Bernoulli (Eq. 1) and Newton’s law of a mechanical force (F=ma) yielding the following expression

$$q = .613\, V^2 \quad (\text{N/m}^2)$$
$$q = .00256\, V^2 \quad (\text{lb/ft}^2) \qquad \text{(Eq. 5)}$$

where (Eq.1) is modified to compensate for the units which relate the mass density of air (1.22 kg/m^3 at 15^{0}C or .07651 lb/ft^3 at 59^{0}F) to force (N/m^2 or lb/ft^2) by mean’s of Newton’s second law for which acceleration is g = 980.7 cm/sec^2 (32.2 ft/sec^2) (Ambrose 1995).

The CFD software computed pressure values and their distribution over the windward surface for the simulation are illustrated in (fig. 6). The positive (inward acting) pressures range from 2861 Pa (59.75 lb/ft^2) to 167.4 Pa (3.5 lb/ft^2) with a small quantity of negative (outward acting) pressure very near the perimeter of the windward surface where turbulence is created at the edges. The majority value of the pressure is in the 2861 Pa (59.75 lb/ft^2) range and the calculated average pressure over the 960 cells is 2310 Pa (48.25 lb/ft^2).

Fig 5: Fluid Velocity profile simulation

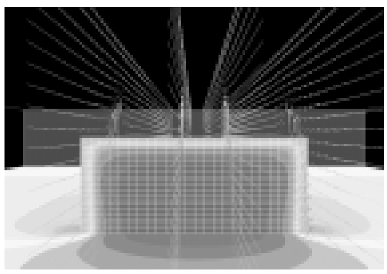


Fig 6: wind flow and pressure distribution - windward surface

To compare the software generated values with those obtained through the *ASCE 7-98* method; the following calculations were made

$$p = q\, G\, C_p - q_i\, (GC_{pi}) \quad \textit{ASCE 7-98} \quad \text{(Eq. 6)}$$

q = .613 (1.04) (63 m/s)^2 = 2530.31 N/m^2 (52.8 lb/ft^2)
p = 2530.31 (.85) (.8) - 2530.31 (-.18) = 2176.06 N/m^2 (45.44 lb/ft^2)

and similarly by using the *Uniform Building Code* method;

$$p = q\; C_e\, C_q\, q_s\, I \quad \textit{UBC 1994} \quad \text{(Eq. 7)}$$

q = .613 (49.2 m/s)^2 = 1483.85 N/m^2 (30.9 lb/ft^2)
p = (1.45) (.8) (1483.85) = 1721.26 N/m^2 (35.9 lb/ft^2)

Note that for the same geographic location, the *UBC* uses the lower "fastest mile" wind speed and a higher value for the gust coefficient compared to the *ASCE 7-98*. Also, the *ASCE 7-98* computation includes internal pressure on the windward surface. The importance factor in each equation is ignored. As the wind profile for the simulation was developed for a 3-second gust wind speed, the simulated values correspond more closely to the *ASCE 7-98* method as demonstrated. The simulation result 2310 N/m^2 is slightly above the *ASCE 7-98* result 2176 N/m^2 and would be acceptable as a design value for a simple structure. While the *UBC* figures are lower, they serve as a reference for comparison. Similar calculations can be made for the leeward and sidewalls as well as for the roof (not demonstrated here).

In conclusion, the development of the initial CFD model established a clear relationship between the simulated wind phenomena and its interaction with the building structure. The preliminary results of the study were verified for accuracy by comparing a sample of its predictions against results using established methods and, therefore, demonstrate the application of CFD modeling in structural design. While the immediate potential of CFD modeling for use in wind engineering exist primarily in its extraordinary graphic capabilities for visualizing complex flow phenomena, the current study suggests that as research and validation of CFD applications in building design are developed and critically reviewed, the simulation model will provide engineers and architects with a virtual tool to assist in mitigation of hurricane damage to buildings.

Bibliography

Albertson, Maurice L., James R. Barton and Daryl B. Simons, "Fluid Mechanics for Engineers", Prentice Hall, Englewood Cliffs, N.J., 1960.

Ambrose, James and Dimitry Vergun, "Simplified Building Design For Wind and Earthquake Forces", John Wiley & Sons, Inc., New York, N.Y., 1995.

American Society of Civil Engineers (ASCE), "ASCE 7-98, Minimum Design Loads for Buildings and Other Structures", New York, N.Y., 2000.

Ferziger, Joel H. and Milovan Peric, "Computational Methods for Fluid Dynamics", Springer-Verlag, Berlin, Germany. 1999.

Schodek, Daniel L., "Structures", 4th ed., Prentice Hall, Upper Saddle River, N.J., 2001.

Versteeg, H. K. and W. Malalasekera, "An Introduction to Computational Fluid Dynamics," Prentice Hall - Pearson Education Limited, Essex, England. 1995.

Ward, Del and Stan Crawly, "Wind Forces", National Council of Architectural Registration Boards, Washington, D.C., 1998.