Flow Structure Environment Simulation

A Comparative Analysis of Wind Flow Phenomena and Building Structure Interaction

KUENSTLE Michael W., AIA
Assistant Professor, School of Architecture, University of Florida, USA
kuenstle@ufl.edu

This paper documents the progress of research to investigate the integration of 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” wind flow over a low rectangular building located in a coastal region. 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.

Keywords: CFD, Wind Load Analysis, Building Structures

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

Figure 1. Simulated hurricane force wind flow over a low rectangular building located downwind of an escarpment
concerns, the study of the behavior of building structures subjected to high winds is of great importance to architects, engineers and city planners. Wind engineering research involving environmental flows 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.

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, Phoenics 3.4 with the implementation of a “standard” k-? 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.

The Computational Fluid Dynamics Approach and Flow Visualization

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.

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-ε 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-ε model continue to be developed for special applications, the accepted use of the “standard” k-ε 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.
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. 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.

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 Phoenics 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.

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).

The CFD software computed pressure values and their distribution over the building structure for the three schemes are illustrated in (fig. 4). 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.

**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.

**Acknowledgements**

This developing research project has been made possible, in part, by a grant from the FRSA Educational and Research Foundation, Inc. and the encouragement of the architecture faculty at the University of Florida. Dr. Gary Chen provided technical assistance for the initial CFD modeling and numerical solutions. Todd Whitehead and Ji-Youn Han provided assistance with the computer modeling as part of a special studies course taught at the University of Florida Graduate School of Architecture.

**References**


