Computational Fluid Dynamics and Physical Model Comparisons of Wind Loads and Pedestrian Comfort Around a High Rise Building

Robert N. Meroney[1], P.E. Ph.D., David E. Neff[1], Ph.D.,
Cheng-Hsin Chang[2], Ph.D, and Rani Pradoto[1]

Introduction

A significant characteristic of modern building design is lighter cladding and more flexible frames. These features produce an increased vulnerability of glass and cladding to wind damage and result in larger deflections of the building frame. In addition, increased use of pedestrian plazas at the base of the buildings has brought about a need to consider the effects of wind and gustiness in the design of these areas. Tall structures have historically produced unpleasant wind and turbulence conditions at their bases. The intensity and frequency of objectionable winds in pedestrian areas is influenced both by the structure shape and by the shape and position of adjacent structures. The two primary design methods available today to evaluate wind effects on tall structures are physical modeling (wind or water tunnels) and computational fluid dynamics (cfd). A case study is considered where both methods are compared to consider their advantages, disadvantages, and evidence for agreement in results.

Principal Concepts

Techniques have been developed for wind-tunnel modeling of proposed structures which allow the prediction of wind pressures on cladding and windows, overall structural loading, and also wind velocities and gusts in pedestrian areas adjacent to the building. Information on sidewalk level gustiness allows plaza areas to be protected by design changes before the structure is constructed. Accurate knowledge of the intensity and distribution of the pressures on the structure permits adequate but economical selection of cladding strength to meet selected maximum design winds and overall wind loads for the design of the frame for flexural control. A detailed discussion of the similarity requirements and their wind-tunnel implementation can be found in Cermak (1971,1976). In general, the requirements are that the model and prototype be geometrically similar, that the approach mean velocity at the building site have a vertical profile shape similar to the full-scale flow, that the turbulence characteristics of the flows be similar, and that the Reynolds number for the model building be greater than a critical value. These criteria are satisfied by constructing a scale model of the structure and its surroundings and performing the wind tests in a wind tunnel specifically designed to model atmospheric boundary-layer flows.

Simulation of flow and forces around buildings using cfd requires consideration of many of the same concerns and constraints as physical modeling. One must assure equivalence of inlet and boundary conditions, geometric similarity of the model, consideration of spatial resolution of any numerical grid, and appropriateness of the computational turbulence model. Various authors have considered these topics at length in recent specialty conferences, (Murakami, 1992; Meroney, 1996, Baker, 2000).

Building Case Study

The wind-engineering study is performed on Building 67 of the Denver Federal Center, Denver, Co. The building group was modeled at a scale of 1:240 in the Environmental Wind Tunnel at Colorado State University. The structure is modeled in enough detail to provide accurate flow

---

[1] Civil Engineering Department, Colorado State University, Fort Collins, CO

[2] Graduate Institute of Environmental Engineering, National Taiwan University, Taiwan

patterns in the wind passing over the building surfaces. The building under test is located in a surrounding where nearby buildings and terrain are incorporated out to 500 m from the complex. The test model, equipped with many pressure taps (376 for this study), was exposed to an appropriately modeled atmospheric wind in the wind tunnel and the fluctuating pressure at each tap measured electronically. The model, and the modeled area, was rotated 10 degrees and another set of data recorded for each pressure tap.

Based on the visualization (smoke) tests and on a knowledge of heavy pedestrian use areas, several locations were chosen at the base of the building where wind velocities can be measured to determine the relative comfort or discomfort of pedestrians in plaza areas, near building entrances, near building corners, or on sidewalks.

Various configurations of Building 67 were also considered by numerical modeling using the commercial code Fluent 5.5. Computational grids were prepared for various wind angles using the preprocessor code Gambit. Alternative turbulence models were considered to check sensitivity of the results to model selection. Calculated pressures and flow velocities were plotted against comparable physical model measurements.

Results

Pressure coefficients of several types were calculated for each physical model piezometer tap. Integration of test data with wind data resulted in prediction of peak local wind pressures for design of glass or cladding. Pressure contours were drawn on the developed building surfaces showing the intensity and distribution of peak wind loads on the building. Examination of winds at ground level for pedestrian comfort identified several alternative windbreak and landscaping scenarios that reduce pedestrian winds.

Similar pressure and wind flow presentations were produced by the physical model measurements and cfd calculations. Initial conclusions are:

- The general character of flow about the building produced by the two modeling alternatives is very similar. Regions of flow acceleration, separation, reattachment, and evidence for rooftop and ground level vortices are evident.
- Mean pressure coefficients calculated by the two modeling alternatives are very similar. Regions of positive pressure and negative suction are coincident in both models.
- Landscaping and architectural alternatives considered during the two model alternatives produced similar conclusions concerning useful modifications to the plaza area.

References

Cermak, J. E., "Laboratory Simulation of the Atmospheric Boundary Layer," *AIAA Jrnl.*, Vol- 9, September 1971.

Cermak, J. E., "Aerodynamics of Buildings," *Annual Review of Fluid Mechanics*, Vol. 8, 1976, pp. 75-106.

Murakami, S., Editor, "Proceedings of 1st Int. Symposium on Computational Wind Engineering", U. of Tokyo, Japan, 21-24 August 1992, also Jrnl. *of Wind Eng. and Ind. Aero.,* Vol. 46-47, 1993.

Meroney, R.N and Bienkiewicz, B. Editors, "Proceedings of 2nd Int. Symposium on Computational Wind Engineering (CWE96)", Colorado State University, Fort Collins, 4-8 August 1996, also *Computational Wind Engineering 2*, Elsevier, Amsterdam, 1997.

Baker, C., Editor, "Proceedings of 3rd Int. Symposium on Computational Wind Engineering", Univ. of Birmingham, UK, 4-7 September 2000.