A CWE/WT Study of the Flow over High- and Low-Rise Buildings, with Anisotropic Mesh Optimization

Frédéric Tremblay, Martin Aube, Wagdi G. Habash, Congjun Wang, Bencai Huang, Guojian Wang

"Newmerical Technologies international, Montreal, Canada
"Tongji University, Shanghai, PRC
"Shanghai Modern Architectural Design Group, Shanghai, PRC

ABSTRACT: This paper addresses the critical issue of the accuracy of CFD predictions in wind engineering. Flows around both a high-rise building, the Jin Mao Tower, and low-rise (but large-span) buildings from the Pudong International Airport, are computed with the Navier-Stokes solver FENSAP and compared to experiments in a unique academic-architectural collaboration framework (China-Canada Architectural Wind Simulation Center). Computations are carried out for two wind directions, with FENSAP solving the steady-state ensemble-averaged Navier-Stokes equations and the Spalart-Allmaras turbulence model. Pressure coefficients compare well with wind tunnel experiments. The accuracy of the flow solutions is further improved by using automatic mesh adaptation that dynamically places grid points where the flow physics require them, while keeping the number of unknowns (and hence the solution time) substantially at the same level.

KEYWORDS: Computational wind engineering; Finite element method; Mesh adaptation; Wind tunnel; High-rise building; Low-rise building.

1 INTRODUCTION

CFD is a mature science that is extensively used since more than four decades for the design and analysis of complex industrial components involving fluid flows. As decision makers are relying more and more on CFD results, its credibility, however, becomes more recently a critical issue. It all started in 1998 when the Computational Fluid Dynamics Committee on Standards of the AIAA issued the Guide for the Verification and Validation of Computational Fluid Dynamics Simulations. The objective of this report was to quantify, and reduce, errors in the computational model and its numerical solution. As highlighted by Oberkampf et al., when a numerical procedure is stable, consistent and robust, the five primary sources of errors in CFD are: insufficient spatial discretization, insufficient temporal discretization, insufficient iterative convergence, computer round-off, and computer programming. Increasing the confidence in CFD results required clear guidelines on how to quantify, control and ideally reduce each of these errors.

Such effort is currently underway in Computational Wind Engineering (CWE) where working groups such as QNET-CFD and the COST Action C14 in Europe, and the AIJ cooperative project for CFD prediction of wind environment, all issued best-practice guidelines. These mainly focus on boundary conditions, on the size and grid-quality of the computational domain, and on turbulence modeling. To minimize grid-related errors, Franke et al. proposed that the grid should be made fine enough to capture shear layers and vortices with sufficient resolution, that the grid stretching/compression should be small in regions of high gradients, and that
neighboring cells should be made as parallel as possible to walls. Such guidelines definitively open the door for an automatic approach where solution quality is maintained independently of the grid, physics and numerical scheme.

The general trend in large-scale CFD problems is the use of fine grids as much as possible to resolve physical features such as boundary layers, flow separation, and recirculation regions. Taking this to a sometimes implausible limit, mesh-independence seekers use finer and finer meshes until the solution no longer changes. With most complex multi-scale and multi-physics phenomena, attaining such mesh independence in 3-D, even for those best equipped in terms of computers, can be a chimera.

The alternative approach proposed in this paper is to seek solutions on anisotropic meshes that improve results without necessarily increasing the overall number of mesh points. Its high performance has been assessed for external aerodynamics (Habashi et al.\textsuperscript{5}). It is extended to wind engineering applications by computing flows around both a high-rise building, the Jin Mao Tower, and low-rise (but large-span) buildings from the Pudong International Airport, both well-known in Shanghai. Initial and automatically adapted flow solutions from the Navier-Stokes solver FENSAP and the mesh adaptation tool OptiGrid, are compared to experiments in a unique academic-architectural collaboration framework (China-Canada Architectural Wind Simulation Center).

2 RESULTS AND DISCUSSION

2.1 Validation of FENSAP turbulent for wind engineering applications

The validation of FENSAP against boundary layer wind tunnel tests and full-scale field measurements is performed on the well-known surface-mounted cubic model from Silsoe Research Institute (see Richards et al.\textsuperscript{6} for a complete description of the geometry and available data). The numerical model was built according to the real scale of the cube. The computational domain was discretized using 218,000 tetrahedral and 124,000 prism elements. A fixed pressure boundary condition was imposed at the exit plane, while an atmospheric boundary layer profile with a mean velocity of 10 m/s was imposed on the inlet plane. FENSAP results were compared with four turbulent boundary layer wind tunnel tests and a full-scale field measurement, all at different but close experimental conditions, as shown in Figure 1.

![Figure 1. Vertical central section mean-Cp with wind normal to one face.](image)

2.2 The Jin Mao high-rise

The Jin Mao building is located in the Pudong area of Shanghai, China. It consists of a 365 meters tall tower and an adjoining low-rise building, about 40 meters in height. The wind tunnel tests were carried out by TongJi University at a length scale of 1:500, with pressure...
measurements on the surface of the building. Measurement points were positioned at several heights, as shown in Figure 2. Their turbulent intensities ranged from 10% to 20%.

![Figure 2. Azimuth angles (left) and, distribution of taps in experiment (right). Only the zero-degree incidence is considered in this paper.](image)

Results for the zero-degree incidence wind direction are shown in Figure 3 for, respectively, 145, 245, 286 and 332 meter heights. The flow solution computed by FENSAP on the initial grid (shown with the dashed lines) matches the experiments with good accuracy. Some discrepancies can however be observed when flow separates. The accuracy of the numerical prediction is clearly improved in these zones by mesh adaptation (shown with full lines).

![Figure 3. Cp distributions at different heights from 145 to 332 meters, with zero-degree inlet flow incidence. Results on the initial grid with dashed lines, while results with adaptation with full lines.](image)

2.3 The Shanghai Pudong International Airport

The Shanghai Pudong International Airport is located in an open field, as shown in Figure 4. One group of buildings had already been built, while the other is now under construction.

![Figure 4. The Pudong International Airport (left) showing the two new buildings and, the two incidence wind directions (135 and 180 degrees) considered in this work (right).](image)
Results for both the 135 and 180-degree incidence wind directions are shown in Figure 5 for measurement planes 1, 2 and 5 on building 1. On most experimental taps, especially in the separated flow regions, mesh adaptation has improved the quality of the comparisons. It should be noted, however, that due to the complexity of wind tunnel tests, the experimental data contains some level of error.

Figure 5. Cp distributions on building 1 at 135- (left) and 180-degree (right) flow incidences. The FENSAP solution on the initial grid is shown with dashed line, while the full lines represent results on the adapted grid.

3 CONCLUSIONS
This work, conducted by a unique academic-architectural collaboration framework (China-Canada Architectural Wind Simulation Center), validated successfully the Navier-Stokes solver FENSAP by comparing flow results with two new experimental datasets from TongJi University. Flow predictions on the Jin Mao high-rise and on the new low-rise (but large-span) buildings at the Pudong international airport, both compared well to experiments.

The automatic mesh adaptation (OptiGrid) has improved the accuracy of the numerical simulations by clustering grid points where they are needed most. It can thus be seen as a powerful approach to ensure solution accuracy.

4 REFERENCES
1 AIAA, Guide for the verification of computational fluid dynamics simulations, American Institute of Aeronautics and Astronautics, AIAA-G-077-1998, Reston VA.