Wind and Structures, Vol. 5, No. 2~4 (2002) 193-208
DOI: http://dx.doi.org/10.12989/was.2002.5.2_4.193

The numerical wind tunnel for industrial aerodynamics: Real or virtual in the new millennium?

T. Stathopoulos†

Centre for Building Studies, Concordia University, Montreal, H3G 1M8, Canada

Abstract. Previous studies have shown that Computational Wind Engineering (CWE) is still in its infancy and has a long way to go to become truly useful to the design practitioner. The present work focuses on more recent studies to identify progress on outstanding issues and improvements in the numerical simulation of wind effects on buildings. The paper reviews wind loading and environmental effects; it finds that, in spite of some interesting and visually impressive results produced with CWE, the numerical wind tunnel is still virtual rather than real and many more parallel studies - numerical and experimental - will be required to increase the level of confidence in the computational results.

Key words: aerodynamics; computational fluid dynamics; design; pressure; velocity; wind.

1. Introduction

Although Computational Wind Engineering (CWE), dealing with the utilization of Computational Fluid Dynamics (CFD) for the solution of problems in wind engineering and industrial aerodynamics, has been developed since the late 1970's, there are still several problems in the application of numerical solutions for cases of practical interest. Such cases include the assessment of wind loads in buildings, the evaluation of wind environmental conditions in the urban environment, the dispersion of pollutants around buildings etc. Physical modelling in spite of its own limitations is still, at the beginning of new millennium, the predominant practice for industrial aerodynamics applications. The concept of establishing a numerical wind tunnel does not seem to have materialized as yet, despite the tremendous progress in the computational technology and the significant efforts made in Japan, Europe and North America. Nevertheless, a remarkable progress in this direction creates a justified optimism that the numerical wind tunnel will become a reality.

In principle, it is of course possible to solve the complete, time dependent Navier-Stokes equations of fluid motion numerically without making any modelling assumptions regarding turbulence. The computer power necessary to carry out this direct numerical simulation (DNS) for any realistic situation is, unfortunately, much beyond the current computational capabilities. Consequently, in almost all cases, the time-averaged equations of motion are utilized and assumptions are made for the extra terms generated by the averaging process (turbulence modelling). Modelling errors are over and above the discretization errors inherent in the selection of mesh and the domain of interest required for the numerical solution. Additional uncertainties

† Professor

are generated by the critical selection of boundary conditions (initial conditions for time-dependent equations) that are known only approximately due to the various atmospheric conditions involved in wind engineering problems. However, it should be noted that the accuracy requirements in industrial aerodynamics are generally lower than those necessary in aeronautical problems.

Past achievements and future challenges in CWE have been reviewed by Stathopoulos (1997). The study concluded that CWE is still in its infancy and has a long way to go to become truly useful to the design practitioner. The present work focuses on more recent studies to identify progress on outstanding issues and improvements in numerical simulation of wind effects on buildings.

2. Wind pressure loads on buildings

The great majority of the applications regarding numerical evaluation of wind pressures on buildings refer to the basic cube shape exposed to wind perpendicular to its face. This is because (i) the cube has simple geometry but all the important complex features of a real building flow; (ii) several experimental studies and results are available for cubes; and (iii) wind perpendicular to the face represents the basic - although not unique - design wind direction. Wright and Easom (1999) predicted mean pressure coefficients on the middle vertical plane of a cube using four different turbulence models, namely :

- Standard k-ε model
- RNG k-ε model (Yakhot *et al.* 1992), which is derived from renormalization group analysis of the Navier-Stokes equations; the model revises the k-ε constants and modifies the values of eddy viscosity.
- MMK k-ε model (Tsuchiya *et al.* 1997), which intends to improve the prediction of turbulent kinetic energy and eddy viscosity for bluff body flow fields; the standard model constant C_μ becomes a function of the ratio of vorticity to shear, so that, when this ratio is less than one (e.g., flow stagnation), the eddy viscosity is reduced.
- DSM (Differential Stress Model) of Launder *et al*. (1975), which is a more complex anisotropic turbulence model; it solves individual transport equations for each of the six Reynolds stresses leading to increased computational effort.

Fig. 1 shows their results, previous predictions based on k-ε model, and experimental data, in terms of mean pressure coefficients. The predictions on the windward face are in good agreement with the experimental data. At the roof, numerical predictions appear problematic. The standard k-ε model fails to predict the suction pressure at the upwind roof edge. Recent modifications of the k-ε model have improved the discrepancies between the experimental and numerical values on the roof, although such adjustments may be of an ad hoc nature and improve the situation only for some particular cases. Overall however, the range of the pressure coefficients evaluated numerically by using the k-ε model with all its forms and improvements is very comparable to the variations of the experimental data obtained in previous studies and highlighted in Fig. 1.

Tamura *et al.* (1997) reported the activities of the Architectural Institute of Japan (AIJ) concerning CWE. Contour maps of mean pressure coefficients measured and computed by the standard k-ε model and two of its improved versions (MMK, k-ε-ϕ) on the envelope of a low square building, whose height is equal to half of its width, are shown in Fig. 2. The k-ε-ϕ model was introduced by

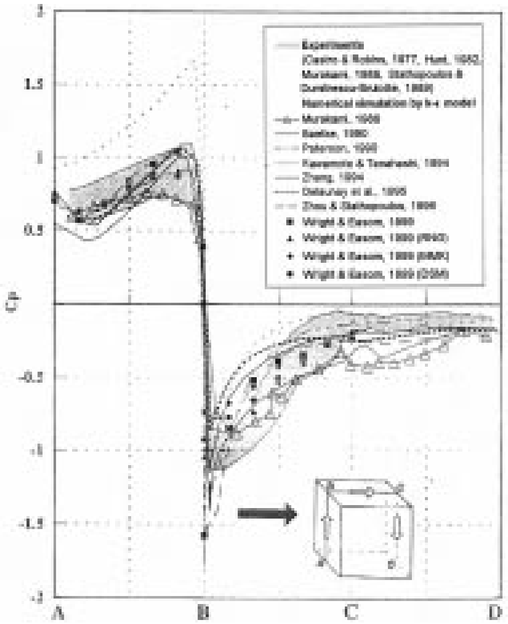

Fig. 1 Comparison of mean wind pressure coefficients: Experimental measurements and numerical simulations by using several turbulence models

Kawamoto (1997); the model includes the anisotropic parameter ϕ of the Reynolds stress, which affects the kinematic eddy viscosity. It is interesting that this model yields results comparing reasonably well with the wind tunnel data even for oblique wind directions, e.g., 45^o, for which the delta wing type of vortices are difficult to be generated numerically; needless to say that there is a significant penalty in terms of computational time required when using this model. In addition, Tsuchiya *et al.* (1997) reported that MMK was capable of modelling the edge vortices better than the standard k-ε model, though the agreement with experimental results still shows discrepancies. As already pointed out, these model improvements are not general but suitable only for some particular problems. The same study concludes that the LES (Large Eddy Simulation) technique can be used even for unfavorable flow fields such as transitional flows, separated flows or complex behaviours of vortices.

A number of studies have used LES modelling to reproduce numerically not only the mean but also the peak pressure coefficients measured in the field experiments of the Texas Tech building but with little success - e.g., Selvam (1997). Contours of space-correlations of fluctuating pressure on the roof of a low-rise building have been presented by Tamura *et al.* (1997). The comparison with wind tunnel test results obtained in smooth flow appears satisfactory when the Smagorinsky model is used in LES, as opposed to a dynamic subgrid-scale model. In general, LES works well in the numerical prediction of the mean velocity and pressure fields for the case of wind normal to the surface of a prism representing a rectangular high-rise building.

Fig. 3 shows the time-averaged streamwise velocity measured and computed along the central line upstream and downstream of the rectangular prism. Clearly, the computational data agree well with the experimental results obtained upstream of the prism. However, there are significant discrepancies in the velocity measured or computed in the wake of the building. Although the k-ε model fails to

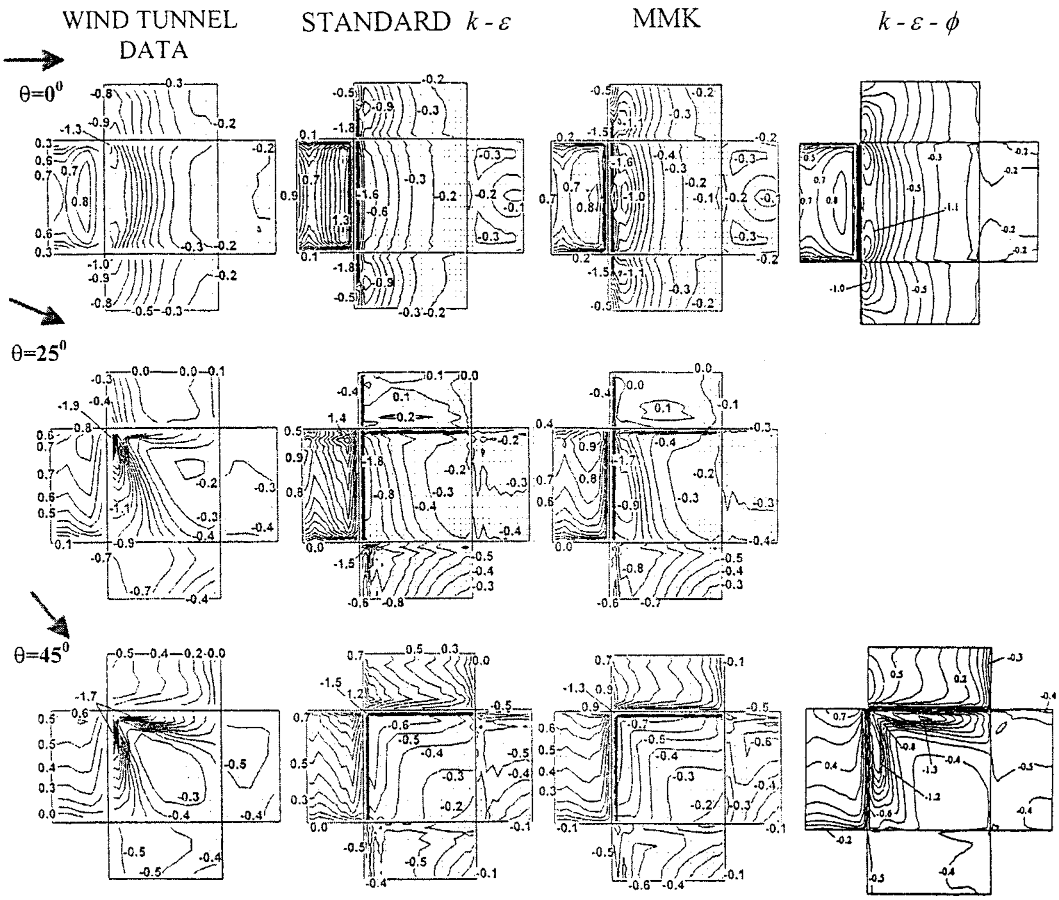

Fig. 2 Contours of mean pressure coefficients (wind tunnel data and numerical results by several turbulence models)

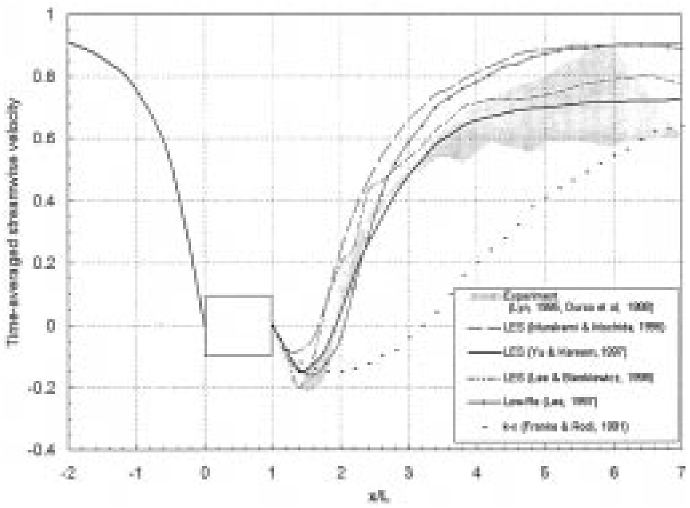

Fig. 3 Time-averaged streamwise velocity upstream and downstream of a square prism

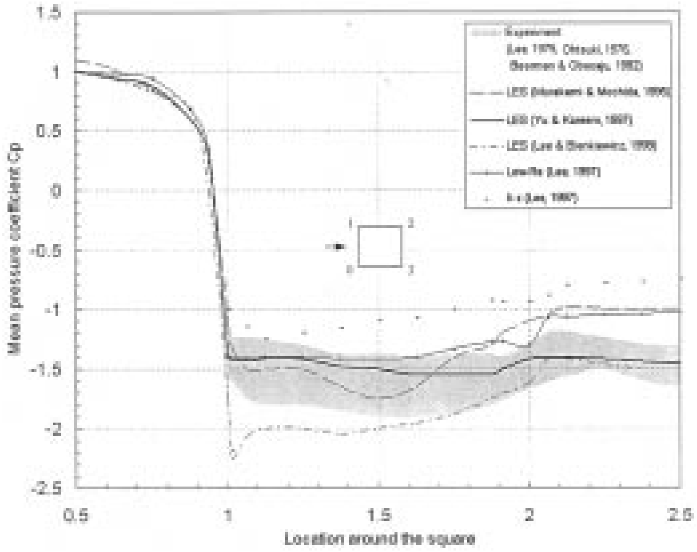

Fig. 4 Distribution of mean wind pressure coefficients on the walls of a square prism

reproduce the velocity in the wake, the low-*Re* k-ε model behaves better; LES is of course the winner, particularly the solution provided by Yu and Kareem (1997). The discrepancy between the computational results of Murakami & Mochida (1995), Yu and Kareem (1997), and Lee and Bienkiewicz (1998), although all based on LES, can be attributed to the employed numerical methods and boundary conditions. For example, the treatment of the outflow boundary by a convective boundary condition as used by Yu and Kareem (1997), is more realistic than a free boundary ($\partial/\partial n = 0$) condition.

Mean pressure coefficients on the windward, side and leeward surfaces of the prism are shown in Fig. 4 for a normal wind direction. Experimental and numerical results are presented for half of the wall surfaces due to symmetry considerations. Again, there is no problem with the windward wall, for which computational models (k-ε, Low-*Re* k-ε, LES) provide results in full agreement with the experimental data. On the other hand, the k-ε model fails to predict the pressures in the wake region (side and leeward walls) whereas LES is generally satisfactory. Also the Low-*Re* model, specified in Bradshaw *et al*. (1994), behaves reasonably well, particularly on the sidewall of the prism. No data seem to be available for peak pressure coefficients, although the latter are those useful for structural design.

It should be stressed however, that LES as a procedure of turbulence modeling is going to be truly useful only if it reaches the stage of producing peak instantaneous pressure coefficients, with some reasonable accuracy. Extreme value analysis can then be used to extract design pressure values with a specified probability of exceedance. Although there has been a trend towards this direction, no successful cases have yet been reported in the literature.

There has been very little effort dedicated to the computational evaluation of wind pressure acting on buildings of non-rectangular shapes. Again, considering the modeling difficulties that the experimentalists come across in the physical simulation of wind flow around building models with curved surfaces, in which the reproduction of *Re* is important, and the relative simplicity of grid generation routines, it appears clear that the computational evaluation of wind pressures on buildings with curved surfaces would meet a real need and would be extremely beneficial.

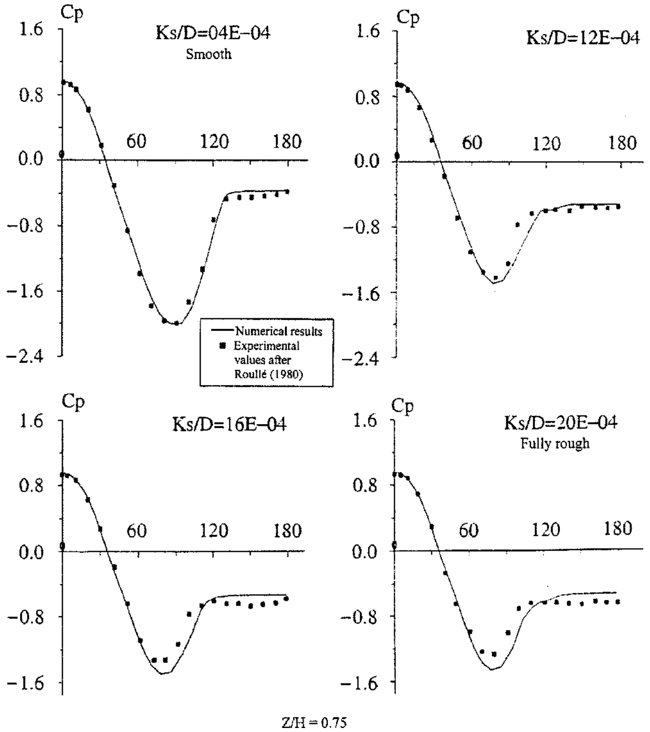

Fig. 5 Comparison of pressure coefficients measured and computed on the surface of a circular cylinder with different surface roughnesses (after Lakehal 1999)

Lakehal (1999) carried out the computational modeling of turbulent flows over rough-walled circular cylinders. He utilized the RANS (Reynolds-Averaged Navier-Stokes Equation) solution with the k-ε turbulence model. He considered the surface roughness through the inclusion of a sink term in the momentum equations and a source term in the k-ε equation. Fig. 5 compares some of his results obtained for cylinders with four different roughnesses (ranging from smooth to fully rough) with the experimental data of Roullé (1980). The comparison concerns pressure coefficients obtained/measured at the horizontal level 3/4 up the cylinder height and is generally satisfactory. However, it is clear that the quality of comparison increases with the reduction of roughness of the cylindrical surface.

Shimada *et al.* (1998) computed aerodynamic forces of a 200 m tall stack with an elliptical and a recessed corner rectangular cross-section. They applied DNS, so calculations were performed for Re<1000 and two-dimensional smooth uniform flow. The validation of the numerical code was made only with rectangular cylinders in smooth uniform flow. The advantage of the study consists of the reproduction of unsteadiness without requiring any turbulence models; the disadvantage is specified by the authors: “by using convex C3840 vectorized parallel computer, it took at least a few weeks to obtain stable results in aerodynamic coefficients”.

3. Wind flow over complex terrain

Wind pressure on buildings and other structures, pedestrian level winds, wind-induced dispersion of pollutants in urban locations depend, among other factors, on the velocity profile and turbulence

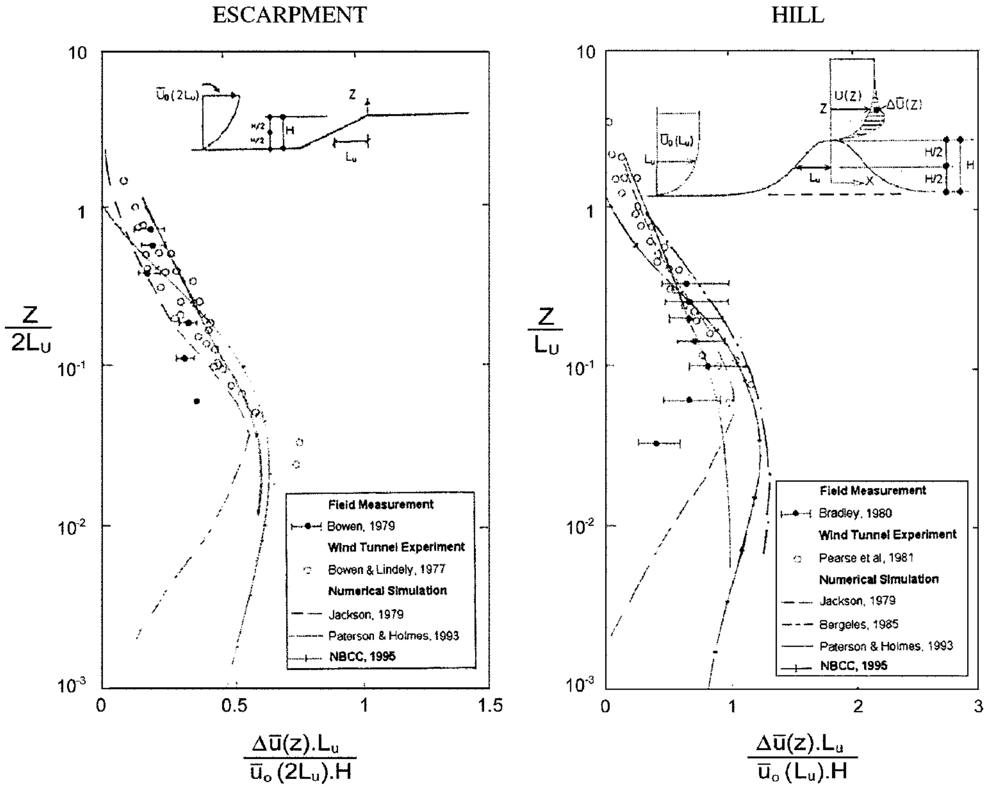

Fig. 6 Dimensionless velocity speed-up factors

characteristics of the upcoming wind. These, in turn, depend on the roughness and general configuration of the upstream terrain. Consequently, wind standards and codes of practice typically assume upstream terrain of homogeneous roughness or provide explicit corrections for specific topographies such as hills or escarpments; for more complex situations they refer the practitioner to physical simulation in a boundary layer wind tunnel. With the evolution of the numerical wind tunnel, it would be of course fully desirable to utilize it in order to evaluate the wind velocities over complex terrain. In fact, significant progress has been made for specific cases of escarpments, single and multiple hills, as well as valleys. Fig. 6 shows typical comparative results for the variation of velocity profiles above an escarpment or on a hilltop. Field measurement results and experimental data from physical simulations have been obtained for a number of geometries and roughnesses. For the general variation of escarpments ($H / (2L_u) = 0.1$ to 0.6) and of hills ($H / L_u = 0.2$ to 0.6), as well as upstream roughness of ($H / z_0 = 400$ to 40,000), data are quite similar. Numerical solutions based on the k-ε model appear very good when the differences between wind tunnel and field data are taken into account. It is worth noting that the speed-up values derived from the National Building Code of Canada (NBCC 1995) compare well with the computational results.

Carpenter and Locke (1999) investigated the variation of wind speeds over steep and shallow, single and multiple two-dimensional sinusoidal hills. Computational results were obtained by using the finite element approach with the standard high *Re* k-ε turbulence model except near the hill surface where a thick layer of special elements was used. A typical comparison of numerical and

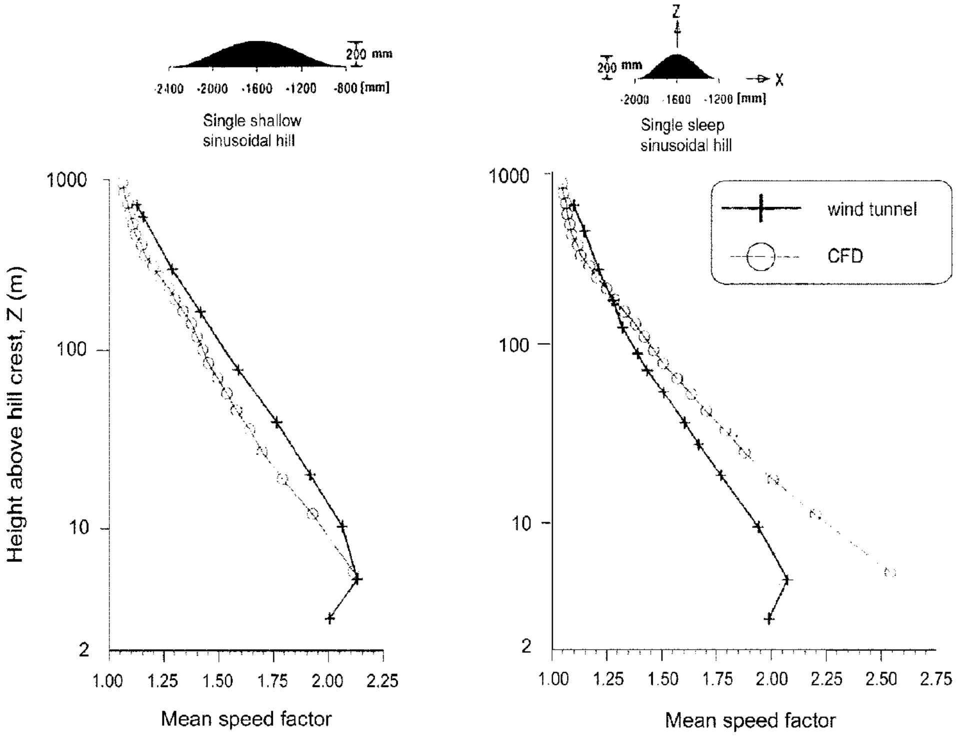

Fig. 7 Comparison of wind tunnel and CFD mean speed amplification factors at the crest of a shallow sinusoidal hill and a steep sinusoidal hill (after Carpenter and Locke 1999)

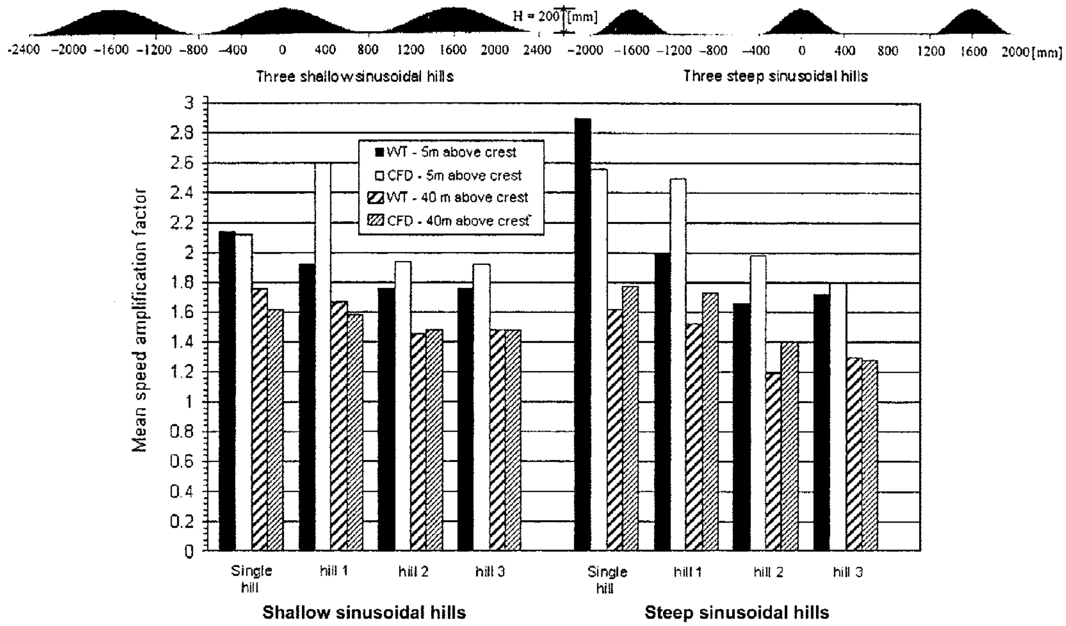

Fig. 8 Comparison of wind tunnel and CFD mean speed amplification factors at the crest of the multiple shallow sinusoidal hills and multiple steep sinusoidal hills (after Carpenter and Locke 1999)

wind tunnel data is shown in Fig. 7, in which mean speed amplification factors are plotted. The agreement is reasonable, although it is clearly better for the shallow hill and for points farther away from the ground in the case of steep hill. The same trend can be demonstrated in Fig. 8, in which a system of three sinusoidal hills, both steep and shallow, has been considered. In addition, the agreement between computational and experimental results seems to be somewhat better downstream than upstream in the case of steep hills. Kim and Boysan (1998) have used a general-purpose computational program with three different k-ε models and the Reynolds-Stress model to evaluate the flow above a two-dimensional hill and compare their results with relevant experimental data. Pressure coefficients measured and evaluated on the surface of the hill show a rather good

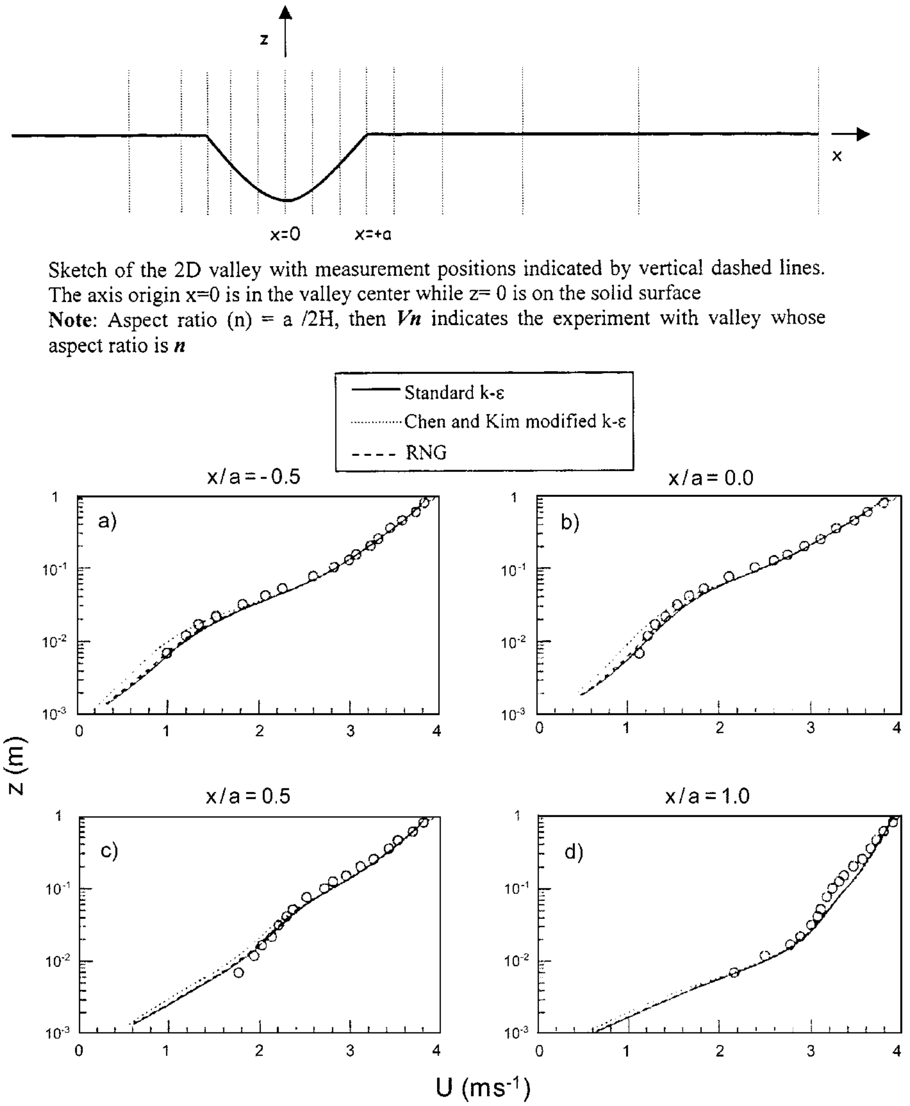

Fig. 9 Vertical profiles of horizontal mean velocity for V8 at four longitudinal locations (after Maurizi 2000)

agreement over the upstream half of the hill. The agreement becomes not so good in the downstream of the hillcrest, where the k-ε models over-predict the pressure recovery in the recirculation region. Overall the second-moment closure model gives the best prediction. Quinn *et al.* (1998) have carried out full-scale measurements over the exposed small embankments of the 23° and 24° slope and compared their results with those obtained from CFD, with a commercial program utilizing three different k-ε models. They found the CFD results in good agreement with the experimental data, although near-ground effects on the embankment crest were poorly reproduced.

Maurizi (2000) presented useful computational results dealing with two-dimensional valleys of various slopes and compared them with the experimental data from the EPA RUSVAL wind tunnel test (1993). Three different k-ε models were utilized: the standard; the one described by Chen and Kim (1987), which accounts for different time scales by an extra production term in the ε-equation depending on the time scale related to the production of k; and the same modification for the RNG version, see Yakhot *et al*. (1992). Fig. 9 shows excellent agreement between all experimental results (regardless of type of k-ε model used) and the experimental data in terms of mean velocities above four longitudinal locations in the case of a gently sloping value. However, the numerical results are not so good when the slope becomes higher, say $a/2H = 3$, and flow recirculation occurs. Overall, the RNG version appears to be most suitable and its application does not increase the computational cost.

4. Environmental wind effect problems

In the main interest of wind engineering these problems can be classified broadly under the following types :

- Pedestrian-level winds
- Snow dispersion and accumulation
- Dispersion of pollutants in the near-building or urban environment.

Pedestrian-level winds can be described quite adequately in terms of mean velocities in the presence and absence of a new building within a specific urban environment. Although it can be argued that pedestrians are mostly affected by gust effects and mean wind speeds may not be sufficient to produce satisfactory results, the fact remains that several major cities require only the satisfaction of certain mean (sustainable) speeds with a specified probability of exceedance. In this regard, studies published by Stathopoulos and Baskaran (1996) have demonstrated that by using a simple version of k-ε model one may obtain computational results very comparable with corresponding values originating from respective wind tunnel studies. This has been confirmed by several authors. For instance, Timofeyef (1998) evaluated the wind flow around a five-storey high development in Kazakhstan and produced full-scale results, wind tunnel data and numerical results by using the discrete vortex method (two-dimensional flow). Surprisingly enough, computational results compare better with corresponding full-scale data than the latter with wind-tunnel results! This means that this rather crude computational approach provides more representative results than wind tunnel testing, at least in this particular case. Finally, in a cooperative study between Portugal and Canada, Ferreira *et al.* (1999) produced wind flow around a group of low-rise buildings (Expo' 98 - Lisbon). Both wind tunnel and field data were compared with numerical results obtained with the standard formulation and the RNG extension of the k-ε turbulence model. By and large the comparisons are satisfactory, at least for engineering purposes.

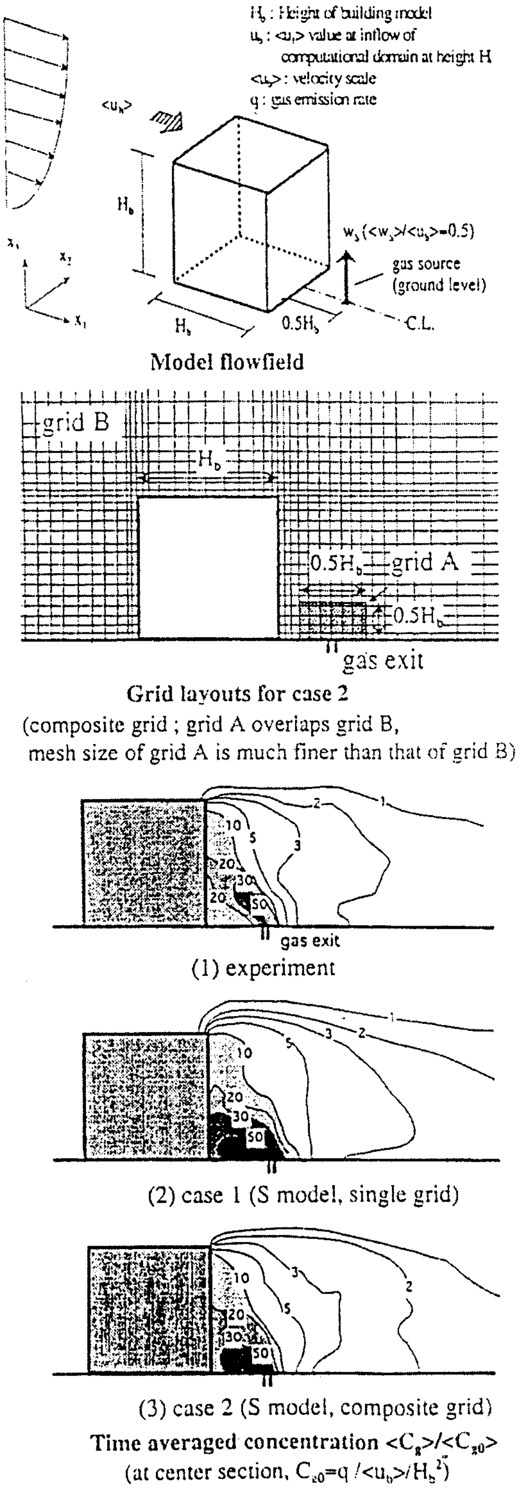

Fig. 10 Dispersion of pollutants around a building (after Murakami 1998)

Snow dispersion and accumulation problems are hard enough to handle with physical simulation in wind tunnels or water flumes. Although CFD has great potential to assist us in this direction, there has been only one recent study by Tominaga and Mochida (1998) in this area. A 3-D CFD program with a modified version of Launder-Kato (L-K) model has been used to evaluate the flow field around a 9-storey apartment building in Nagoaka city in Japan. The flow field was checked by comparison with experimental data: the modified L-K model provided satisfactory results (better than the k-ε) for a cube in channel flow. Two different grid spacings were used and only suspension of snow was considered due to its importance for snowdrift into the area of interest (elevator hall). A snowfall velocity of 0.5 m/s was selected and a transport equation similar to that used for airborne particles in a clean room was utilized. Results have shown the effect of a wind/snow break on the snow drifting density in the elevator hall. However, these data cannot be considered validated at this point in time and much more work is required in this area.

Several studies have addressed the dispersion of pollutants in the building and urban environment. Fig. 10 shows typical results obtained by Murakami *et al.* (1998) regarding the concentration field in the wake of a cube when the pollutant source is located at the ground level near the leeward wall of the building.

The comparison with the experimental data indicates that the utilization of a composite grid increasing its density around the area of the source of the pollutant is extremely beneficial. Other studies have also indicated that the numerical evaluation of concentrations of pollutants in the wake of a building when they are also exhausted from the wake of the building is feasible.

However, the situation becomes more complicated when the exhaust comes out from the roof. For instance, Fig. 11 shows the concentration of pollutants on the roof of a building when the source is on the windward side of the roof. Numerical data evaluated by either the standard k-ε model or by

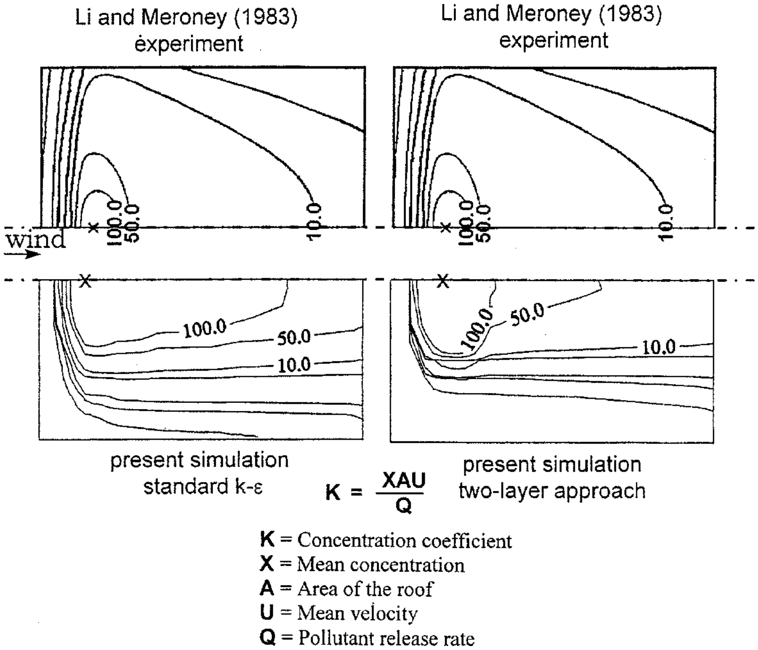

Fig. 11 Dimensionless concentration field on the roof of the cubic building for upwind roof vent (after Li 1998)

using a two-layer approach have been compared with the experimental data of Li and Meroney (1983) with rather limited success. There is no doubt that more work is required to achieve better agreement between the experiments and numerical results of concentration values.

More recently, Anagnostopoulos and Bergeles (1998) applied a new model and calculated the flow field and pollutant concentrations over complex terrain. Although turbulence closure was obtained by the simple k-ε model, a technique of partially covered cells in conjunction with the collocated grid arrangement was introduced for the representation of complex terrain features. The results reproduce reasonably well the diurnal variation of the wind and pollutant concentration fields at the mesoscale level.

5. Commercial CFD codes

Castro and Graham (1999) provided an excellent review of the current problems with the application of commercial CFD codes in the solution of problems in wind engineering and industrial aerodynamics. Due to the sophistication and generality of these codes to deal with a great variety of practical problems in an open-ended range of applications, they leave several options open to the user for selection and implementation. These options include but are not limited to the selection of the size of computational domain, the grid mesh, the numerical scheme, the boundary conditions, the turbulence model etc. This explains the different results obtained by different users for the solution of the same, even usually simple, problem! Nevertheless, there seems to be an ever-increasing confidence in the results obtained by CFD codes and more and more papers propagate the idea that the numerical wind tunnel does exist today and produces results ready to be used by practitioners. In the author’s opinion this is at best premature and at worst dangerous with the exception of very limited cases.

This is supported by Meroney *et al.* (1998), who after performing a validation exercise to determine “if a relatively robust commercial CFD package using reasonable boundary and initial conditions could be used to simulate wind engineering situations without massaging the results interactively” concluded, among others, the following :

- Upwind, side and front rooftop flow structures were replicated by the numerical models, but the wake cavity was too large.
- Mean pressure coefficients produced over the model surface were reasonably accurate, but not very sensitive to the degree of grid adaptation or turbulence model chosen.
- Concentration magnitudes about sources of tracer released in the vicinity of bluff bodies are consistently over-predicted by numerical models using conventional Reynolds-averaged type turbulence models.
- In some cases, different turbulence models actually predicted reversal of flow for the same situation.

6. Conclusions

In spite of some interesting and visually impressive results produced with CWE, the numerical wind tunnel is still virtual rather than real. Its potential however, is extremely high and its progress should be monitored carefully. Many more parallel studies - numerical and experimental - will be necessary in order to increase the present level of confidence in the computational results.

Practitioners should be warned about the uncertainties of the numerical wind tunnel results and urged to exercise caution in their utilization. Finally, more effective efforts should be made in the numerical simulation of fluctuating flow field and the numerical evaluation of peak values of variables necessary for design.

Acknowledgements

The author would like to express his appreciation to Professor Chris Baker, Chair of the 3rd International Symposium on Computational Wind Engineering for his invitation to deliver this keynote lecture. The paper was written with the assistance of Mr. T.B. Girma, graduate student, whose contribution is gratefully acknowledged.

References

Anagnostopoulos, J.S. and Bergeles, G. (1998), "A numerical model for wind field and pollutant concentration calculations over complex terrain", *J. Wind Eng. Ind. Aerod*, **73**, 285-306.

Baetke, F., Werner, H. and Wengle H. (1990), "Numerical simulation of turbulent flow over surface-mounted obstacles with sharp edges and corners", *J. Wind Eng. Ind. Aerod*, **35**, 129-147.

Bearman, P.W. and Obasaju, E.D. (1982), "An experimental study of pressure fluctuations on fixed and oscillating square-section cylinders", *J. Fluid Mech.*, **119**, 297-321.

Bergeles, G.C. (1985), "Numerical computation of turbulent flow around two-dimensional hills", *J. Wind Eng. Ind. Aerod.*, **21**, 307-321.

Bowen A.J. (1979), "Full scale measurements of the atmospheric turbulence over two escarpments", *Proc. 5th Int. Conf. on Wind Engineering*, **1**, 161-172, Pergamon.

Bowen A.J. (1983), "The prediction of mean wind speeds above simple 2D hill shapes", *J. Wind Eng. Ind. Aerod.*, **15**, 259-270.

Bowen, A.J. and Lindely, D.A. (1977), "Wind tunnel investigation of the wind speed and turbulence characteristics close to the ground over various escarpment shapes", *Boundary Layer Meteorology*, **12**, 259-271.

Bradley, E.F. (1980), "An experimental study of the profiles of wind speed, shearing stress and turbulence at the crest of a large hill", *Quart. J. R. Met. Soc.*, **106**, 121-124.

Bradshaw, P., Launder, B.E. and Lumley, J.L. (1994), "Collaborative testing of turbulence models", in: K.N. Ghia, U. Ghia, D. Goldstein (Eds.), *Advances in Computational Methods in Fluid Mechanics*, 77-82, ASME, New York.

Carpenter, P. and Locke, N. (1999), "Investigation of wind speed over multiple two-dimensional hills", *J. Wind Eng. Ind. Aerod.*, **83**, 109-120.

Castro, I.P. and Robins, A.G. (1977), "The flow around a surface-mounted cube in uniform and turbulent streams", *J. Fluid Mech.*, **79**(2), 307-335.

Castro, I.P. and Graham, J.M.R. (1999), "Numerical wind engineering: the way ahead?", *Proc. Inst. Civil Engrs, Struct. & Bldgs*, **134**, 275-277.

Chen, Y.S. and Kim, S.W. (1987), "Computation of turbulent flows using an extended k-ε turbulence closure model", *NASA*, CR 179204.

Delaunay, D., Lakehal, D. and Pierrat, D. (1995), "Numerical approach for wind load prediction on buildings and structures", *J. Wind Eng. Ind. Aerod.*, **57**, 307-321.

Durao, D., Heitor, M. and Pereira, J. (1988), "Measurements of turbulent and periodic flows around a square cross-section cylinder", *J. Fluid Mech.*, **6**, 298-304.

Ferreira, A.D., Viegas, D.X. and Sousa, A.C.M. (1999), "Numerical and experimental study of the wind flow around a group of low-rise buildings", *Proc. 10th Int. Conference on Wind Eng.*, June 21-24, Copenhagen, Denmark.

Franke, R. and Rodi, W. (1991), "Calculation of vortex shedding past a square cylinder with various turbulence models", *Proc. 8th Symp. Turbulent Shear Flows*, 189-204, Tech. Univ. Munich, Springer Berlin.

Hunt, A. (1982), "Wind-tunnel measurement of surface-mounted cube in uniform and turbulent streams", *J. Wind Eng. Ind. Aerod.*, **10**, 137-163.

Jackson, P.S. (1979), "The influence of local terrain features on the site selection for wind energy generating systems", Faculty of Engineering Science, University of Western Ontario, Report BLWT-1-1979, 83.

Kawamoto, S. (1997), "Improved turbulence models for estimation of wind loading", *J. Wind Eng. Ind. Aerod.*, **67&68**, 589-599.

Kawamoto, S. and Tanahashi T. (1994), "High-speed GSMAC-FEM for wind engineering", *Comput. Meth. Appl. Mech. Eng.*, **112**, 219-226.

Kim, S.E. and Boysan, F. (1998), "Application of CFD to Environmental Flows", Inter. Workshop on "CFD for wind Climate in Cities", Aug., 24-26, Hayama, Japan.

Lakehal, D. (1999), "Computation of turbulent shear flows over rough-walled circular cylinders", *J. Wind Eng. Ind. Aerod.*, **80**, 47-68.

Launder, B.E., Reece, G.J. and Rodi, W. (1975), "Progress in the development of Reynolds stress turbulence closure", *J. Fluid Mech*, **68**, 537-566.

Lee, B.E. (1975), "The effect of turbulence on the surface pressure field of a square prism", *J. Fluid Mech.*, **69**, 263-282.

Lee, S. (1997), "Unsteady aerodynamic force prediction on a square cylinder using k-ε turbulence models", *J. Wind Eng. Ind. Aerod.*, **67&68**, 79-90.

Lee, S. and Bienkiewicz, B. (1998), "Finite element implementation of large eddy simulation for separated flows", *J. Wind Eng. Ind. Aerod.*, **77&78**, 603-617.

Li, W.W. and Meroney, R.N. (1983), "Gas dispersion near cubical building", *J. Wind Eng. Ind. Aerod.*, **12**, 15-39.

Li, Y. (1998), "Numerical evaluation of wind-induced dispersion of pollutants around a building", Ph.D. thesis, Concordia University, Montreal, Canada.

Lyn, D., Einav, S. and Rodi, J. (1995), "A Laser Doppler velocimetery study of ensemble-averaged characteristics of turbulent wake of a square cylinder", *J. Fluid Mech.*, **304**, 285-319.

Maurizi, A. (2000), "Numerical simulation of turbulent flows over 2D valleys using three versions of the k-ε closure model", *J. Wind Eng. Ind. Aerod.*, **85**, 59-73.

Meroney, R.N., Leith, B.M., Rafailidis, S. and Schatzmann, M. (1998), "Wind-tunnel and numerical modeling of flow and dispersion about several building shapes", *Int. Workshop on "CFD for Wind Climate in Cities"*, Aug., 24-26, Hayama, Japan.

Murakami, S., Ooka, R., Mochida, A., Yoshida S. and Kim, S. (1998), "CFD analysis of wind climate from human scale to urban scale", *Int. Workshop on "CFD for Wind Climate in cities"*, Aug. 24-26, Hayama, Japan.

Murakami, S. and Mochida, A. (1988), "3-D numerical simulation of airflow around a cubic model by means of the k-ε model", *J. Wind Eng. Ind. Aerod*, **31**, 283-303.

Murakami, S. and Mochida, A. (1995), "On turbulent vortex shedding flow past 2D square cylinder predicted by CFD", *J. Wind Eng. Ind. Aerod.*, **54&55**, 191-211.

National Building Code of Canada - NBC 1995 Structural Commentaries (Part 4), Canadian Commission on Building and Fire Codes, National Research Council of Canada, Ottawa, 1995.

Ohtsuki, Y. (1976), "Wind tunnel experiments on aerodynamic forces and pressure distributions of rectangular cylinders in a uniform flow", *Proc. 5th Symp. on Wind Effects on Structures*, 169-175, Tokyo, Japan.

Paterson, D. and Apelt C. (1990), "Simulation of flow past a cube in turbulent boundary layer", *J. Wind Eng. Ind. Aerod.*, **35**, 149-176.

Paterson, D.A. and Holmes, J.D. (1993), "Computation of wind flow over topography", *J. Wind Eng. Ind. Aerod.*, **46&47**, 471-478.

Pearse, J.R., Lindely, D. and Stevenson, D.C. (1981), "Wind flow over ridges in simulated atmospheric boundary layers", *Boundary-Layer Meteorology*, **21**(1), 77-92.

Quinn, A.D., Robertson, A.P., Hoxey R.P. and Burgess L.R. (1998), "The effect of small embankments on wind speeds", *Wind and Structures*, **1**(4), 303-316.

Roullé, C. (1980), "Reproduction des régimes hypercritiques sur formes rondes en soufflerie atmosphérique", Ph.D. thesis, University of Rouen, France.

Selvam, S.P. (1997), "Computation of pressures on Texas Tech university building using large eddy simulation", *J. Wind Eng. Ind. Aerod*, **67&68**, 647-657.

Shimada, K., Takei, K, Yamamoto, S. and Sugimoto, H. (1998), "Application of 3D numerical analysis on the prediction of aerodynamic forces of a tall stack cross section", *Int. Workshop on "CFD for Wind Climate in Cities*", Aug. 24-26, Hayama, Japan.

Stathopoulos, T. (1997), "Computational wind Engineering: Past achievements and future challenges", *J. Wind Eng. Ind. Aerod.*, **67&68**, 509-532.

Stathopoulos, T. and Baskaran, A. (1996), "Computational simulation of wind environmental conditions around buildings", *Eng. Str.*, **18**, 876-885.

Stathopoulos, T. and Dumitrescu-Brulotte M. (1989), "Design recommendations for wind loading on buildings of intermediate height", *Canad. J. Civil Eng*., **16**(6), 910-916.

Tamura, T., Kawai, T., Kawamoto, S., Nozawa, K., Sakamoto, S. and Ohkuma, T. (1997), "Numerical prediction of wind engineering of wind loading on building and structures - Activities of AIJ cooperative project on CFD", *J. Wind Eng. Ind. Aerod.*, **67&68**, 671-685.

Timofeyef, N. (1998), "Numerical study of wind mode of a territory development", *Proc. 2nd East European Conf. on Wind Engineering*, Sept. 7-11, Prague, Czech Rep.

Tominaga, Y. and Mochida A. (1998), "CFD prediction of flowfield and snowdrifts around building complex in snowy region", *Intern. Workshop on "CFD for Wind Climate in Cities*", Aug., 24-26, Hayama, Japan.

Tsuchiya, M., Murakami, S., Mochida, A., Kondo, K. and Ishida, Y. (1997), "Development of new k-ε model for flow and pressure fields around bluff body", *J. Wind Eng. Ind. Aerod.*, **67&68**, 168-182.

Wright, N.G. and Easom, G.J. (1999), "Comparison of several computational turbulence models with full-scale measurements of flow around a building", *Wind and Structures*, **2**(4), 305-323.

Yakhot, V., Orzag, S.A. Thamgams, S., Gatski, T.B. and Speziale, C.G., (1992), "Development of turbulence models for shear flows by a double expansion technique", *Physics of Fluids: A: Fluid Dynamics*, **4**(7), 1510-1520.

Yu, D. and Kareem, A. (1997), "Numerical simulation of flow around rectangular prisms", *J. Wind Eng. Ind. Aerod.*, **67&78**, 195-208.

Zhang, C.X. (1994), "Numerical predictions of turbulent recirculating flows with a k-ε model", *J. Wind Eng. Ind. Aerod.*, **51**, 177-201.

Zhou, Y.S. and Stathopoulos, T. (1996), "Application of two-layer methods for the evaluation of wind effects on a cube building", *ASHRAE Trans*., **102**(1).