NRC Publications Archive Archives des publications du CNRC

Prediction of wind effects on buildings using computational methods : review of the state of the art

Baskaran, B. A.; Stathopoulos, T.

This publication could be one of several versions: author's original, accepted manuscript or the publisher's version. /
La version de cette publication peut être l'une des suivantes : la version prépublication de l'auteur, la version
acceptée du manuscrit ou la version de l'éditeur.

Publisher's version / Version de l'éditeur:

Canadian Journal of Civil Engineering, 21, 5, pp. 805-822, 1994-10-01

NRC Publications Record / Notice d'Archives des publications de CNRC:

<https://nrc-publications.canada.ca/eng/view/object/?id=a7fb75f8-7c97-4815-9f83-8ab2b24aca6d>

<https://publications-cnrc.canada.ca/fra/voir/objet/?id=a7fb75f8-7c97-4815-9f83-8ab2b24aca6d>

Access and use of this website and the material on it are subject to the Terms and Conditions set forth at

<https://nrc-publications.canada.ca/eng/copyright>

READ THESE TERMS AND CONDITIONS CAREFULLY BEFORE USING THIS WEBSITE.

L'accès à ce site Web et l'utilisation de son contenu sont assujettis aux conditions présentées dans le site

<https://publications-cnrc.canada.ca/fra/droits>

LISEZ CES CONDITIONS ATTENTIVEMENT AVANT D'UTILISER CE SITE WEB.

Questions? Contact the NRC Publications Archive team at

PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca. If you wish to email the authors directly, please see the first page of the publication for their contact information.

Vous avez des questions? Nous pouvons vous aider. Pour communiquer directement avec un auteur, consultez la première page de la revue dans laquelle son article a été publié afin de trouver ses coordonnées. Si vous n'arrivez pas à les repérer, communiquez avec nous à PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca.

ER
H1
21d
.2
LDG
o. 3996
994

National Research
Council Canada

Conseil national
de recherches Canada

Institute for
Research in
Construction

Institut de
recherche en
construction

NRC-CNRC

Prediction of Wind Effects on Buildings Using Computational Methods - Review of the State of the Art

by A. Baskaran and T. Stathopoulos

Reprinted from:
Canadian Journal of Civil Engineering
Vol. 21, pp. 805-822, 1994

(IRC Paper No. P-3996)

NRCC 38715

CISTI/ICIST NRC/CNRC
IRC Ser
Received on: 12-06-94
IRC paper

IRC paper
__Bev Creighton ANALYSE
ANALYZED

Prediction of wind effects on buildings using computational methods — review of the state of the art¹

A. BASKARAN

*Institute for Research in Construction, National Research Council of Canada, Montreal Road,
Ottawa, ON K1A 0R6, Canada*

AND

T. STATHOPOULOS

*Centre for Building Studies, Concordia University, 1455 de Maisonneuve Boulevard West,
Montreal, QC H3G 1M8, Canada*

Received June 26, 1992

Revised manuscript accepted February 7, 1994

Advancements in computer software and hardware technology provide a new direction for analyzing engineering problems. Recently the field of wind engineering has gained significant momentum in the computer modelling process. This paper reviews the state of the art in computational wind engineering, including the finite element method, finite difference method, and control volume technique. A portion of this paper summarizes the research in this area carried out by the authors. Computations have been made for a variety of building configurations, including normal wind flow conditions for a building with different aspect ratios, and modelling wind environmental conditions around groups of buildings. The computer modelling technique may eventually enhance the design of buildings and structures against wind loading and supplement the current design practice of using building codes and standards or performing experiments in wind tunnels.

Key words: buildings, computer modelling, pressure, velocity, wind engineering, wind tunnels.

Des progrès importants dans le développement des ordinateurs et des logiciels de modélisation ont été observés dans le domaine de l'aérodynamique et permettent de donner une nouvelle orientation à l'analyse des problèmes techniques. Cet article examine l'état actuel de la technologie informatique dans ce domaine, y compris la méthode des éléments finis, la méthode des différences finies et la technique du volume de contrôle. Une partie de cet article résume la recherche effectuée dans ce domaine par les auteurs. Des calculs ont été effectués pour une variété de configurations de bâtiment, y compris des conditions normales d'écoulement du vent pour un bâtiment ayant divers rapports de forme, et une modélisation des conditions environnementales autour de groupes de bâtiments a été réalisée. La technique de modélisation informatique permettra éventuellement d'améliorer la conception des bâtiments et des structures en ce qui concerne les surcharges dues au vent. Elle constitue un complément aux exigences contenues dans les codes et les normes du bâtiment et aux essais en soufflerie.

Mots clés : bâtiments, modélisation informatique, pression, vitesse, aérodynamique, souffleries.

[Traduit par la rédaction]

Can. J. Civ. Eng. 21, 805–822 (1994)

1. Introduction

As buildings become tall and complex, it becomes imperative to account for wind loads. Wind not only affects the load distribution pattern on buildings but also influences various design parameters. Wind effects on buildings are grouped in Fig. 1 under three major aspects; structural, environmental, and energy. Some of the subcategories for each stage is also listed. It is the responsibility of a building engineer to design safe and economical buildings by considering all these features.

In Canada, building designers and engineers quantify these effects in accordance with the National Building Code (NBC 1990). Its wind section has been derived mainly from the results of systematic wind-tunnel experiments. Also note that the National Building Code is one of the world's best documents available for the description of wind effects on buildings. Engineering practice in Canada has extended to fabricating and testing scale models in wind tunnels for

buildings with unique geometries or structural properties. Wind-tunnel studies on the Commerce Court Building and the CN Tower are typical examples.

In this paper, previous and current mathematical modelling of turbulent, as well as steady, wind flow conditions around buildings are reviewed. Only a few studies have modelled turbulent wind around buildings; they are presented in the next section. However, significant efforts have been made in analyzing the variety of existing studies developed to predict steady wind conditions (see Sect. 3). Attempts include application of the finite element method, the finite difference method, and the control volume method for wind engineering problems. The present review excludes many details of the equations involved and the numerical formulation, in order to emphasize the applications of the studies. These details can be found in published dissertations, such as those of Vasilic-Melling (1977), Baetke (1986), Paterson (1986), and Baskaran (1990). Section 4 provides a flavor of ongoing research by presenting computations performed for a variety of building configurations. This includes computing normal flow conditions for a building with different aspect ratios, and modelling wind environmental conditions around a group of buildings. Validation of the computed results has always been performed by comparison with the measured data from different boundary layer wind tunnels. Finally, Sect. 5 reflects

NOTE: Written discussion of this paper is welcomed and will be received by the Editor until February 28, 1995 (address inside front cover).

¹Part of this paper has been presented in a variant form at the Second Canadian Conference on Computing in Civil Engineering, Ottawa, Ontario, August 5–7, 1992.

Printed in Canada / Imprimé au Canada

FIG. 1. Perspective of wind-induced effects on buildings (Baskaran 1990).

on the future of computerized approaches for the numerical evaluation of wind effects on buildings.

2. Computing turbulent wind flow conditions

Computing the turbulent flow will provide the instantaneous pressures that are critical for the design of buildings claddings. Their prediction not only demands high computer resources but also needs advanced numerical algorithms. Two approaches have been followed for the computation of turbulent wind flow conditions around buildings: the random vortex method and the large eddy simulation. In the former case, only time-dependent Navier–Stokes equations are considered with constant fluid viscosity, whereas in the latter, the time-dependent form of the k – ϵ turbulence model is also included. This section discusses the application of these methods for wind engineering problems; in-depth details of the methods can be found in review papers by Leonard (1980) for the vortex methods and Ferziger (1983) for large eddy simulations.

2.1. Random vortex method

The random vortex method for the solution of two-dimensional Navier–Stokes equations was first developed and presented by Chorin (1968, 1973, 1978). The incompressible flow of a slightly viscous fluid around a bluff body can often be characterized by three main flow regimes (Summers et al. 1985). Away from the body, the flow is largely irrotational and may thus be formulated in terms of potential flow theory. In the immediate neighborhood of solid surfaces, a boundary layer is formed in which the Prandtl boundary layer approximation is assumed to be valid. Beyond this boundary layer and in the downstream wake of the obstacle, the flow becomes complicated. This area has concentrated vorticity generated by the viscous interaction of the fluid with the obstacle.

Summers et al. (1985) applied the random vortex method to describe the viscous interaction of wind with surface-mounted obstacles of arbitrary cross sections. A logarithmic inlet velocity profile was selected to represent the stationary aspects of the atmospheric boundary layer. In Fig. 2, results from the random vortex method are illustrated for various building roof shapes. The buildings have the same eaves height (0.5 units) and width (1 unit), whereas the roof pitch angle varies (0, 23.5, 30, and 40 degrees). Figure 2 shows the streak lines formed by massless particles entering the flow as the first frame and followed over the successive time-steps, each step corresponding to 0.02 nondimensional time unit. The separation at the windward eaves moves to the ridge as pitch angle increases and reattachment can occur on the leeward roof side.

Bui and Oppenheim (1987) presented simulated results for air flow over a cube using the random vortex method. The two-dimensional computational flow domain is divided into two regions, one inside the boundary layer, where vortex sheets simulate the potential flow, and the other outside the boundary layer, where vortex bulbs develop in the flow. Mean wind velocity vectors are evaluated for a two-dimensional square model sitting inside a channel with a uniform approach velocity. Attempts were not made to compare these results with any measured data.

Along the lines of the random vortex method, the flow around bluff bodies can also be represented by the discrete vortex method. Inamero and Saito (1985) used the discrete vortex method to study the unsteady flow around two-dimensional bodies having different aspect ratios. An extension of the discrete vortex method is also used by Inamero and Saito for modelling the flows with various angles of attack; comparisons of the computed results with the experimental data show a significant difference when the wind angle is greater than 20 degrees. By applying the discrete vortex method, Bienkiewicz and Kutz (1989) computed Strouhal numbers for square and rectangular prisms and compared them with measured wind-tunnel data. The random vortex method has been improved over the years. One important improvement is the random sheet method presented by Chorin (1978). In this method the Prandtl boundary layer equation was utilized instead of the Navier–Stokes equation in order to eliminate singularity problems. Nevertheless, the nonrotating vortex sheets cannot be valid for turbulent boundary layer flows (Chorin 1986).

Recently, a fast vortex method for computing two-dimensional viscous flows has been presented by Baden and Puckett (1990) and Turkiyyah and Reed (1993). These studies advance the state of the art of vortex methods by taking advantage of the evolution in computer hardware technology. For instance, the CPU time for the development of vector code is 10 times faster than conventional vortex methods. Ramsay (1991) also presented potential applications of rapid distortion theory in wind engineering. His approach involves the linearization of the governing equations, and it has been reported that the error due to neglecting the nonlinearity dynamics is small.

The random vortex method is advantageous for fluid flow problems because of its grid-free character. There is no need for an initial guess of the pressure field or flow separation points. Furthermore, the bluff body may have any arbitrary shape and its edges should not necessarily be sharp. However, the extension of this method to three-dimensional problems

FIG. 2. Computed flow field using random vortex method (Hanson et al. 1984).

is not trivial. The other major drawback of this method is that it is not compatible with any standard turbulence models. This creates many difficulties in predicting the recirculation zones of the wind flow field. In any event, vortex methods have not been very attractive for wind engineering applications, at least so far, because of the complexities in predicting high Reynolds number flows and other problems, as previously mentioned. Further discussion on these matters is found in Baetke (1986), Chorin (1980, 1986), Hanson et al. (1986), and Summers et al. (1986).

2.2. Large eddy simulation

The large eddy simulation solves the Navier–Stokes equation in time-dependent form and yields the fluctuating motions of turbulence. It is also possible in the simulation to analyze the turbulence statistics of the flow field at any instant. In this methodology, the main assumption is that the largest eddies dominate the physics of the turbulent process. Ferziger (1979) presented the following differences between large and small eddies:

(1) The large eddies interact strongly with the mean flow and the small eddies are created mainly by nonlinear interactions among the large eddies.

(2) Most of the transport of mass, momentum, and energy is due to the larger eddies. The small eddies dissipate fluctuations of these quantities but affect the mean properties only slightly.

(3) The structure of large eddies is very strongly dependent on the geometry and nature of the flow, whereas the small eddies are much more universal.

(4) Due to their dependence on the geometry, the larger eddies are highly anisotropic. The small eddies are much more nearly isotropic and therefore universal.

(5) The time scales of the large eddies approximate the

time scales of the mean flow. For flow past a body, the large eddy scale is approximately equal to the dimension of the body divided by the free stream velocity. The small eddies seem to be created and destroyed much more quickly.

Murakami et al. (1987) applied the large eddy simulation in two steps to compute time-dependent wind flow conditions around buildings. First, the solution procedure dealt with the development of the boundary layer and then the effect of air flow around a cubic model was analyzed. The time-dependent Navier–Stokes equations were solved by using the marker-and-cell method. All spatial derivatives were approximated by the central difference scheme. To model the turbulence, the Reynolds stress components were replaced by the large eddy simulation, as explained in Ferziger (1979), along with two Smagorinsky constants whose values were given by Laurence (1987). The execution of the algorithm takes 5–10 h in a super computer HITAC 810-20 to predict the wind velocities and turbulence intensities around a cubic model, as well as the pressure field of the flow. This study demonstrates the capability of super computers to solve the complexities of turbulent Navier–Stokes equations. Unfortunately, frequent access to super computers is not common for most designers.

Numerical modelling using the large eddy simulation can predict instantaneous wind-induced parameters, as demonstrated by Song and He (1993). Even though the incoming flow was normal to the building, both the entire flow fields around the building and the flow fields of only half of the building, due to symmetrical conditions, were modelled. Typical results are shown in Fig. 3 in terms of lift and drag coefficient fluctuations. Clearly, the fluctuations are larger when only half of the building was considered, as normally happens in the modelling of steady wind flow conditions in order to reduce the computer resource requirements.

FIG. 3. Computed fluctuating lift and drag coefficients (Song and He 1993).

FIG. 4. Two-dimensional finite element arrangement around a fence (Street and Kot 1980).

However, considering the Song and He findings, the assumption of symmetry for spatial variation would be questionable for the simulation of time-dependent wind flow conditions. Thus time-dependent wind simulation will significantly increase the computer resource requirement.

3. Computing steady wind flow conditions

3.1. Finite element method

The era of finite element methods has little affected computational wind engineering, compared to the major breakthrough it has made in the aeronautical engineering. Nevertheless, finite element methods have been utilized in a few computational wind engineering studies. As a first step in developing a three-dimensional finite element code for turbulent separating flows, Street and Kot (1980) presented a surface separation algorithm which takes the corner node of a three-dimensional bluff body immersed in turbulent flow and determines all the midpoints. Once the node location is established on the building, the optimum finite element mesh distribution around the building is formulated. The boundary conditions are imposed through a "stiffness" matrix, and the Gaussian elimination method is used for back substitution of the solution vector.

Figure 4 shows a vertical section of this mesh distribution over a two-dimensional fence immersed in a turbulent boundary layer. Another interesting feature of this algorithm is the optimization process of the flow conditions. As the flow

evolves from the initial configuration, the elements lying on the separation surface move along with it and the other elements are then properly adjusted, based on the new separation location. Yu and He (1985) attempted to compute the wind effects on a tall building using the finite element method. Triangular elements were used and pressure and velocities were computed using the same interpolating functions. The two-dimensional algorithm was found stable for various mesh arrangements and the equations were solved through the weighted residual methods of Baker (1983).

Uemura and Saito (1993) developed an automatic mesh generation algorithm to account for complex building configurations. A typical three-dimensional hexahedral mesh distribution is shown in Fig. 5. It is clear that the finite element method can accommodate more complex building shapes and orientations than the conventional finite difference formulation. Nevertheless, the freedom in applying the various numerical schemes, such as HDS and SIMPLE (Patankar 1980), is restricted. Furthermore, the extension of the finite element method into three-dimensional problems is still in the research stage and needs significant refinement before it can be applied to wind engineering problems (Habashi and Peters 1987).

3.2. Finite difference method

The finite difference method is used in most studies for the computation of wind flow conditions around buildings.

FIG. 5. Three-dimensional finite element arrangement around a building complex (Uemura and Saito 1993).

FIG. 6. Comparison of the measured velocity profiles with the predicted data (Vasilic-Melling 1977).

Computational wind engineering continues to enjoy the benefits of finite difference techniques and this trend will probably continue mainly because of two reasons: first, the finite difference method contains a wealth of established solution algorithms and numerical schemes that can be directly imported to computational wind engineering; second, these methods have the flexibility of accommodating turbulence models and time-dependent forms of the flow characteristics that are vitally important for the simulation of wind flow conditions. In finite difference method, wind flow problems can be simplified into flow over two-dimensional or three-dimensional bluff bodies; these problems are reviewed in the following subsections.

3.2.1. Two-dimensional works

One of the early attempts in numerical modelling of turbulent flow over bluff bodies was made by Vasilic-Melling (1977). Two-dimensional problems were considered first, to gain familiarity with the numerical solution procedure. Velocity and pressure predictions include (1) an infinitely wide obstruction mounted to a free stream flow, (2) a wall-mounted thin fence, and (3) a freely suspended thin fence in a uniform flow. For case (2), comparisons of the computed predictions with the available experimental data are shown in Fig. 6. The velocity profiles show large differences between the predicted and measured values within the shear layer and the recirculation zone. Underestimation of the acceleration

of the flow resulting from a block effect is also shown. Part of the discrepancies between the results near the wall may be caused by experimental errors. In particular, the measured profiles in the recirculating flow, say $z/h = 2$, show a trend to an increasing reverse velocity near the wall rather than a tendency towards zero velocity, which is expected for a condition of no slip at the wall. Benodekar et al. (1985) also used the Vasilic-Melling methodology to predict the flow over surface-mounted ribs.

Hanson et al. (1984) investigated the possibilities of developing a numerical model for wind flow problems around buildings. Their attempts included both the random vortex method and the control volume method for two-dimensional laminar flow of wind without adding the effect of turbulence intensity. The control volume method used originated from Caretto et al. (1972). Both methods were applied only to buildings with a symmetrical roof of arbitrary pitch. Attempts have not been made to compare the results with other studies or with the experimental data. However, the approach of this study brings out the advantages of the control volume method over the random vortex method.

Yeung and Kot (1985) computed turbulent flow over arbitrary two-dimensional surface mounted obstructions by using the body-fitted coordinate system. One of the main applications of body-fitted coordinate grid arrangement is modelling arbitrary geometrical configurations such as triangular and semicircular obstructions. The body-fitted coordinate

FIG. 7. Comparison of the measured pressure coefficients with the predicted data (Paterson 1990).

grid systems are generated using the method of Thompson (1980, 1986), and in this process, one coordinate line will align with the solid geometry. The two-dimensional form of the k - ϵ turbulence model is solved in closed form through the upwind difference scheme. Elliptical equations are obtained by transforming the Navier-Stokes equation into the vorticity-stream function form and solved by using the point successive over relaxation method. The computed reattachment length is found to be smaller in comparison with the experimental data of Counihan and Hunt (1974) and Woo et al. (1977). Attempts have also been made to improve the reattachment length by changing the values of the "universal constants" in the k - ϵ turbulence models. Moreover, the artificial viscosity error introduced by the upwind difference scheme was noticed and discussed.

Mathews (1985) attempted to compute the wind-induced pressure coefficients on a single building. A two-dimensional grid arrangement was utilized, and empirical values were used for the length scale and the intensity of turbulence. Mean pressure coefficients were computed for two buildings having different geometries. The velocity of the approaching flow was assumed to be uniform with height and normal to the predominant building dimension. The turbulence constants which had been proposed for the atmospheric boundary layer by Yeung and Kot (1985) were also used by Mathews (1987) and Mathews et al. (1988) for the computation of pressure coefficients on a semicircular greenhouse. The dependence of the pressure coefficients on the Reynolds number and on the exposure factor was evaluated. It was reported that the pressure coefficients are independent of Reynolds number greater than 3000. Computations were performed for different inlet boundary layer profiles; only a small variation was noted on the computed pressure coefficients.

Djilali (1987) used the TEACH code (Gosman and Pun 1974) to predict separated reattaching flow around a two-dimensional bluff rectangular section. For the laminar flow computations, two numerical schemes, namely the hybrid difference scheme and the bound skew hybrid difference scheme, are used. The results computed with the bound skew hybrid difference scheme agree better with the experimental data. So it was decided to use the bound skew

hybrid difference scheme for the turbulent flow computations with the standard k - ϵ turbulence models. However, the computed results underestimate the reattachment length by about 30%. The curvature correction of Leschziner and Rodi (1981) was applied to the turbulence models; as a result, the reattachment length, the kinetic energy, and the pressure field improved substantially. Durao et al. (1987) also noticed discrepancies between the computed and measured results when computing the two-dimensional flow around a square cross-sectional building by using the standard turbulence models with their universal constants.

Selvam (1989) performed two-dimensional computations using the stream function and vorticity method. Parameters of two full-scale studies — Jensen and Frank (1965) on Albertslud house and Marshall (1977) on a mobile home — were input to the model and the computed results were compared with those from the wind tunnel, as well as with the full-scale data. The pressures from the computer modelling were reasonably close to the measured data, except for the leeward wall. Imposing better boundary treatment as opposed to the wall functions and identifying a suitable method for predicting the pressure field were suggested for improvements.

3.2.2. Three-dimensional works

Vasilic-Melling (1977) extended the procedure for three-dimensional flows on surface-mounted cubes. A sensitivity analysis of the numerical solution was made using a uniform upstream velocity profile. Experiments on flow visualization and velocity measurements were also undertaken. Comparisons of the computed results were then made with the measured data for mean pressures only. The study concluded that the numerical simulation did not faithfully represent the shear layer and recirculating flow which develops on the downstream side of the obstacle. A better treatment of the turbulence model parameters and the use of more suitable, instead of one-dimensional, wall functions have been suggested for future work. The significance of the study by Vasilic-Melling lies in the formation of a basic solution algorithm which has been subsequently modified on several occasions in order to solve the complicated nature of the interaction between wind flow and building.

Results of three-dimensional simulations of wind flow around buildings were reported by Hanson et al. (1986). The discretized nonlinear equations of each cell was solved using a tridiagonal matrix algorithm with a successive over-relaxation factor of 1.5. No direct attempts were made to include the effects of turbulence. However, the viscosity of the incoming flow was increased by using an effective term in the numerical modelling. Since the study was concerned with the reverse-vortex flow which could develop between parallel buildings, only wind velocities were computed and compared with another study (Borges and Saraiva 1979) which measured the reverse wind velocity using sand erosion techniques. The wind speed ratios computed was also compared with the result of a wind-tunnel study (Penwarden and Wise 1973). From the comparison it was noted that the deficiency of the simulation — its failure to reproduce the compactness of the wake — could be improved by a more detailed treatment of the turbulence. Semi-empirical turbulence models such as the $k-\epsilon$ model have been suggested for further work in this area.

As a step toward validating the results obtained from mathematical models, Summers et al. (1986) used the wind tunnel data of Everett and Lawson (1984). From the various comparisons, it is clear that better computational tools are needed for the evaluation of wind effects around building configurations. Furthermore, the velocities computed at greater heights from the ground level agreed better than those computed at lower levels with respect to measured velocity values. This indicates that computational methods may provide better results for wind effects on tall buildings than on low buildings, which are sitting at a greater distortion of turbulence. Wind tunnels encounter similar difficulties in the measurement of wind effects on low buildings.

By averaging the Navier-Stokes equation with respect to time, one can obtain the Reynolds equations. Paterson (1986) made a successful attempt at solving the Reynolds equations. The effect of turbulence in the flow was conveniently accounted for by means of the standard $k-\epsilon$ turbulence models developed by Launder and Spalding (1972, 1974). The control volume method was employed to discretize the nonlinear partial differential equations. Algebraic equations were derived using the hybrid difference scheme. The alternative direction implicit method was applied for the solution of the resulting linear algebraic equations, and the SIMPLE algorithm of Patankar and Spalding (1972) was used to fulfill the condition of continuity. The computed results were compared extensively with three sets of wind-tunnel experimental data (Castro and Robins 1977, Woo et al. 1977, and Davies et al. 1979), as well as with two full-scale measurement results (Joubert et al. 1967 and Melbourne and Joubert 1971).

All computations and comparisons were made for a single building, square or rectangular in plan form, placed in a continuous flow stream. The approach flow was always maintained normal to the obstacle, which brings the advantage of symmetry along the main axis for computational modelling. Due to the discretization of the Reynolds equation only mean pressures were computed and compared with the measured data. An example is shown in Fig. 7, which compares the computed pressure coefficients with the measured data of Castro and Robins (1977). Overall, the computed results agreed with the measured data. Note that an additional term (C_{p0}) as the reference pressure coefficient was given

FIG. 8. Computed velocity field with different turbulence models: (a) wind-tunnel experiment; (b) $k-\epsilon$ model; (c) algebraic stress model; (d) large eddy simulation (Murakami 1993).

value of 0.07 to match the experimental data. For some cases, Paterson noticed neither the recirculation zone nor the pressure on the sides of the buildings seemed to be acceptable in comparison with the wind-tunnel data or full-scale values. Even for such simple cases like single objects with a normal incidence of wind flow, the computed parameters

FIG. 9. Comparison of the upstream velocities with and without turbulence models (Baskaran and Stathopoulos 1989).

need improvements, particularly at the separation of the flow and in the wake.

Wind flow conditions around a cube have been studied by Baetke (1986) on a super computer in order to evaluate the efficiency of vector codes. It was found that the CRAY machine requires only 1/10 of the computing time in comparison to the scalar computer CDC CYBER 175. The code developed can run on any machine with architecture similar to that of the CRAY. An elementary cubic-shape building was used and extensive efforts were made in predicting the wind-induced pressure values for various grid arrangements. The quadratic upstream interpolation scheme of Leonard (1979) was utilized for the numerical discretization of the momentum equations. Comparisons of the computed results with the experimental data reveal significant differences for turbulent inflow conditions (Baetke et al. 1987). Inclusion of a better alternative method than the wall function approach is recommended to predict the wall shear stress at boundaries of separating and recirculating flow regions.

By using the standard $k-\epsilon$ turbulence model, Murakami and Mochida (1988, 1989) computed the steady wind conditions around a cubic model. The standard wall functions of Launder and Spalding (1974) were used to bridge the boundary cell with the outer domain for all variables. To study the influence of boundary conditions on the numerical solution, a constant pressure field and a zero pressure field were specified and only small differences were noticed on the computed results. Mesh resolutions around the windward and leeward corners were studied. Murakami (1993) also computed the wind flow conditions by applying three different turbulence models, namely, the standard $k-\epsilon$ turbulence model, the algebraic stress model, and the large eddy simulation. Computed mean flow fields were compared with the experimental data from the wind tunnel. Figure 8 shows

the vertical planes of the vector plots. In general, all three turbulence models produced similar mean wind flow conditions around the building.

4. Current research efforts

Current research efforts into the computer modelling of wind effects on buildings can be summarized into three parts: computations with conventional turbulence modelling (Sect. 4.1), computations with improved turbulence modelling (Sect. 4.2), and computations using microcomputers (Sect. 4.3). Both the momentum and continuity equations are solved together with semi-empirical turbulence models in three-dimensional form. The control volume method is used for discretization and the hybrid difference scheme is applied for the convective term interpolation. The linearized algebraic equations are solved using the alternative direction implicit method and the SIMPLE algorithm finds the unknown pressure field in a staggered grid arrangement. To perform the computations efficiently, a new modular structure computer code named TWIST — Turbulent Wind Simulation Technique — has been developed. Details of the coding and methodologies can be found in Baskaran (1990).

4.1. Computation with conventional turbulence modelling

This section examines the importance of turbulence modelling in the computation of wind effects around buildings. This has been conveniently demonstrated by analyzing two computed cases: one without including any turbulence model and the other by incorporating the $k-\epsilon$ turbulence model. Both computed results are compared with the measured data from the boundary layer wind tunnel. In addition, this section examines the capabilities of the standard $k-\epsilon$ turbulence model and the conventional wall function approach for the boundary treatment in modelling wind environmental conditions around buildings.

FIG. 10. Comparison of the downstream velocities with and without turbulence models (Baskaran and Stathopoulos 1989).

FIG. 11. Computational domain, proximity region, and buildings considered (Stathopoulos and Baskaran 1990a).

FIG. 12. Plan and side view of the computational grid (Stathopoulos and Baskaran 1990a).

Figure 9 presents the time-averaged longitudinal velocity component, u . Data are presented as percentages of velocity ratios referenced to free stream values of a particular location and they are compared for a number of points, at two locations at the building upstream, for flow normal to the building face. Because of symmetry, both measurements and computations were made only for half of the flow domain, at two heights of approximately 5 and 16 cm, representing 10 and 31 m, respectively, in full scale. Three curves on each set represent the computed values of the present study; the computed values of Summers et al. (1986); and the wind-tunnel data, also reported by them. Baskaran and Stathopoulos (1989) included the standard turbulence model during the computation, whereas the computed values of Summers et al. (1986) were generated without considering turbulence. The measured data are obtained from a boundary layer wind tunnel. The incident wind was modelled by the power law form with the exponent equal to 0.21 and the reference height equal to 38 cm. The building block had the following dimension ratios: width:length:height = 4:2:3. Further details of the measurements are provided in Everett and Lawson (1984).

From the comparisons it is clear that the computational

tools can predict the overall nature of the flow conditions. However, deviations from the wind-tunnel values smaller than those of Summers et al. (1986) were noted by Baskaran and Stathopoulos (1989). A similar feature appears in Fig. 10, which shows the comparisons for two downstream locations in the same format as in Fig. 9. Poor agreement of both sets of computational values is evident near the building. For the downstream side, when the measurement height is 10 m from the ground level, the numerical simulations determine negative values, i.e., a reverse flow is calculated, whereas the wind-tunnel measurements show at these locations a positive (i.e., streamwise) flow. However, the values obtained in the present study are generally closer to the wind-tunnel data. As shown in the figure, the solution grids are slightly different and did not exactly correspond in the two studies, but this makes little difference in these comparisons. The pertinent question of Summers et al. (1986): "Whether the most obvious deficiency of the simulation — its failure to reproduce the compactness of the wake — could be improved by the introduction of a more detailed treatment of turbulence," is considered in Baskaran and Stathopoulos (1989) by incorporated the well-known $k-\epsilon$ models in the computation. The better agreement of these

FIG. 13. Vector plots of the computed velocity field around the buildings (Stathopoulos and Baskaran 1990a).

results with the experimental data is attributed to this additional mathematical treatment of the flow.

The feasibility of evaluating wind environmental conditions around multiple building configurations has also been examined (Stathopoulos and Baskaran 1990a). In this study, an attempt has been made to simulate the wind environmental conditions around an actual downtown location in Montreal. The cluster of buildings A, B, C, D, E, and F around the Concordia University Hall building, X, is shown in Fig. 11. These are the major buildings in the so-called proximity region, which have been modelled for the numerical computation as well as for the experimental measurements. These buildings have been assumed to have rectangular cross sections. Approximate dimensions of all these buildings and the exact locations of the points of measurements of wind speeds are also shown in the figure. The selected points are at the sidewalks or corners of the building under consideration to monitor the local wind characteristics.

A computational grid mapping of the domain is shown in Fig. 12, which displays the relative building location. Both the plan view and the side view of the buildings are shown. In the plan view, there are 82 nodes along the wind direction and 69 nodes along the lateral direction. There are 48 nodes in the vertical direction. The size of the computational domain in each direction is also shown. The grid distribution is denser in the proximity region and the arrangement contains in total 267 648 nodes. From the figure, it is also clear that the assumption of symmetrical conditions is not valid, even for the normal (SW) wind flow conditions. This makes the size of the computational domain different from the study of wind flow around a single building. Moreover, available computer resources do not permit any increase in the number of nodes. On the other hand, attempts to reduce the number of nodes to $52 \times 55 \times 42$ led to diver-

FIG. 14. Comparisons of the computed and measured velocity ratios at 2 m from the ground level (Stathopoulos and Baskaran 1990a).

gence of the computational algorithm. This is because with the reduction of the number of nodes the grid spacing increases and this creates larger discontinuities of the calculated variables, which in turn causes divergence in the computation.

FIG. 15. Comparison of the measured pressure coefficients with the predicted data (Baskaran 1990).

FIG. 16. Comparison of the streamline plots around a building (Baskaran 1990).

Figure 13 shows the plan view of the computed velocity fields around the buildings at 2 m from the ground level. This two-dimensional velocity plot takes into consideration the longitudinal, u , and lateral, v , components of the velocity vectors. Flow separation points, changes in wind directions along the two sides of the Hall building, X, and wake regions are clearly shown in the figure. The velocity components are smaller in the recirculation area and also at the centre of the building front. The changes in the wind flow direction around the surrounding buildings are also clearly indicated. Vector plots, such as those of Fig. 13, are useful as preliminary information for the designer, architect or engineer. Quick derivation by the computer and flexibility in incor-

poration of changes are sufficiently impressive to stimulate enthusiasm about computer evaluation of wind effects on buildings as opposed to the traditional wind tunnel modelling approach.

To evaluate the changes in the local wind environmental conditions, the calculated velocities are converted into conventional velocity ratios. These ratios are obtained by dividing the magnitude of the velocities around the building under consideration by the velocities in the absence of the building. Since the velocities are taken at the same height for a particular location, these ratios will directly indicate the influence of the building under consideration on the local wind conditions. Values greater than unity indicate an increase in

the velocity due to the presence of the buildings. On the other hand, ratios less than 1 indicate a reduction of local velocity when the building is added. These ratios can be further used to evaluate pedestrian comfort around building X and to examine the impact of the construction of building X, if this is a proposed building, on the wind environmental conditions in the area. The velocity ratios (amplification or reduction factors) are compared in Fig. 14 for the multiple building configuration. Magnitudes of the ratios are higher at the building corners in comparison to the other locations. The results show generally good agreement (within 30% discrepancy) with the exception of two points. Both these points are in highly complex recirculating flow regions. Consequently, neither measured nor computed values are considered accurate in these locations.

Comparisons between the computed and measured pressures on the building envelope have also been made to establish the adequacy of the computational approach. Pressures are converted into the conventional form of nondimensional pressure coefficients normalized with the dynamic pressure at the building roof height. Figure 15 shows typical comparisons for the walls. The coefficients are plotted against the ratios of z/H , where z is the height of the pressure tap or the grid location measured from the ground level and H is the height of the building. Experimental values provided in Stathopoulos and Dumitrescu-Brulotte (1990) are used as the measured wind-tunnel data. These data originate from the model of a 55 m high building tested in a turbulent boundary layer wind tunnel. Measurements from normal wind flow conditions are considered for the comparisons. Overall the computed pressure coefficients agree well with the measured values. There is little difference between the measured and computed values for the front and leeward walls. However, deviations are more significant when computed negative values are compared with suction measured on the sidewall, particularly near the bottom and the top of the wall. This discrepancy is attributed to the standard turbulence model, which may not be representative of the streamline curvature of the flow.

4.2. Computation with improved turbulence modelling

The previous section refers to computations made using the standard $k-\epsilon$ model to mimic the turbulence in the flow. The conventional wall functions have been used as boundary conditions for all variables involved in the computation. Comparisons of the computed results with respective wind-tunnel data have revealed the following:

- (1) considered mathematical equations and boundary specifications can predict overall flow features around buildings;
- (2) computed velocity fields at the downstream of the flow and the length of recirculation zones behind the building are significantly underestimated;
- (3) induced suction values on the building envelope have not been computed accurately, particularly on building sidewalls where usually the flow is complex.

Validity of the standard $k-\epsilon$ turbulence model and its constants is questionable for recirculating and separating flows (Rodi 1975; Hanjalic and Launder 1979, 1980; Rodi and Scheuerer 1986; Bernard 1986). The calculated turbulent shear stress is insensitive to the streamline curvature of the flow and hence the turbulent viscosity is not affected by the local flow behavior. In addition, only a low rate of dissipation of turbulent kinetic energy is computed when standard $k-\epsilon$ turbulence models are used. To account for these effects and to improve the predictions, two simple modifi-

FIG. 17. Computed and measured averaged pressure coefficients on the building envelopes (Baskaran 1990).

cations on the standard $k-\epsilon$ turbulence models, namely streamline curvature correction (Leschziner and Rodi 1981) and preferential dissipation correction (Gibson 1978), have been identified and applied (Baskaran and Stathopoulos 1989).

Specification of boundary conditions plays a major role in the numerical modelling process. Previous studies used the conventional wall functions to bridge the solid boundary control volumes with the outer cells. When wall functions are applied for all variables of computation (u , v , w , k , ϵ), the near wall turbulence properties are underestimated. This is due to the fact that the wall functions are developed based on one-dimensional Couette flow assumptions (Van Der

FIG. 18. CPU time requirements with different computer systems (Baskaran and Stathopoulos 1992).

Berg 1975) and they are valid only for the velocity variables (Patel et al. 1985; Launder and Shima 1989). Thus for the turbulence variables (k and ϵ), a new effective zonal treatment method has been developed, whereby the local Reynolds number is used to monitor the near wall flow region and expressions derived are based on the Taylor series expansion technique. Stathopoulos and Baskaran (1990b) reported that the results of this approach agree well with the measured data.

Figure 16 displays the flow field by means of streamline plots by using the zonal treatment and the more traditional wall functions approved. A vertical section passing through the centre of the building is shown. Both approaches utilize the same optimum grid distribution and other input parameters. The change in the flow path relative to the computational domain is clearly shown in the figure. Recirculation and distribution of the eddies are predicted reasonably well by the new approach. Moreover, the upwind vortex at the windward side of the building is shown clearly for the zonal treatment method. Furthermore, experimental results from Stathopoulos and Dumitrescu-Brulotte (1990) and Baskaran (1986) have been used to compare wind-induced pressures. Several building configurations and exposure conditions have been considered. Computed and measured pressures have been transformed into nondimensional pressure coefficient forms referenced to the respective building roof height. From the pressure coefficients computed or measured, the maximum value that has been found on each horizontal wall section has been retained. The arithmetical mean of all these values provides an average critical pressure coefficient for each wall. Thus a single parameter (i.e., average pressure coefficient) for each wall is obtained by post-processing large amounts of available data in order to judge the general accuracy of the numerical predictions.

Figure 17 shows the comparisons of such average pressure coefficients for a square building with different H/B ratios. The results computed using the new zonal treatment method are mostly in good agreement with the measured data. Clearly, encouraging improvements are obtained for the building sidewall irrespective of the building height. The

suction values measured for the leeward wall increase as H/B increases. In the computational approaches, this trend breaks down for $H/B > 2$; at $H/B = 2.4$, a significant difference can be noticed between the computed and measured values. This discrepancy may also be due to experimental uncertainties involved in the measurements, including differences between the location of pressure taps and computational grid points. In general, the zonal treatment method is preferable in comparison with the wall function approach. Nevertheless, such comparisons also show the level of agreement that is currently achieved between the measured and computed results.

4.3. Computation using microcomputers

Computational speed and the storage capacity of micros are increasing day by day, as well as their usage by the engineering community. To permit the use of this present research by professional engineers, a version of the computer code has been developed and implemented in various microcomputers. Two major problems were encountered during this process: storage capacity of the micros and floating point handling. The storage capacity of the micros limits the size of the independent array to 64K, and it reduces the total program memory size to about 565K. These restrictions will vary, depending on the memory left over after the system requirements; the authors' computations took less than 540K DOS memory. However, to overcome the restriction of 64K, the compiler data threshold option was used along with huge memory model setting. Handling the floating point depends on the math co-processor of the respective micros. The tolerance threshold values are suitably adjusted, depending on the micros.

The number of computer grid nodes plays a major role in the CPU time required for the numerical evaluation of wind effects on buildings. Keeping the extent of the computational domain constant, the number of control volumes inside the domain has been varied to establish the parameters for economical computation. Runs were made on different computer systems based on feasibility, i.e., capacity limits. Figure 18 presents the CPU time as a function of the number of grid nodes. The CPU time corresponding to microcomputers represents the direct, continuous access time. The VAX machines operate with the unique page faulting and virtual memory address technology, and their CPU time is taken under batch mode operation. Microcomputers with longer word length and higher clock speed consume less CPU time, as expected. On the other hand, the hard disk access time does not affect the CPU time because of the iterative nature of the problem. Both IPC/12 and DELL 286/20 have the same Intel 286 microprocessor but the high clock speed DELL takes less CPU time. It is also interesting to note that both DELL and AST have the same clock speed, but the AST with longer word length consumes less CPU time. Figure 18 also contains an empirical relation for the CPU time required to run under different computer systems.

Baskaran and Stathopoulos (1992) concluded the following for the computation of wind effects on buildings in different computer systems:

- (1) Microcomputers require a CPU time which is 9–36 times more than the time necessary for the same run in the VAX machines.
- (2) Among the various computational parameters, the CPU time required for computation increases quasi-linearly with the number of grid nodes.

- (3) Since the problem considered is iterative in nature, the disk access time has only marginal influence on the total CPU time required for the solution. This was true of all three computer runs examined.
- (4) Economical computations can be achieved using 32-bit machines with high clock speed.

5. Future outlook, directions, and concluding remarks

Numerical modelling of wind effects on buildings is a new technique under continuous development and can be used to complement the existing wind-tunnel experiments. None of these various computational methodologies presented in the paper have a wide range of practical applications and thus research continues unabated on all fronts. Identifying future directions in this research field is rather difficult, because of the major involvement of computer resources and their unpredictability, both in terms of hardware and software. This section simply provides a bird's eye view of some influences on the calculation of the complex wind flow conditions around buildings.

Very few attempts have been made in the calculation of unsteady turbulent wind flows in computational wind engineering. Need for extensive computing resources on one hand and lack of direct simulation schemes on the other seem to be the major constraints. Research efforts using high-level turbulence models (Murakami et al. 1987) and super computer facilities (Baetke 1986) faced problems of validating their computed results directly with experimental data. One way to overcome this problem is to concentrate on the probability distribution functions of the unsteady components, rather than on single individual peak values. Wind-tunnel and full-scale studies also treated the time-dependent peaks in this fashion (Dalglish 1971; Peterka and Cermak 1975; Stathopoulos 1980).

When reviewing the computation of steady wind flow conditions, the following numerical procedure was found to be successfully for many building configurations: control volume method for the discretization of the nonlinear partial differential equations, SIMPLE algorithm to fulfill the flow continuity, and alternative direct implicit method to solve the linear algebraic equations. A staggered grid arrangement also goes along with this package to take advantage of the unknown pressure field. However, for buildings with different shapes, the body-fitted coordinate grid systems may be appropriate (Richards 1989). To address the geometrically complex buildings, further research is necessary. Using orthogonal grid strategies or converting the complex domains into definable domains using decomposing techniques or developing suitable boundary adjustment methods with the body-fitted coordinate systems are worth investigating.

The review of turbulence closure can be found in Spalding (1982), Rodi (1984), Markatos (1986), Hunt (1988), and Leschziner (1989). One feature that is clear from these studies is that the engineering community continues to enjoy the k - ϵ turbulence model or its variants. This attraction is due to the formulation of the k - ϵ turbulence model using transport equations containing convection, dissipation, and source terms such that they can be compatible with the momentum equation in the solution procedure. This model can also predict reasonable results for engineering applications at an affordable computational cost. The authors' experience, although limited, also reveals that the k - ϵ model is generally more suitable for wind environmental applications than the

evaluation of the structural loads. Much research is needed to study the full capacity and limitations of the k - ϵ equations or their variants for the computation of wind flow conditions. The application of other forms of turbulence closure, such as Reynold stress models, algebraic stress models, and other stress-flux models, is basically in the initial stage.

Another important tissue in the numerical modelling of wind effects on buildings is validating the computed results. This is considered an integral part of model development, since only systematically validated codes can earn the confidence of designers. Wind flow mechanisms are often complex, partly because the fundamental relationships are non-linear and partly because there are many physical processes involved. Therefore when developing the code, clear knowledge of the basic flow mechanisms for simplified conditions (e.g., a single building exposed to normal wind conditions) is necessary. Experimental data are always used to evaluate computational results, although this is disputable in some cases. Nevertheless, documenting and using well-scrutinized wind-tunnel data should exclude experimental errors and differences. For low buildings, the results from the Aylesbury comparative experimental program (Sill and Cook 1989) may be used, since a comparison with the field data is often desirable. The wind engineering research facility, a collaborative program of Colorado State University and Texas Tech University supported by National Science Foundation, is also a unique resource for experimental data. For tall buildings, Dalglish and other members of the National Research Council Canada monitored the 57-story Commerce Court building during the period 1973–1980 and collected data that include wind-induced surface pressures. The computational wind engineering community can make use of these systematically collected data to validate their computer codes.

Computational fluid dynamics and its application to wind engineering are rapidly growing. In the last two or three years several symposia and congresses have taken place adding new research findings to the state of the art. In North America, a task committee has been formed in 1991 under the auspices of the Aerodynamics Committee of the American Society of Civil Engineers (ASCE) and has organized several computational wind engineering conference sessions. The first computational wind engineering conference was held at the University of Tokyo (Murakami 1992). The conference had about 230 attendees from 20 countries, presenting 92 papers in 23 sessions. For the first time, experts from computational fluid dynamics were brought together with those from wind engineering to address the complexity in numerical modelling of wind effects on buildings. There were 12 invited lectures, 4 from U.S.A., 2 each from Japan, Canada, and United Kingdom, and 1 each from Germany and France. Furthermore, the 7th U.S. National Wind Engineering Conference also had several presentations on the application of computational fluid dynamics to the wind engineering field (Hart 1993).

It is clear that the computer modelling technique may eventually enhance the design process of buildings and structures against wind loading, ventilation, and other environmental conditions, including dispersion of pollutants. This will complement the current design practice of using building codes and standards or performing experiments in wind tunnels. Experiments and computer modelling have to be carried out in parallel and to complement each other in identifying

- the critical parameters of wind effects on buildings. Initially, the computer modelling approach can be used as a predictive tool to obtain a wider range of design alternatives at reduced cost. Wind-tunnel experiments can follow up based on the finalized (reduced) options. Field monitoring, undertaken after construction, provides feedback for the validation of the computer modelling and wind-tunnel simulation approaches. The knowledge gathered can be further used for similar constructions in the future.
- Baden, S.C., and Puckett, E.G. 1990. A fast vortex method for computing 2D viscous flow. *Journal of Computational Physics*, **91**: 278–297.
- Baetke, F. 1986. Numerische berechnung der bulenten umstromung eines kubischen korpers. Ph.D. thesis, Technische Universitat Munchen, Germany.
- Baetke, F., Werner, H., and Wengle, H. 1987. Computation of turbulent flow around a cube on a vector computer. *Proceedings of the 6th Symposium on Turbulence and Shear Flows*, 9, France, pp. 20-2-1 to 20-2-6.
- Baker, A.J. 1983. *Finite element computational fluid mechanics*. Hemisphere Publications, London, England.
- Baskaran, A. 1986. Wind loads on flat roofs with parapets. M.Eng. (Building) thesis, Concordia University, Montreal, Que.
- Baskaran, A. 1990. Computer simulation of 3D turbulent wind effects on buildings. Ph.D. thesis, Concordia University, Montreal, Que.
- Baskaran, A., and Stathopoulos, T. 1989. Computational evaluation of wind effects on buildings. *Journal of Building and Environment*, **24**(4): 325–333.
- Baskaran, A., and Stathopoulos, T. 1992. Influence of computational parameters on the evaluation of wind effects on the building envelope. *Journal of Building and Environment*, **27**(1): 39–49.
- Benodekar, R.W., Goddard, A.J.H., Gosman, A., and Issa, R.I. 1985. Numerical prediction of turbulent flow over surface-mounted ribs. *AIAA Journal*, **23**(3): 359–366.
- Bernard, S.P. 1986. Limitations of the near wall k - ϵ turbulence model. *AIAA Journal*, **24**(4): 619–622.
- Bienkiewicz, B., and Kutz, F.R. 1989. Applying the discrete vortex method to flow about bluff bodies. *Proceedings of the 6th U.S. National Wind Engineering Conference*, Houston, Tex., pp. A-7-11 to A-7-20.
- Borges, A.R.J., and Saraiva, J.A.G. 1979. An erosion technique for assessing ground level winds. *Proceedings of the 5th International Conference on Wind Engineering*, Fort Collins, Colo., Vol. 1, pp. 235–242.
- Bui, T.D., and Oppenheim, A.K. 1987. Evaluation of wind effects on model buildings by the random vortex method. *Journal of Applied Numerical Mathematics*, **3**: 195–207.
- Caretto, L.S., Curr, R.M., and Spalding, D.B. 1972. Two numerical methods for three-dimensional boundary layers. *Computer Methods in Applied Mechanics and Engineering*, **1**: 39–57.
- Castro, I.P., and Robins, A.G. 1977. The flow around a surface-mounted cube in uniform and turbulent streams. *Journal of Fluid Mechanics*, **79**(2): 307–335.
- Chorin, A.J. 1968. Numerical solution of NSE. *Journal of Mathematics and Computation*, **22**: 745–762.
- Chorin, A.J. 1973. Numerical study of slightly viscous flow. *Journal of Fluid Mechanics*, **57**(4): 785–796.
- Chorin, A.J. 1978. Vortex sheet application of boundary layers. *Journal of Computational Physics*, **27**: 428–442.
- Chorin, A.J. 1980. Vortex models and boundary layer instability. *SIAM Journal on Scientific and Statistical Computing*, **1**(1): 1–21.
- Chorin, A.J. 1986. Vortex methods and turbulence theory. Report No. PAM-324, Center for Pure and Applied Mathematics, University of California, Berkeley, Calif.
- Counihan, J., and Hunt, J.C.R. 1974. Wakes behind two-dimensional surface obstacles in turbulent boundary layer. *Journal of Fluid Mechanics*, **64**(3): 529–563.
- Dalglish, W.A. 1971. Statistical treatment of peak gusts on cladding. *ASCE Journal of the Structural Division*, **97**(ST9): 2173–2187.
- Davies, M.E., Quincey, V.G., and Tindall, S.J. 1979. The near-wake of a tall building block in uniform and turbulent flows. *Proceedings of the 5th International Conference on Wind Engineering*, Fort Collins, Colo., Vol. 1, pp. 289–298.
- Djilali, N. 1987. An investigation of two-dimensional flow separation with reattachment. Ph.D. thesis, Department of Mechanical Engineering, The University of British Columbia, Vancouver, B.C.
- Durao, D.F.G., Heitor, M.V., and Pereira, J.F. 1987. The turbulent flow in the near-wake of a squared obstacle. *Proceedings of 1987 ASME Applied Mechanics, Bio-Engineering and Fluids Engineering Conference*, Cincinnati, Ohio, pp. 45–50.
- Everett, T.W., and Lawson, T.V. 1984. Wind-tunnel measurements of pressure and velocity around a simple building in a turbulent shear flow to allow validation of values derived from a computer solution of the Navier–Stokes equations. Department of Aeronautical Engineering, Report No. TVL/8401, Bristol University, Bristol, England.
- Ferziger, J.H. 1979. Large eddy numerical simulation of turbulent flows. *AIAA Journal*, **15**(9): 261–267.
- Ferziger, J.H. 1983. Higher level simulations of turbulent flows. *In Computational methods for turbulent, transonic and viscous flows*. Edited by J.A. Essers. pp. 93–181.
- Gentzsch, W. 1987. Vectorialization of computer programs applied to CFD. Notes on Numerical Mechanics, 8, Fredr, Vieweg and Sohn, Braunschweig, Germany.
- Gibson, M.M. 1978. An algebraic stress and heat-flux model for turbulent shear flow with streamline curvature. *International Journal of Heat and Mass Transfer*, **21**: 1609–1617.
- Gosman, A.D., and Pun, W.M. 1974. KASE problems for the TEACH computer programs. Report No. HTS/74/3, Department of Mechanical Engineering, Imperial College of Science and Technology, London, England.
- Habashi, W.G., and Peters, M.F. 1987. Finite element solution of the compressible NSE. *AIAA Journal*, **25**(7): 944–948.
- Hanjalic, K., and Launder, B.E. 1979. Preferential spectral transport by irrotational straining. *ASME, Turbulent Boundary Layer*, **102**: 101–109.
- Hanjalic, K., and Launder, B.E. 1980. Sensitizing the dissipation equation to irrotational strains. *Transactions of ASME, Journal of Fluids Engineering*, **102**: 34–40.
- Hanson, T., Summers, D.M., and Wilson, C.B. 1984. Numerical modelling of wind flow over buildings in two dimensions. *International Journal of Numerical Methods and Fluids*, **4**: 25–41.
- Hanson, T., Summers, D.M., and Wilson, C.B. 1986. A three-dimensional simulation of wind flow around buildings. *International Journal of Numerical Methods and Fluids*, **6**: 113–127.
- Hart, G.C. (Editor). 1993. *Proceedings of the 7th U.S. National Conference on Wind Engineering*, University of California, Los Angeles, Calif., June 27–30.
- Hunt, J.C.R. 1988. Studying turbulence using direct numerical simulation: 1987. Center for Turbulence, NASA Ames/Stanford Summer Programme. *Journal of Fluid Mechanics*, **190**: 375–392.
- Inamero, T., and Saito, T. 1985. A numerical analysis of unsteady separated flow by discrete vortex method. *Proceedings of the 5th U.S. National Conference on Wind Engineering*, Lubbock, Tex., Vol. 5, pp. B-17-5 to B-17-24.
- Jensen, M., and Frank, N. 1965. Model scale test in turbulent wind, Part II. The Danish Technical Press, Copenhagen, Denmark.
- Joubert, P.N., Stevens, L.K., Good, M.C., Hofmann, E.R., and

- Perry, A.E. 1967. The drag of bluff bodies immersed in a turbulent boundary layer. *Proceedings of International Research Seminar, Wind Effects on Buildings*, Ottawa, Ont., Vol. 1, pp. 297–330.
- Lauder, B.E., and Shima, N. 1989. Second-moment closure for the near-wall sublayer: development and application. *AIAA Journal*, **27**(10): 1319–1325.
- Lauder, B.E., and Spalding, D.B. 1972. *Lectures in mathematical model of turbulence*. Academic Press, London, England.
- Lauder, B.E., and Spalding, D.B. 1974. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, **3**: 269–289.
- Laurence, D. 1987. Turbulence modelling through one point closures — applications — enlighting by large eddy simulation. *Special Course on Modern, Theoretical and Experimental Approches to Turbulence Flow Structure and its Modelling*, Belgium, AGARD 755, pp. 2-1 to 2-22.
- Leonard, A. 1980. A review — vortex methods for flow simulation. *Journal of Computational Physics*, **37**: 289–335.
- Leonard, B.P. 1979. A stable and accurate convective modelling procedure based on quadratic upstream interpolation. *Computer Methods in Applied Mechanics and Engineering*, **19**: 59–98.
- Leschziner, M.A. 1980. Practical evaluation of three finite differences schemes for the computation of steady-state recirculating flows. *Computer Methods in Applied Mechanics and Engineering*, **23**: 293–312.
- Leschziner, M.A. 1989. Modelling turbulent recirculating flows by finite volume methods — current status and future directions. *International Journal of Heat and Fluid Flow*, **10**(3): 186–202.
- Leschziner, M.A., and Rodi, W. 1981. Calculation of annular and twin parallel jets using various discretization schemes and turbulence-model variations. *Transactions of ASME, Journal of Fluids Engineering*, **103**: 352–360.
- Markatos, N.C. 1986. The mathematical modelling of turbulent flows. *Journal of Applied Mathematical Modelling*, **10**: 190–217.
- Marshall, R.D. 1977. The measurement of wind loads on a full-scale mobile home. NBSIR 77-1289, National Bureau of Standards, Washington, D.C.
- Mathews, E.H. 1985. The prediction of natural ventilation in buildings. Ph.D. dissertation, University of Potchefstroom, Transvaal, South Africa.
- Mathews, E.H. 1987. Prediction of the wind generated pressure distribution on buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, **25**(2): 219–221.
- Mathews, E.H., Crosby, C.P., Visser, J.A., and Meyer, J.P. 1988. Numerical prediction of wind loads on buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, **31**: 241–250.
- Melbourne, W.H., and Joubert, P.N. 1971. Problems of wind flow at the base of tall buildings. *Wind Effects on Buildings and Structures*, *Proceedings of the 3rd International Conference on Wind Effects on Buildings*, Tokyo, Japan, pp. I.11-1 to I.11-10.
- Murakami, S. (Editor). 1992. *Proceedings of the First International Symposium on Computational Wind Engineering*, University of Tokyo, Tokyo, Japan, August.
- Murakami, S. 1993. Comparison of various turbulence models applied to bluff body. *Journal of Wind Engineering and Industrial Aerodynamics*, **46**: 21–36.
- Murakami, S., and Mochida, A. 1988. 3-D numerical simulation of air flow around a cubic model by means of the $[k-\epsilon]$ model. *Journal of Wind Engineering and Industrial Aerodynamics*, **31**: 283–303.
- Murakami, S., and Mochida, A. 1989. Three-dimensional numerical simulation of turbulent flow around buildings using $k-\epsilon$ turbulence model. *Journal of Building and Environment*, **24**(1): 51–64.
- Murakami, S., Mochida, A., and Hibi, K. 1987. Three-dimensional numerical simulation of air flow around a cubic model by means of large eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics*, **25**: 291–305.
- NBC. 1990. National building code of Canada. National Research Council of Canada, Ottawa, Ont.
- Patankar, S.V. 1980. *Numerical heat transfer and fluid flow*. McGraw-Hill, London, England.
- Patankar, S.V., and Spalding, D.B. 1972. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal of Heat and Mass Transfer*, **15**: 1787–1806.
- Patel, M.K. 1982. On the false diffusion problem in the numerical modelling of convection-diffusion processes. Ph.D. thesis, Thames Polytechnic, London, England.
- Patel, V.C., Rodi, W., and Scheuerer, G. 1985. Turbulence models for near-wall and low Reynolds number flows: a review. *AIAA Journal*, **23**(9): 1308–1319.
- Paterson, D.A. 1986. Computation of wind flows over three-dimensional buildings. Ph.D. thesis, Department of Civil Engineering, University of Queensland, Australia.
- Paterson, D.A., and Apelt, C.J. 1989. Simulation of wind flow around three-dimensional buildings. *Journal of Building and Environment*, **24**(1): 39–50.
- Penwarden, A.P., and Wise, A.F.W. 1973. *Wind environment around buildings*. BRE Report, Her Majesty's Stationery Office, London, England.
- Peterka, J.A., and Cermak, J.E. 1975. Probability distribution of local peak pressure. *Proceedings of the 4th U.S. National Conference on Wind Engineering*, pp. 141–147.
- Ramsay, S.R. 1991. Application of rapid distortion theory in wind engineering. *Proceedings of the 9th Structures Congress*, Indianapolis, Ind., pp. 721–724.
- Richards, P.J. 1989. Computational modelling of wind flow around low-rise buildings using PHOENICS. Report No. DN1508, AFRC Institute of Engineering Research, Wrest-Park, Silsoe, Bedford, England.
- Rodi, W. 1975. A note on the empirical constant in the Kolmogorov-Prandtl eddy-viscosity expression. *Transactions of ASME, Journal of Fluids Engineering*, **97**: 386–389.
- Rodi, W. 1982. Examples of turbulence models for incompressible flows. *AIAA Journal*, **20**(7): 872–879.
- Rodi, W. 1984. Turbulence models and their applications in hydraulics — a state-of-the-art review. *International Association for Hydraulic Research*, Delft, Netherlands.
- Rodi, W., and Scheuerer, G. 1986. Scrutinizing the $k-\epsilon$ turbulence model under adverse pressure gradient conditions. *Transactions of ASME, Journal of Fluids Engineering*, **108**: 174–179.
- Selvam, R.P. 1989. Computer simulation of wind load on a house. *Proceedings of the 6th U.S. National Conference on Wind Engineering*, Houston, Tex., pp. A-7-21 to A-7-28.
- Sill, B.L., and Cook, N.J. 1989. The Aylesbury comparative experiment: a status report. *Proceedings of the 6th U.S. National Conference on Wind Engineering*, Houston, Tex., pp. B7–11.
- Song, C.S.C., and He, J. 1993. Computation of wind flow around a tall building with large eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics*, **46**: 219–228.
- Spalding, D.B. 1982. *Turbulence models, A lecture course*. Report No. CFD/82/4, Computational Fluid Dynamics Unit, Imperial College of Science and Technology, London, England.
- Stathopoulos, T. 1980. PDF of wind pressure on low-rise buildings. *ASCE Journal of Structural Engineering*, **106**(ST5): 973–990.
- Stathopoulos, T., and Baskaran, A. 1990a. Computer simulation of wind environmental conditions around buildings. *Annual Conference, Canadian Society for Civil Engineering*, Hamilton, Ont.
- Stathopoulos, T., and Baskaran, A. 1990b. Boundary treatments for the computation of 3-D wind flow conditions around a building. *Journal of Wind Engineering and Industrial Aerodynamics*, **35**: 176–200.

- Stathopoulos, T., and Dumitrescu-Brulotte, M.L. 1990. Design recommendations for wind loading on buildings of intermediate height. *Canadian Journal of Civil Engineering*, **16**: 910-916.
- Street, R.L., and Kot, Y.P. 1980. Numerical simulation of three-dimensional separating flows: a first step. *Proceedings of the 1st International Conference on Environmental Forces on Engineering Structures*, Imperial College, London, England, pp. 19-37.
- Summers, D.M., Hanson, T., and Wilson, C.B. 1985. A random vortex simulation of wind-flow over a building. *International Journal of Numerical Methods and Fluids*, **5**: 849-871.
- Summers, D.M., Hanson, T., and Wilson, C.B. 1986. Validation of a computer simulation of wind flow over a building model. *Journal of Building and Environment*, **21**(2): 97-111.
- Thompson, J.F. 1980. Numerical solution of flow problems using body fitted coordinate system. *In Computational fluid dynamics. Edited by W. Kollmann*. Hemisphere, Washington, D.C.
- Thompson, J.F. 1986. Composite grid generation code for general 3D regions — the EAGLE codes. *AIAA Journal*, **26**(3): 271-272.
- Turkiyyah, G.M., and Reed, D.A. 1993. A computational modeling environment for wind engineering. *Proceedings of the 7th U.S. National Conference on Wind Engineering*, University of California, Los Angeles, Calif., June 27-30.
- Uemura, K., and Saito, T. 1993. Automatic mesh generation for FEM simulation of wind flow around tall buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, **46**: 357-362.
- Van Der Berg, B. 1975. A three-dimensional law of the wall for turbulent shear flows. *Journal of Fluid Mechanics*, **70**(1): 139-160.
- Vasilic-Melling, D. 1977. Three-dimensional turbulent flow past rectangular bluff bodies. Ph.D. thesis, Imperial College of Science and Technology, University of London, England.
- Williamson, G.T. (Editor). 1991. *Proceedings: Structural Congress '92*, American Society of Civil Engineers, New York, N.Y.
- Woo, H.G.C., Peterka, J.A., and Cermak, J.C. 1977. Wind tunnel measurements in the wakes of structures. Report No. JE40, Department of Civil Engineering, Colorado State University, Fort Collins, Colo.
- Yeung, P.K., and Kot, S.C. 1985. Computation of turbulent flows past arbitrary two-dimensional surface mounted obstructions. *Journal of Wind Engineering and Industrial Aerodynamics*, **18**(2): 177-190.
- Yu, X., and He, G. 1985. An efficient method for the computation of wind effects on tall buildings. *Proceedings of the 3rd International Conference on High Rise Structures*, pp. 718-722.