Engineering Structures, Vol. 18, No. 11, pp. 876–885, 1996

0141-0296/96 $15.00 + 0.00
0141-0296(95)00155-7

Computer simulation of wind environmental conditions around buildings

T. Stathopoulos

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

B. A. Baskaran

Institute for Research in Construction, National Research Council of Canada, Ottawa, Ontario, Canada, K1A 0R6

Mean wind environmental conditions around buildings necessary for the assessment of pedestrian comfort, dispersion of pollutants, as well as snow and dust transport have been evaluated numerically. Modified Navier–Stokes equations and k-ε turbulence models used in the simulation are discretized by the control volume method. The SIMPLE algorithm is applied to fulfill the condition of continuity. A non-uniform staggered grid arrangement containing 235,000 nodes is utilized for the three-dimensional numerical modelling. A typical Montreal location near the downtown campus of Concordia University has been selected as a test case for the computation. Validation of the computed results has been carried out by using data from experiments conducted in the boundary layer wind-tunnel of the Centre for Building Studies at Concordia University. Computed and measured data indicate that the most significant features of the wind environmental conditions around buildings can be predicted with reasonable accuracy. The advantages and drawbacks of computer simulation are discussed in the paper.

Keywords: building, comfort, design, environment, simulation, velocity, wind

Introduction

In modern cities, older low buildings are usually replaced by tall residental and commercial buildings. This replacement changes not only the outlook of a city, but also creates local wind environmental conditions which are unpleasant and sometimes dangerous for the pedestrians walking around the new buildings. Appropriate by-laws relevant to wind conditions in the urban environment have already been implemented in some cities. The developers may have to carry out wind-tunnel model studies or to consult with a wind engineer for any new construction, in order to ensure that the proposed buildings will not induce unpleasant or dangerous pedestrian-level wind conditions in their vicinity. Conducting wind-tunnel studies may be expensive and time consuming, particularly in the event of additional test requirements with modified building and/or environmental configurations. On the other hand, a technique based on computer simulation of wind flow conditions around buildings appears attractive as a potential alternative tool. However, mean wind flow conditions around simple building configurations can only be considered at present.

In fact, a literature survey shows that few studies have been made on the numerical simulation of wind flow conditions around buildings. Most of the studies carried out in this area[1–6] considered only the wind flow around a single rectangular building for wind direction perpendicular to the wall of the building. Hanson *et al.*[7] attempted predicting the wind flow fields between two parallel buildings without appropriate modelling of turbulence conditions. Haggkvist *et al.*[8] used the commercially available software PHOENICS developed by Spalding[9] for the non-quantitative study of wind flow conditions around a house surrounded by a group of similar houses. Significant differences found between computed results and measured data were attri-

Table 1

ϕ	Γ_ϕ	S	Equation
u,v,w	ν_t	$-\frac{\partial p}{\partial x_i}$	(2),(3),(4)
k	$\frac{\nu_t}{\sigma_k}$	$G - \varepsilon$	(5)
ε	$\frac{\nu_t}{\sigma_\varepsilon}$	$(C_1 G - C_2 \varepsilon)\frac{\varepsilon}{k}$	(6)

buted to the specification of the boundary conditions during the usage of PHOENICS. Murakami and Mochida[5] have also presented the flow field around a group of buildings. However, no attempts were made to validate the computed results by comparison with wind tunnel test data. In contrast, Gadilhe *et al.*[10] did compare the numerically-simulated flow in urban space with wind tunnel data, and attributed the differences to measurement difficulties. Finally, Takakura *et al.*[11] have found that coloured two-dimensional and three-dimensional visualizations of wind flow fields around buildings are effective.

The present study examines the feasibility of extending the development of a software tool that could eventually be used by architects and practicing engineers for the approximate evaluation of wind environmental conditions around buildings. Therefore, the simulation of the wind flow field around a downtown location of Montreal has been attempted. Detailed measurements were also conducted in a boundary layer wind tunnel with a model of the same building configurations as those used for the numerical modelling. The paper discusses the results of the numerical and the experimental study as well as the advantages and drawbacks of the computational approach.

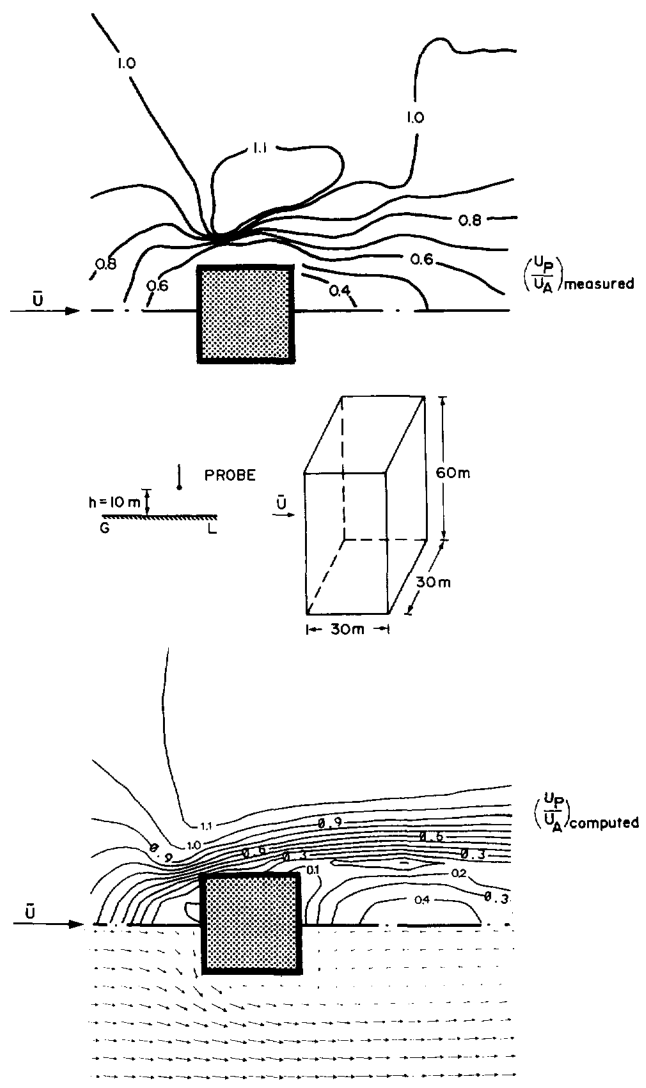


Figure 1 Computed and measured velocity field around a prismatic building

Computational approach

A computer program named TWIST (turbulent wind simulation technique) was developed in a previous study of wind-induced conditions around a single building; the operation of TWIST and details about its structure can be found in Baskaran[12]. The three-dimensional, time-averaged Navier–Stokes equations (NSE), the continuity equation and the standard k-ε turbulence models have been used in the numerical modelling. The differential equations for the computation of turbulent wind flow conditions around buildings are represented in compact form as follows:

$$U_j \frac{\partial \phi}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\Gamma_\phi \frac{\partial \phi}{\partial x_j}\right] + S\,. \tag{1}$$

The individual terms of the above equation are shown in *Table 1.*

From these terms, we obtain

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \tag{7}$$

Note that U_j is the velocity vector having components u, v and w in three directions x, y and z, respectively. The fluid turbulent viscosity, ν_t, is calculated by using the turbulence kinetic energy k and its dissipation rate ε along with some standard constants (Rodi[13]).

Details for transforming differential equations (2)–(6) into difference form by using the control volume method of Caretto *et al.*[14] have been reported elsewhere (Vasilic-Melling[1] and Paterson[3]). The final algebraic form of the discretized equation can be written as follows:

$$a_P\,\phi_P = \left[\sum_{m=1}^{n} a_m\,\phi_m\right] + S_L\,, \tag{8}$$

in which ϕ_m is the dependent variable u, v, w, p, k and ε; P is the grid node on which the dependent variable ϕ is computed; n is the number of nodes surrounding P; a_P is the hybrid difference scheme coefficient; and S_L is the linearized source term.

The well-known SIMPLE algorithm of Patankar[15] is used to correct the velocity field and also, to improve the initially assumed pressure field. The boundaries of the computational domain and the boundaries of the building envelope are placed on the velocity nodes of the staggered grid. In this way specification of pressure values on the boundaries is not required. The standard wall function approach of Launder and Spalding[16] is used to bridge the boundary nodes with the outer computational nodes for all other variables.

Figure 2 Downtown Montreal highlighting the region of the building under consideration

Figure 1 compares the computed velocity ratios around a single rectangular building located in an open country terrain with respective measured data. Velocity ratios in a plane parallel to the ground at 10 m height are considered. Experimental data originate from a wind-tunnel study conducted to evaluate the wind environmental conditions around tall buildings (Stathopoulos[17]). The building considered is 60 m high with a square cross-section of 30 × 30 m exposed to normal wind flow conditions. Due to symmetry, only half of the flow domain is shown in the figure. In addition to the velocity ratios u_P/u_A, in which u_P is the velocity in the presence and u_A is the velocity in the absence of the building, the computed velocity vectors are also included in the figure. Separations of the flow from the building surfaces and re-circulations behind the building are clearly shown. The computed velocity ratios generally compare well with the measured data, but differences also exist. Even though the wind flow conditions around a single building appear geometrically simple, velocities in the developed wakes are difficult to measure, as well as to compute, because of the increased turbulence in these areas. Measurements in pulsating flow conditions tend to overestimate the actual velocities and computations may yield errors since the k-ε turbulence model is not generally representative of wake conditions. This may explain some differences between experimental and computational data.

Multiple building configuration

The approach described in the previous section was also followed by the evaluation of wind conditions around a cluster of buildings, i.e. a more realistic case. A downtown Montreal region, in which the central (Hall) building of Concordia University is located, has been selected for this purpose. *Figure 2* shows the area under consideration as well as the wind direction assumed in the analysis. This direction reflects SW winds.

Figure 3 shows probability estimates for the exceedance of mean hourly wind speed from different directions at a height equal to 300 m above Montreal. The estimates originate from data measured at Dorval airport at 10 m height for a 10-year period during winter daylight hours. It is apparent from the figure that southwesterly winds are the strongest, followed by northeasterly winds. Assuming the same conditions prevail for downtown locations the SW winds are likely to be critical for wind environmental studies.

The cluster of buildings A, B, C, D, E and F around the Hall building X under consideration is shown in detail in *Figure 4*. 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 and several details regarding their configuration have been omitted. Approximate dimensions of all these buildings and the location of the points of measurements of wind speeds are also shown in this figure. Furthermore, the boundaries of the computational domain are indicated. The size of the domain is based on previous experience of computing wind conditions around a single building (Baskaran and Stathopoulos[6]). The domain starts from Guy and ends at Drummond street and for the lateral direction extends from north of Ste Catherine to Sherbrooke street.

A computational grid mapping of the domain is shown in *Figure 5* which displays the relative building location as well. 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 68 nodes along the lateral direction. The non-uniform distribution of the grid points is clear. In the side view there are 48 nodes in the vertical direction. The size of computational domain in each direction is also shown. The grid contains in total 267,648 nodes. Available comptuer resources at present do not permit any increase

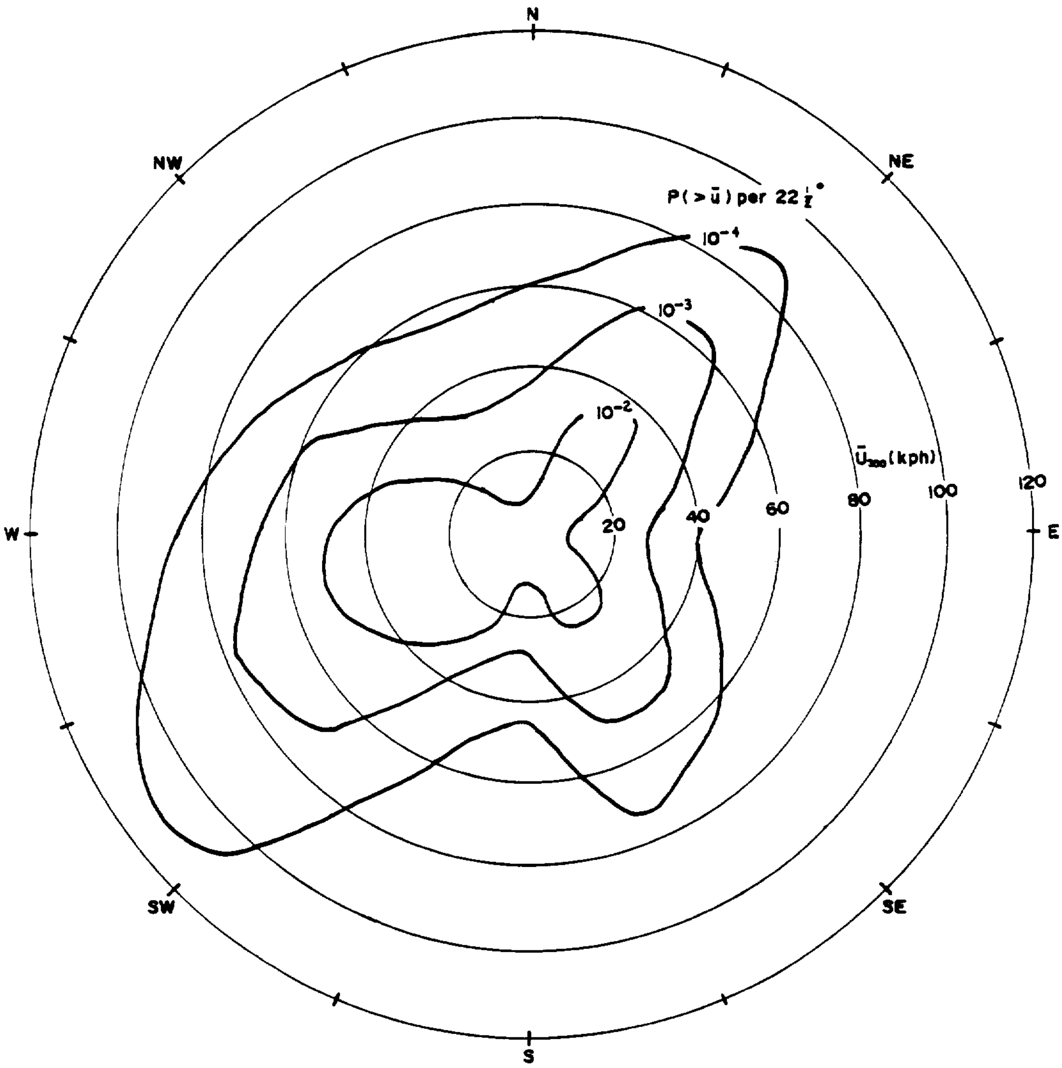


Figure 3 Probability distributions of hourly mean wind speed at 300 m over Montreal for daylight hours (07:00–19:00) during the winter. (Derived from data obtained at 10 m height at Dorval Airport for the period 1974–1983)

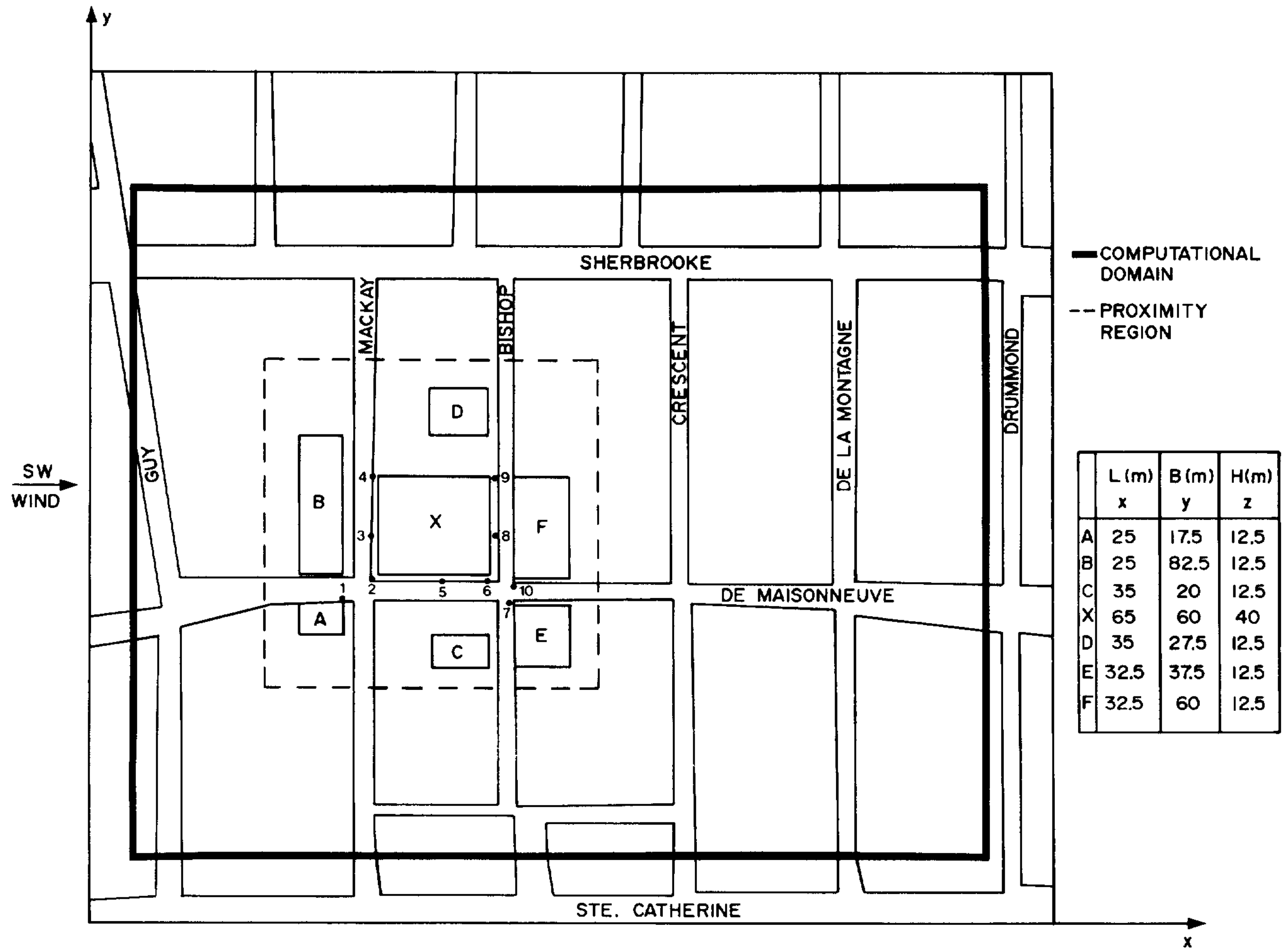
in the number of nodes. However, attempts made to reduce the number of nodes to 52 × 55 × 42 led to divergence of the computational algorithm. This is due to the larger discontinuities of the calculated variable created by the increase of grid spacing. It is worth noting that the implemented algorithm converged after 61 iterations by taking around 10 h of CPU time in the VAX 11/785 (1.2 MIPS) computer system under batch mode operation.

Computed results and discussion

Figure 6 shows the plan view of the computed velocity fields around the buildings at the usual pedestrian level of 2 m from ground. This two-dimensional velocity plot was obtained by considering the longitudinal u and lateral v components of the velocity vector. The arrows in the figure indicate both the magnitude and direction of the velocity for the respective location of the building surroundings. Changes in wind direction along the two sides of the Hall building X, flow separation areas, and wake regions are clearly shown in the figure. The velocity components are smaller in the recirculation area and also at the center of the building front. The changes in the wind flow direction around the surrounding buildings are also clearly indicated.

A vector plot, such as that of *Figure 6*, is useful as preliminary information for the designer-architect or engineer. Derivation by the computer and flexibility in incorporation of changes in the building arrangements and/or geometries are features sufficiently impressive to stimulate enthusiasm about computer evaluation of wind effects on buildings, as opposed to the traditional physical modelling approach. On the other hand this enthusiasm should be viewed with caution, particularly in the short term, because of the difficulties that currently exist in the numerical simulation of small scale flow features associated with separation and re-attachment phenomena near the building envelope. In addition, it should be recalled that building shapes and dimensions are only approximate in the present approach. Currently, the computational methodology can only be used as a guide for further research and study.

To evaluate the changes in the local wind environmental conditions, the calculated velocities are converted into conventional velocity ratios such as those presented in *Figure 1* for the single building. These ratios are obtained by dividing the magnitude of the velocities around the building under consideration by the velocities in the absence of these buildings. Since the velocities are taken at the same height, the ratios directly indicate the influence of the buildings on the local wind environmental conditions. Values greater than unity indicate an increase in the wind speed due to



	L (m) x	B (m) y	H (m) z
A	25	17.5	12.5
B	25	82.5	12.5
C	35	20	12.5
X	65	60	40
D	35	27.5	12.5
E	32.5	37.5	12.5
F	32.5	60	12.5

Figure 4 Computational domain, proximity region and buildings considered

the presence of the buildings. Ratios less than one indicate reduction of local velocity when the buildings are there. Contours of velocity ratios computed at a height of 2 m from the ground are shown in *Figure 7*, which displays high velocity ratio values along the two sides of the Hall building with a maximum value of 1.6 at one corner. A 40% increase is also found near building D (top of the figure). These ratios can be further used to evaluate pedestrian comfort around the Hall building.

Experimental measurements

To validate computed results, such as those discussed in the previous section, experiments were conducted at the boundary layer wind tunnel of the Centre for Building Studies, Concordia University. The wind tunnel is 12 m long and has a working section of 1.8×1.8 m with a roof of adjustable height. The roughness of the wind-tunnel floor can be changed in order to represent different kinds of exposure conditions, such as open country, suburban and urban environments. Any desired wind direction can be obtained by rotating the turntable of the working section, where the model buildings are set. Further details of the wind tunnel parameters and information regarding the simulation conditions can be found in the paper by Stathopoulos[18].

A typical experimental set-up used for velocity measurements is shown diagrammatically in *Figure 8*. The temperature of the circulating air is monitored and the tunnel operates only if the variation of temperature during an experiment does not exceed ± 1°C. A vertical probe with a hot film sensor connected to the anemometer (type TSI 1034) is used for the measurements. The location of the probe is controlled by a traverse gear control arrangement. Collected velocity signals are passed through a lowpass filter set at 200 Hz. The filtered signal is then analyzed in an IBM 286/12 (AT) computer system with a data acquisition board. The signal is discretized at the rate of 500 samples sec^{-1} over a sampling period of 30 sec, during which various statistics such as maximum, minimum, mean and rms velocity values are stored in a file. Collected data files are then transfered to the VAX 11/785 computer for further processing by using the KERMIT network system. The data analysis may also be carried out by using the IBM-AT system, but it will be made at a much slower rate.

In the present study a suburban exposure is considered appropriate with the velocity and turbulence intensity profiles shown in *Figure 9*. The normalized velocity and height from the ground level can be fitted by the conventional power law profile with exponent 0.25. The turbulence intensity is maximum near the ground with a value higher than 20% and reduces to about 5% at the gradient height level. The wind tunnel operates using its maximum speed of 13 m sec^{-1}. Based on simulation criteria a geometric scale of 1:500 is used to fabricate the wooden models representing the buildings under consideration. The fabricated models are glued on a masonite board in their appropriate location and the set-up is placed at the center of the wind

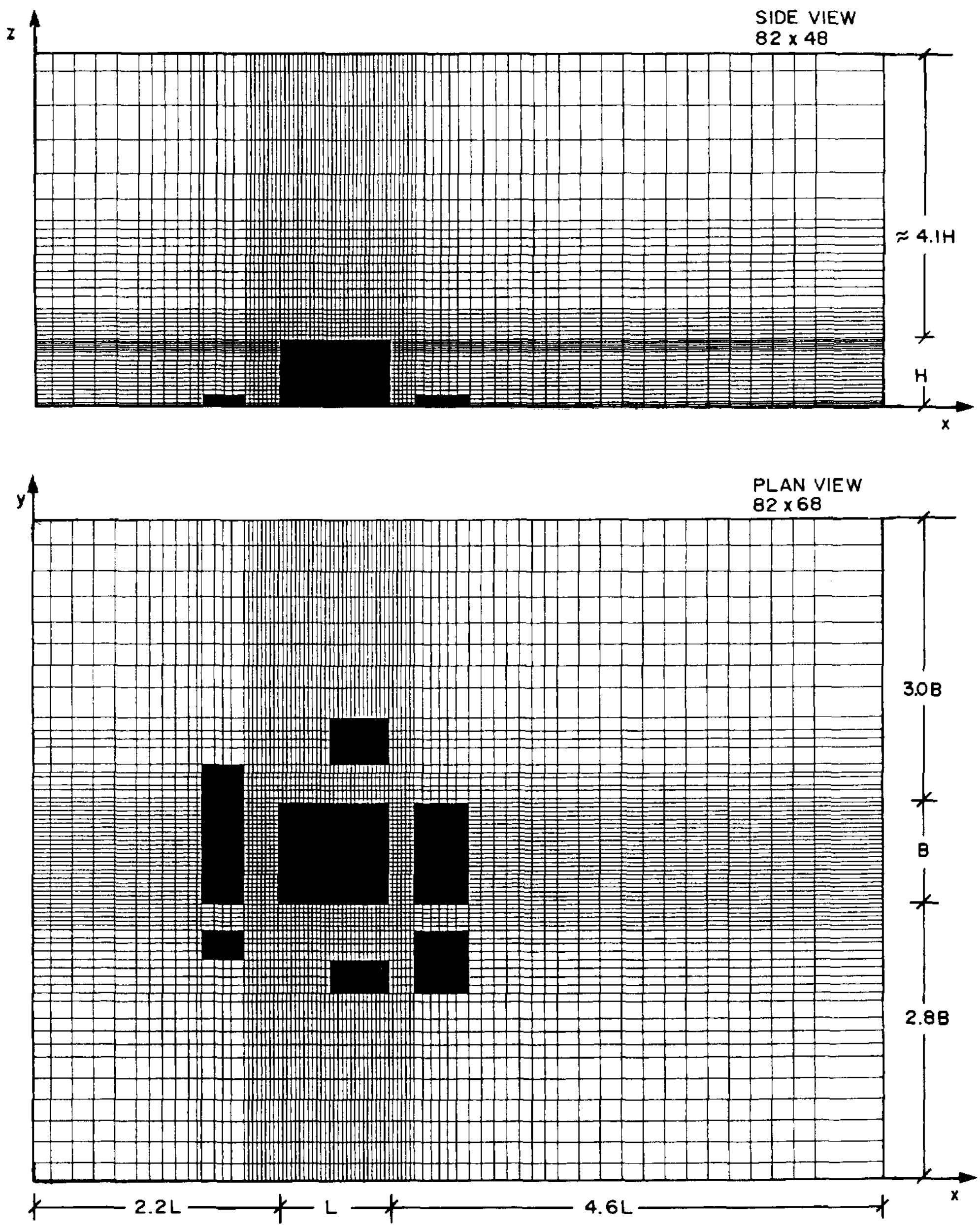


Figure 5 Plan and side view of the computational grid

tunnel working section, such that the Hall Building *X* lies at the center of the turntable. Note that only major features are considered for the representation of the buildings and details are not included. A variety of low-rise buildings on each side of the Hall Building have been replaced by a single long low-rise structure and the height of all surroundings buildings is assumed to be about one third of the height of the Hall Building for simplicity. This is not exactly representative of the actual buildings, but is identical to the configuration that was considered for the numerical modelling for which the same velocity profile shown in *Figure 9* was also used. *Figure 10* shows the building models in the wind tunnel with the simulated suburban terrain conditions upstream.

Simulated wind velocity signals were collected by using the experimental set-up shown in *Figure 8*. For each location, six velocity records were gathered and the arithmetic average of their mean values was calculated as the representative value of the velocity at the location. The vertical hot film sensor, when placed in the flow provides a combined effect of the longitudinal u and lateral v components of the velocity of this location. Therefore, the computed velocities were also evaluated as $(u^2 + v^2)^{1/2}$. It should be mentioned, however, that the experimental velocities are mainly obtained at points of high turbulence intensity, such as locations very close to the building surfaces. Therefore, the hot film anemometer technique used involves an error in the measured values. It has been estimated that this error will be about 13% for a 50% turbulence intensity and less than 2% for an intensity at 20% (TSI[19]). More accurate measurements may be carried out by using more sophisticated measurement systems such as Laser–Doppler anemometry.

Velocity ratios (amplification or reduction factors) similar to those presented in *Figure 1* for the single building case, but at 2 m height, are shown in *Figure 11* for each considered location of the multiple building configuration. Each ratio of the wind velocity in the presence of the buildings over the velocity in the absence of the buildings shows directly the effect of the buildings on the wind environmen-

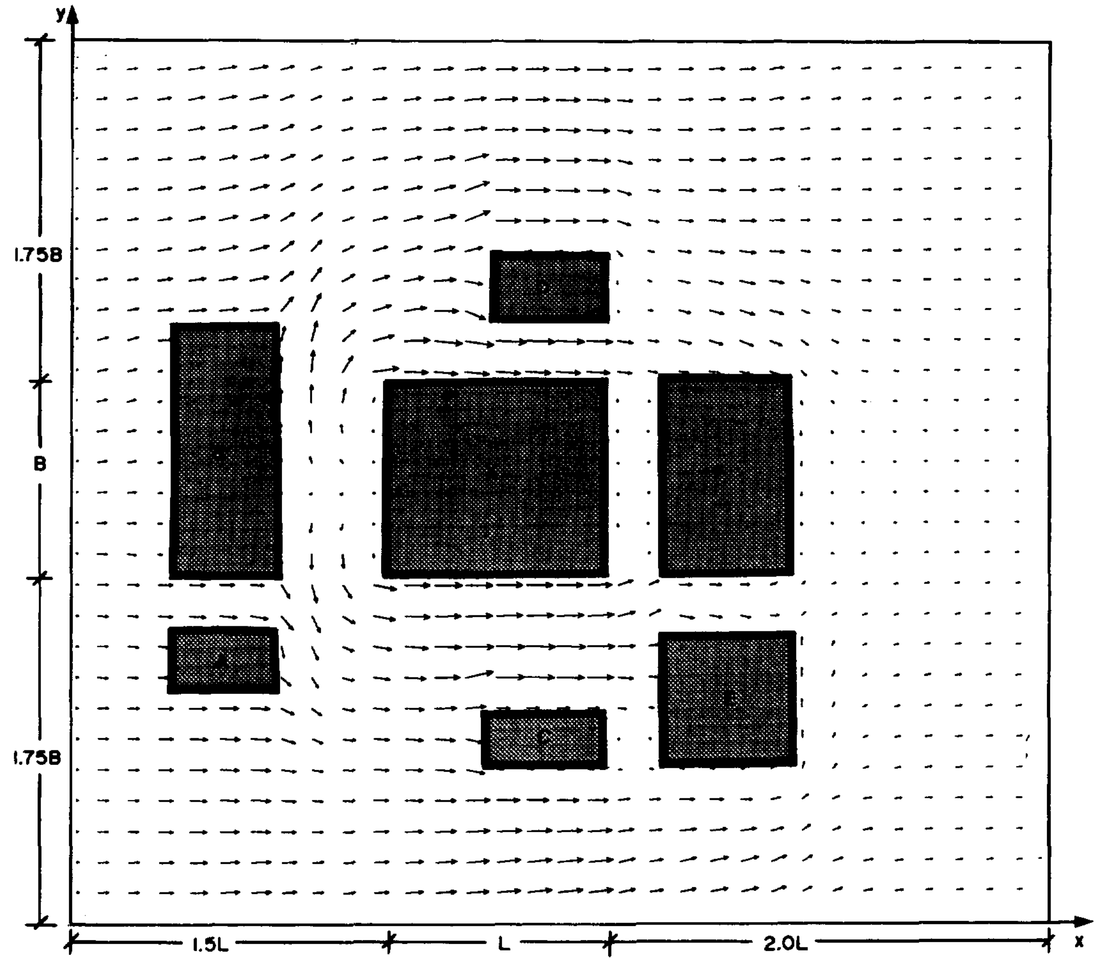


Figure 6 Vector plots of the computed velocity field around the buildings of the selected configuration

tal conditions. Both measured and computed ratios are included with the appropriate scale for comparison purposes. The results show generally fair agreement within a 30% discrepancy with the exception of points 3 and 8, on which measured and computed values differ more significantly. Note that both these points are in locations of highly complex recirculating flow regions. Consequently, neither measured nor computed values are considered accurate in these locations.

For overall performance evaluation of the computation the same data are plotted in different format, as shown in *Figure 12*. A line at 45° with the horizontal is also drawn. The points on the line represent no difference between measured and computed data. Points above the line indicate that the computed ratios are higher than the measured values for that particular location, whereas points below the line correspond to lower computed ratio values. Most of the points are close to the 45° line indicating some encouraging correlation between the measured and computed velocity ratios. However, improvements in the numerical model may be desirable in order to simulate more details in the flow and produce a better agreement with the experimental data.

Finally it should be noted that in the present study only the mean wind components have been considered for the evaluation of the wind environment. Although several researchers have proposed wind environmental criteria for evaluating pedestrian comfort based only on mean wind speeds, others believe that peak speeds of a particular duration are more important. Further work and additional computer resources will be required for the computation of fluctuating components of the wind around buildings.

Conclusions

A computer simulation technique for the evaluation of mean wind environmental conditions around buildings has been presented in this paper. Computed results generally agree with the experimental data for most locations. The advantages of the numerical simulation in terms of high speed, low cost and maximum flexibility to accommodate changes in building configurations are clear. However, several limitations presently exist with the computational approach. These include modelling of small scale phenomena associated with flow and geometrical characteristics, the influence of fluctuating wind components on the wind environmental conditions, as well as the difficulty of inputing detailed data for specific topographical and building forms. Advanced measurement techniques are also important for the experimental velocities to be representative in areas of high turbulence. Nevertheless, the present work is a pilot study indicating high potential for application to wind environmental design. Further study will be required to evaluate the results of the present methodology for different building configurations and for various wind directions.

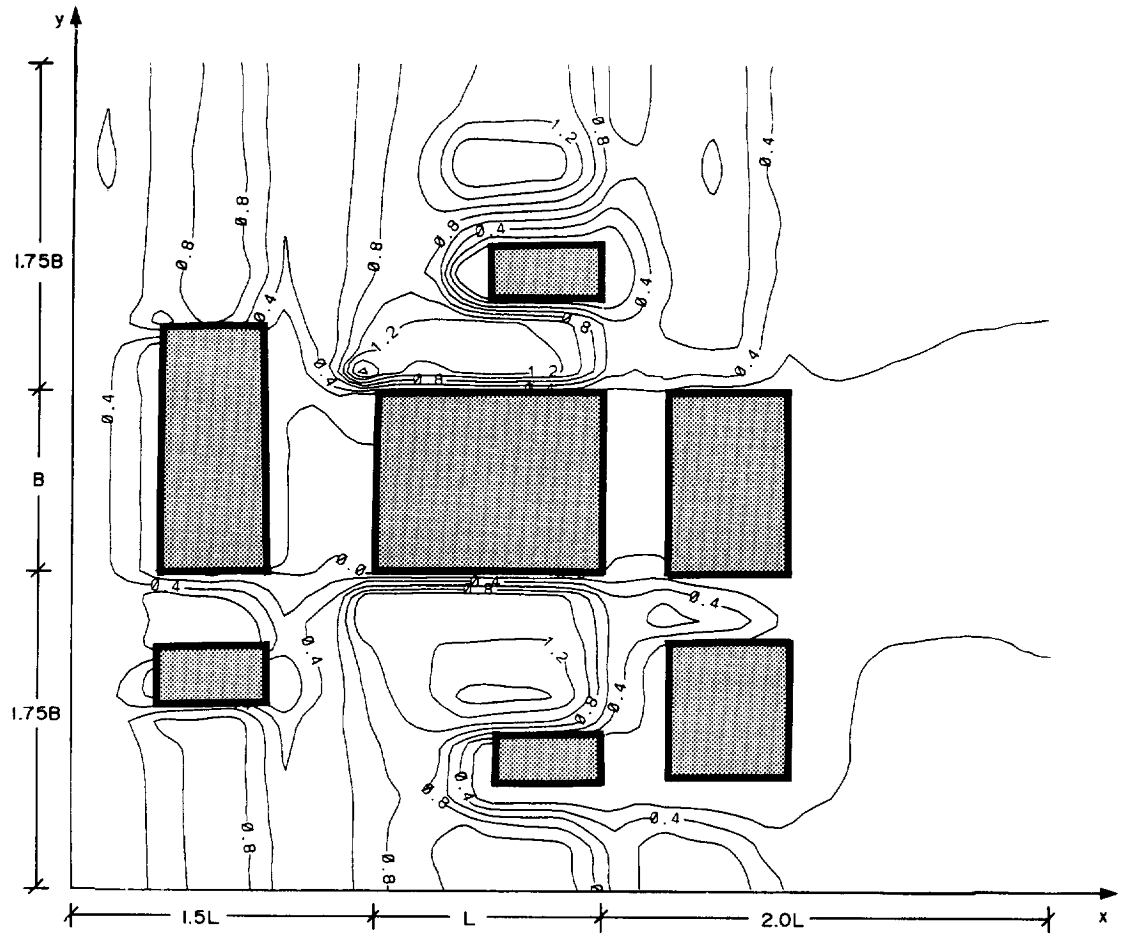


Figure 7 Computed velocity ratios around the buildings of the selected configuration

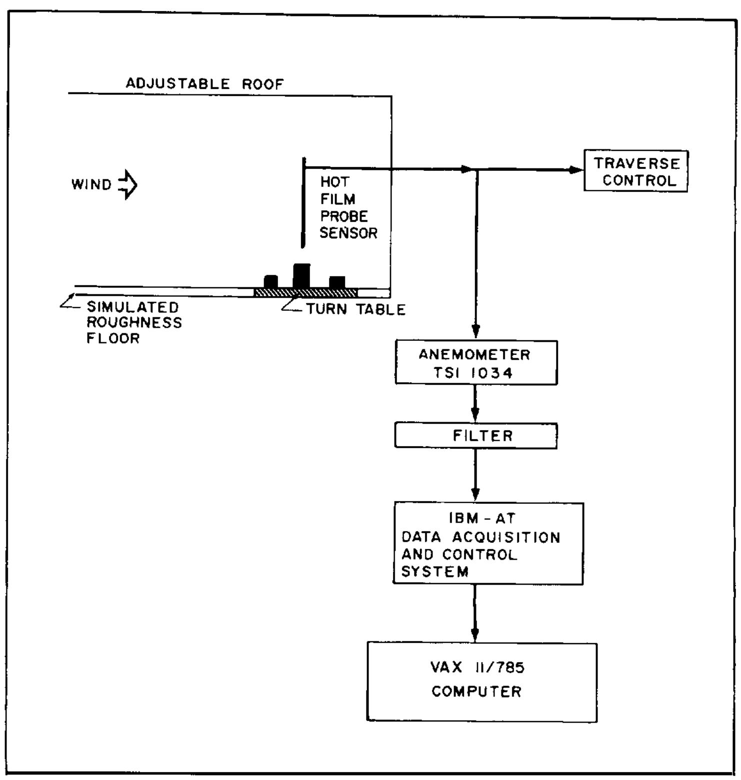


Figure 8 Experimental set-up for velocity measurements

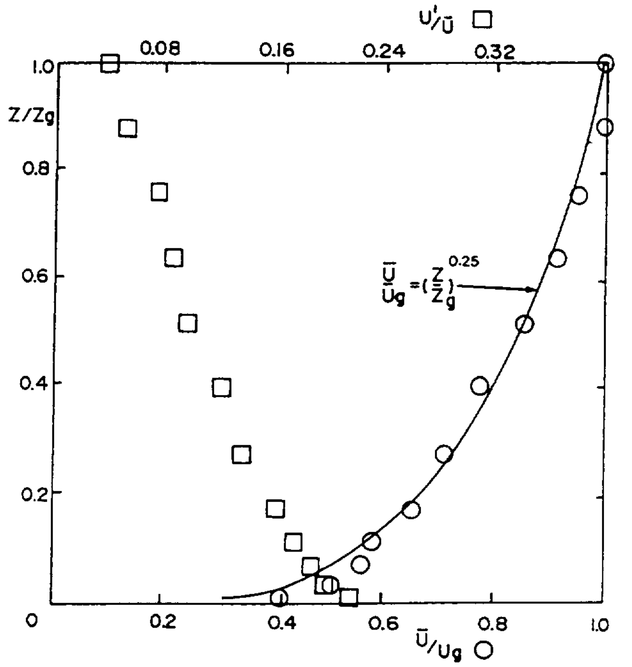


Figure 9 Velocity and turbulence intensity profiles for a simulated suburban exposure in the boundary layer wind-tunnel

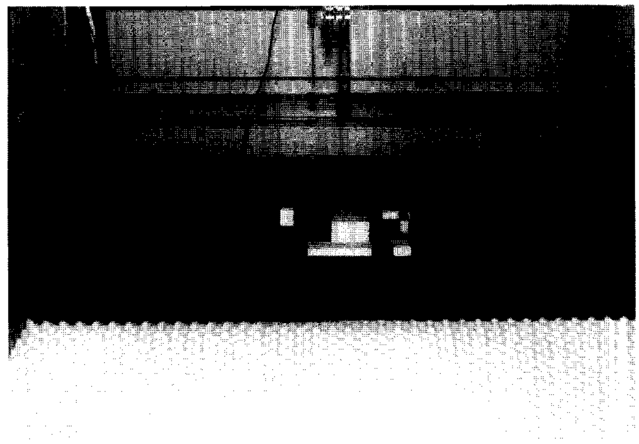


Figure 10 Cluster of building models in the boundary layer wind-tunnel

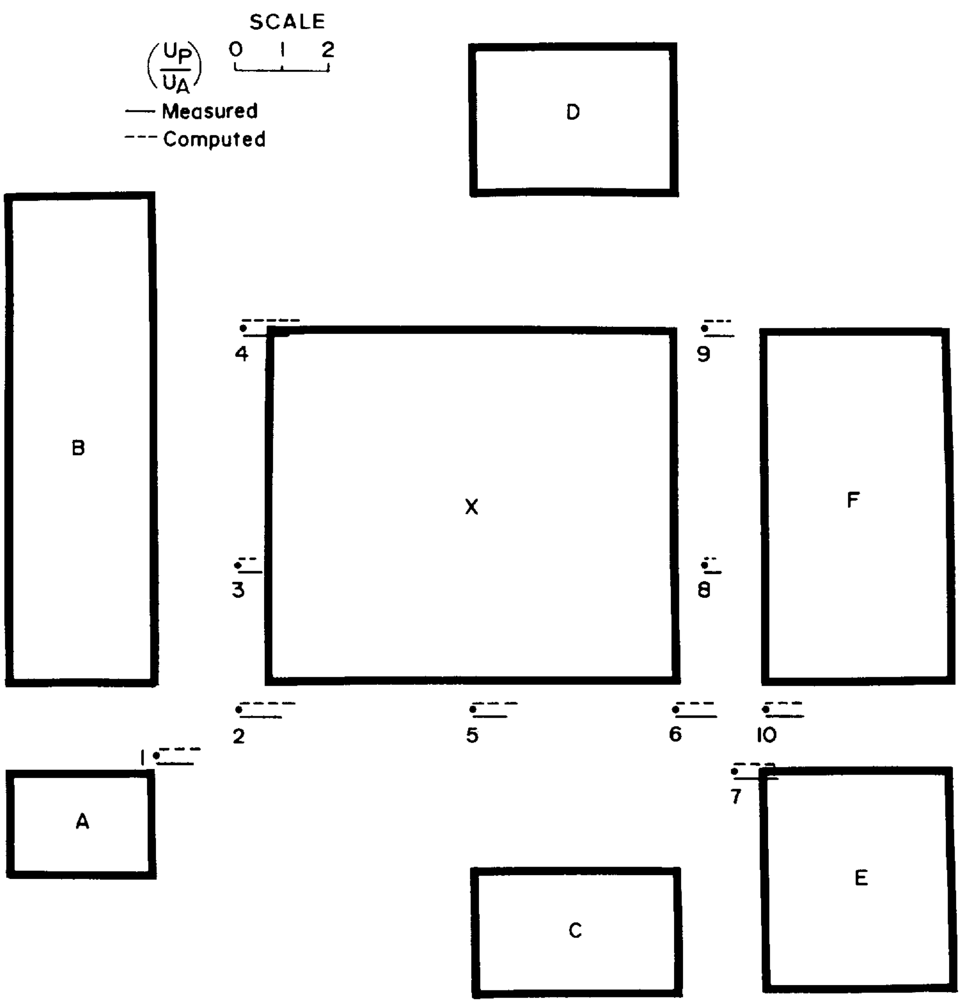


Figure 11 Comparison of the computed and measured velocity ratios at 2 m from the ground level for various locations around the buildings

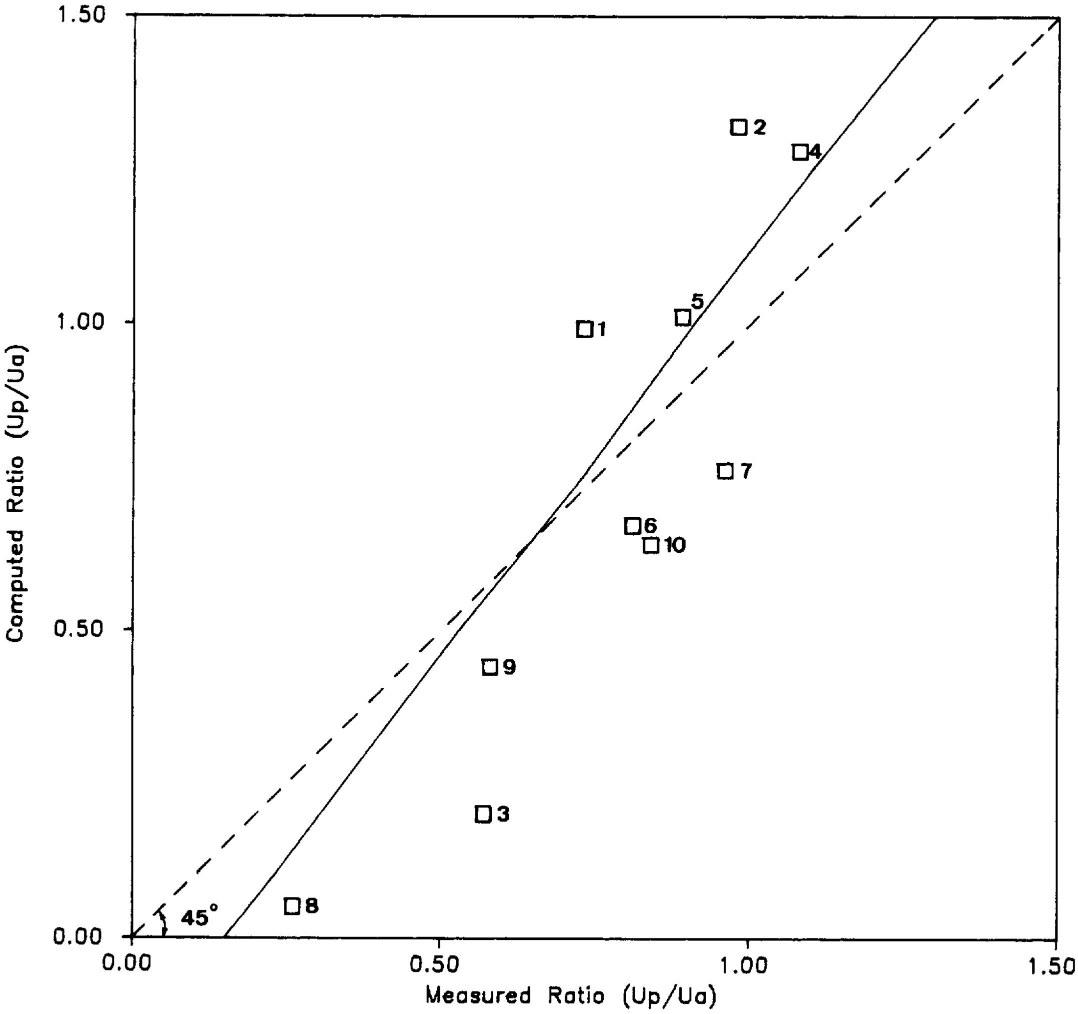


Figure 12 Relation between measured and computed velocity ratios

Acknowledgments

The research presented in this paper was supported by the Natural Sciences and Engineering Research Council of Canada (NSERC) and the Fonds pour la Formation de Chercheurs et l'Aide à la Recherche (FCAR) of the province of Quebec.

References

1 Vasillic-Melling, D. 'Three-dimensional turbulent flow past rectangular bluff-bodies', Ph.D. thesis, University of London, England, 1977
2 Summers, D. M., Hanson, T. and Wilson, C. B. 'Validation of a computer simulation of wind flow over a building model', *J. Bldg Environ.* 1986, **21**, 97–111
3 Paterson, D. A. 'Computation of wind flow over three-dimensional buildings', Ph.D. thesis, University of Queensland, St Lucia, Queensland, Australia, 1986
4 Murakami, S. and Mochida, A. 'Three-dimensional numerical simulation of air flow around a cubic model by large eddy simulation', *J. Wind Engng Ind. Aerodyn.* 1987, **25**, 291–305
5 Murakami, S. and Mochida, A. 'Three-dimensional numerical simulation of turbulent air flow around buildings by means of the k-ε turbulence model', *J. Bldg Environ.* 1989, **24**, 51–64
6 Baskaran, A. and Stathopoulos, T. 'Influence of computational parameters on the evaluation of wind effects on the building envelope', *J. Bldg Environ.* 1993, **27**, 39–49
7 Hanson, T., Summers, D. M. and Wilson, C. B., 'A three-dimensional simulation of wind flow around buildings', *Int. J. Numer. Meth. Fluid* 1986, **6**, 113–127
8 Haggkvist, K., Anderson, C. and Taesler, R. 'Numerical simulation of pressure fields around buildings', *J. Bldg Environ.* 1989, **24**, 65–72
9 Spalding, D. B. 'A general purpose computer program for multidimensional one and two phase flow', *Math. Comput. Simulation* 1981, **8**, 267–276
10 Gadilhe, A., Janvier, L. and Barnaud, G. 'Numerical and experimental modelling of the three-dimensional turbulent wind flow through an urban square', *J. Wind Engng Ind. Aerodyn.* 1993, **46/47**, 755–763
11 Takakura, S., Suyama, Y. and Aoyama, M. 'Numerical simulation of flowfield around buildings in an urban area', *J. Wind Engng Ind. Aerodyn.* 1993, **46/47**, 765–771
12 Baskaran, A., 'Computer simulation of 3D turbulent wind effects on buildings', Ph.D. thesis, Centre for Building Studies, Concordia University, Montreal, Canada, 1990
13 Rodi, W., 'Turbulence models and their application in hydraulics—a state of art review', International Association for Hydraulics Research, Delft, Netherlands, 1984
14 Caretto, L. S., Curry, R. M. and Spalding, D. B. 'Two numerical methods for three-dimensional boundary layers', *Comput. Meth. Appl. Mech. Engng* 1972, **1**, 39–57
15 Patankar, S. V. *'Numerical heat transfer and fluid flow'*, McGraw Hill, New York, 1980
16 Launder, B. E. and Spalding, D. B. 'The numerical computation of turbulent flows', *Comput. Meth. Appl. Mech. Engng* 1974, **3**, 269–289
17 Stathopoulos, T. 'Wind environmental conditions around tall buildings with chamfered corners', *J. Wind Engng Ind. Aerodyn.* 1985, **21**, 71–87
18 Stathopoulos, T. 'Design and fabrication of a wind-tunnel for building aerodynamics', *J. Wind Engng Ind. Aerodyn.* 1984, **16**, 361–376
19 TSI Inc. 'Applications of the heat flux system in low temperature gases and liquids', *Tech. Bull.* 4, Thermo-Systems, St Paul, Minnesota