AIC 601 #1000

Computer simulation of wind flow around buildings

T Hanson, F Smith*, D Summers and C B Wilson

This paper looks at the relationship between wind flow round a building and heat loss from it. The relative merits of numerical and wind tunnel models are discussed and various numerical techniques examined.

computer-aided design, building design, energy

The thermal performance of a building is strongly influenced by the microclimate which envelops it. The variables of this microclimate include the surrounding air temperature and vapour pressure, solar radiation and wind-flow distribution. Wind-flow and solar radiation are particularly sensitive to the shape, height and orientation of a building, as well as to its position within a group of neighbouring buildings. The design of a building which performs well, in terms of its energy response, requires an understanding of both how an advantageous microclimate may be achieved and how the proposed building will react to its microclimate.

In the past, graphical methods and physical modelling have often been used to predict the relationship of a building to the field of solar radiation; however these can be tedious in view of the amount of investigation required, or misleading if the range of variation of form which is explored is too restricted. In order to overcome this problem computer-based methods have been developed in recent years which provide time-integrated calculations of shading and exposure[1]. Additional information is needed if the availability of sunshine[2] or the intensity of radiation[3] is to be included.

Scale-model wind tunnel experiments have commonly been used in the past to predict wind-flow patterns and pressure distributions around buildings. In view of the experimental difficulties and expense of such methods it would be advantageous to develop a computer simulation of wind-flow around buildings and groups of buildings. Unfortunately the calculations associated with fluid flow prediction are more difficult than those for the solar radiation problem: they involve the solution of a non-linear differential equation (the Navier–Stokes equation) for the particularly awkward case of slightly viscous fluid flow around bluff bodies. In the present note two numerical solution methods for this problem are described which, at their present stage of development, show at least qualitative agreement with experimental results.

The emphasis is to develop a research tool with which to conduct parametric studies, rather than to develop a computer program for the naive user. The possible relevance of detailed wind-flow calculations to understanding the thermal performance of a particular building design is obvious. Current design practice utilizes a rather crude categorization of wind exposure which does not take adequate account of the influence of a building's shape on the flow pattern, nor of the influence of the surrounding built environment.

RELATIONSHIP BETWEEN WIND AND HEAT LOSS

Wind-mediated heat loss from buildings can occur through two mechanisms: heat can be convectively transferred from the external surface of the building fabric to the air; and heat loss can occur due to natural ventilation.

The former mechanism can be quantified by an empirical expression which relates the external thermal resistance coefficient, R_{ext}, to the wind velocity, v; this has a form $R_{ext} = (\alpha + \beta v)^{-1}$ where α (= 9.94 $Wm^{-2}K^{-1}$) and β (= 4.1 $Wm^{-3}K^{-1}s^{-1}$) are empirically derived constants. (See for example Markus and Morris[4].) In current British practice the values of v which are used in this equation express the broad wind-exposure categories, 'sheltered', 'normal', and 'severe', corresponding to velocities at roof level of 1, 3 and 9 m/sec respectively. In reality, of course, the wind velocity is a much more variable quantity than these categories might suggest. The flow, may, for example, be a function of space, time, wind-direction, building shape, and the surrounding built form.

In the case of well-insulated walls, the wind-dependent component of the external thermal resistance is usually a small fraction of its total resistance (10 per cent or so for a typical brick wall; less as insulation improves). However if a building contains a large proportion of single-glazed surface, then wind-dependent convective heat transfer becomes more significant and the heat loss may be a sensitive function of local wind flow.

Heat loss due to natural ventilation is caused by infiltration of cold air, through openings in the external surface, which displaces heated air. Such heat loss usually exceeds that associated with the convective process. The volume rate at which air is exchanged per unit time depends upon the pressure distribution on the external walls of the building, and this depends upon the wind velocities there. The sensitivity of ventilation heat loss to wind velocity is highly dependent on building shape as well as the distribution of openings in the surface. For isolated buildings of simple shape such loss may be taken to be directly proportional to a value of the velocity in some unperturbed part of the flow, ie the free-stream flow upstream from the building. The wind velocity in the lee of the building is taken to be zero usually. Obviously for more complicated shapes and for distributions of buildings, more detailed knowledge of the flow would lead to a more accurate estimate of the ventilation losses.

Department of Architecture, University of Edinburgh, 20 Chambers Street, Edinburgh EH1 1JZ, UK
*Department of Mechanical Engineering, University of Birmingham, Birmingham B15 2TT, UK

 0010–4485/82/010027–05 $03.00 © 1982 Butterworth & Co (Publishers) Ltd

ADVANTAGES OF COMPUTER SIMULATION OVER THE WIND TUNNEL

There are a number of factors to be considered when choosing between the two alternative methods of studying the wind flow pattern around built forms:

- *Accuracy:* In the few cases where comparisons have been attempted between wind tunnels and full scale wind studies, good agreement was found[5]. However a suitable boundary layer wind tunnel ought to be used and the site must be accurately modelled to include the effects of neighbouring buildings. Mean wind velocities can agree to within ± 20 per cent[5]. There do not appear to be comparisons of computer simulations to wind tunnel experiments in the literature.
- *Measurement difficulties:* Measurement of air velocity and pressure in a wind tunnel requires elaborate and expensive instrumentation (eg hot-wire anemometer probes and pressure sensing tubes which must be built into the model itself). The number of positions at which measurements can be made during a single run is limited. On the other hand a computer simulation produces values of velocity components and pressure over the whole field of interest simultaneously, and can be displayed graphically at will.
- *Expense:* The costs associated with wind tunnel experiments relate to those of model construction — which is time-consuming — and the running costs of the experiment. In the case of a computer simulation, the costs are those of program development and computer running. We are not yet in a position to make detailed comparisons, but this should be possible at a later stage of program development.
- *Parametric form/flow studies:* The use of wind tunnels in architectural practice has usually been confined to 'testing' well-developed design proposals, though sometimes the need for a satisfactory microclimate has encouraged their use at an earlier, conceptual stage of the design process. On the whole, architects are discouraged from working with a wind tunnel by lack of experience and an inability to judge whether or not it would be worthwhile, as well as by cost and the difficulty of gaining access to a suitable tunnel. Whilst design-specific wind tunnel investigations can undoubtedly provide valuable information in appropriate circumstances, this information is not usually easy to generalize and so adds little to the rather small current stock of applicable knowledge about the relations between built form and wind flow.

An alternative approach to the provision of guidance for designers in dealing with environmental problems, is for studies to be made of the variation of performance resulting from systematic changes in the parameters of typical forms. Examples of investigations along these lines are Penwarden and Wise's study of high wind speeds in the space between a slab block and a lower building upwind[6], Hassan's investigations of flow between parallel blocks of equal height for different block lengths and spacings[7], Smith and Wilson's parametric investigation of flow in a rectangular, unroofed walled enclosure[8], and the studies of Soliman[9], and Lee and Hussain[10] of wind pressures on groups of buildings or three-dimensional roughness elements of varying densities.

Whilst the direct application to design of studies like these is obviously limited to forms and situations more or less contained within the range of types and variations which have been investigated, they have the immediate advantage that, where they can be used, the designer has to work only with graphs, diagrams, or numbers rather than with a wind tunnel. They can also identify particular sets of values of parameters of a form which will lead to optimal performance and indicate the sensitivity of performance to changes in the form. Most practically, perhaps, they can indicate ranges of forms which are likely to perform satisfactorily and others which should be used only with caution if other considerations make their choice inevitable.

A further advantage of parametric form/performance studies is that they provide insight into the nature of the relationships between buildings and their environments and the results obtained from them accumulate into a coherent body of usable knowledge. One reason that there have been so few of them — and many are needed to provide a worthwhile coverage of the very large variety of forms found in the built environment — is that carrying them out in a wind tunnel is an extremely laborious process, since each run may need a large number of measurements. (The latter escalation of effort is of course reduced if a tunnel provided with multiple probes and automated data analysis is used, but the large number of models and runs which are required is unavoidable.) This is an area of application where a computer simulation of wind flow would have outstanding advantages over a wind tunnel.

CHARACTER OF ARCHITECTURAL FLOW

Wind around buildings can be represented as an incompressible fluid flow, the governing differential equation of which is the Navier–Stokes' equation (which is given here in a dimensionless form):

$$\frac{\partial u}{\partial t} + u \cdot \nabla_u u = -\nabla p + R^{-1} \nabla^2 u \qquad (1)$$

together with the continuity equation

$$\nabla \cdot u = 0 \qquad (2)$$

where $u = (u_x, u_y, u_z)$ is the vector velocity field, p is the dimensionless pressure; t is the time. The Reynolds number is defined as $R = UD/\nu$, with U and D scale velocity and distance typical of the flow, and ν is the kinematic viscosity. For wind flow problems the flow is only slightly viscous, thus the Reynolds number is typically very large ($10^6 - 10^7$). In addition, since one is interested in viscous effects, a no-slip boundary condition must be satisfied on solid surfaces, ie

$$u \cdot s = 0 \qquad (3)$$

where s is the tangent vector at the surface.

Equation (1) expresses conservation of momentum during the fluid motion. Because of its non-linearity, it cannot be solved analytically for problems involving boundaries of any degree of geometrical complexity, such as those associated with bluff bodies. The equations must, then, be solved numerically, and the appropriate method will depend upon the precise nature of the boundaries and the Reynolds number of the flow — this latter indicates the ratio of momentum to viscous forces in the flow. In architectural flow, momentum forces dominate most of the flow field due to the low viscosity of air.

However the boundary condition (3) requires that momentum vanish as a solid surface is approached and thus, close to walls, the viscous term (ie the last term in equation (1)) dominates; this gives rise to a viscous boundary layer typically a few centimeters thick. Despite the fact that such a boundary layer occupies a relatively small region of the flow, it plays an extremely important role in determining the overall flow structure, and it characterizes the interaction between the solid bodies and the fluid in which they are immersed.

Most numerical methods for solving equations (1) and (2) are grid-dependent methods, involving a discrete simulation of the differential equations at each grid point in space. The computation time in which a solution can be obtained is highly dependent on the number of grid nodes which must obviously be spaced sufficiently finely in order to resolve the small scale features of the flow. This poses the problem for architectural wind flow, that a grid would need to resolve boundary layers a few centimeters thick, yet need to extend into a region several hundred metres around a building. A grid which acceptably embraces the boundary layer would lead to prohibitive computation cost.

Architectural flow is characterized by separation of the flow at the sharp corners of buildings, leading to formation of 'free-shear' layers in the lee of the building. The position of 'separation points' critically affects the accuracy of the numerical solutions to equation (1). Computation with one of the methods described in this paper (the control volume method) suggests that the difficulty of requiring a grid to resolve boundary layers may be relaxed as long as the separation points are accurately computed. (The saving in computer-time afforded by this observation, resembles that found empirically in the wind-tunnel: the problem of scaling the flow so that R is preserved may be ignored for bluff bodies with fixed separation points. This allows the use of scale-models for microclimatic design purposes and enables pressure coefficients (C_p) to be tabulated for wind loading and ventilation calculations independently of the size of the building and of the incoming wind speed.)

Another aspect of flow problems with high Reynolds number is the occurence of turbulence: this is characterized by the random fluctuation of velocity components about their mean (ie about their time-averaged values). The way in which this is accounted for depends upon the numerical method which is adopted.

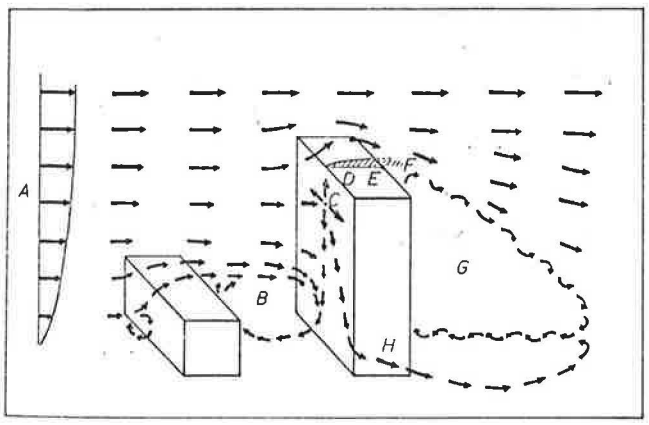

Figure 1. Characteristic features of a simple architectural wind flow (after Penwarden and Wise[6]). A: atmospheric boundary layer velocity profile; B: vortex region; C: windward stagnation point; D: turbulent boundary layer; E: boundary layer separation point; F: free-shear layer; G: recirculating turbulent wake; H: high speed corner-stream

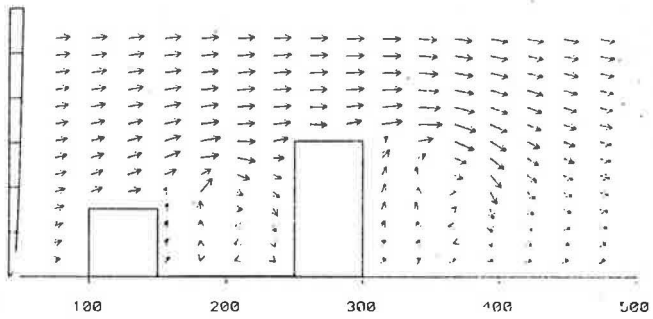

Figure 2. Solution of the two building problem by vortex method (scale in metres)

Some of the flow features relevant to architectural flow are illustrated in Figure 1, which is an illustration based on the results of wind tunnel study by Penwarden and Wise[6]; it shows how a low building upwind of a tall building affects the pattern of wind flow. The profile which describes the atmospheric boundary layer used in their study is also illustrated. In the present note a 2D version of this case, using two methods of computer simulation, will be presented to allow a qualitative comparison to be made.

METHODS OF NUMERICAL SOLUTION

Vortex method

The vortex method (developed by Chorin[11] to treat the particular problem of 2D high Reynolds number flow) has the unusual advantage that it does not require the imposition of a grid over the solution domain. The principle of the method is to simulate the physical process of the generation of vorticity into the flow from solid surfaces. The resulting flow is represented as a finite number of discrete 'vortices' superimposed on a background flow which is assumed inviscid. These 'vortices' are created due to viscous shear near walls and are then convected and diffused into the interior of the flow. The diffusive component of this motion is treated as a random displacement of the 'vortex' whose magnitude is a function of Reynolds number. This random element in the solution may be considered to simulate to some extent the effects of turbulence; the method thus incorporates an implicit turbulence model although it may prove an unsophisticated one.

Figure 2 illustrates the velocity field solution of the vortex method for the 2D representation of the two buildings shown in Figure 1. The method is time-evolutionary and produces at each time-step instantaneous field values of the developing flow, rather than single time-averaged values. In Figure 2 the arrows represent the velocity field values averaged over ten time steps; since the flow may still be developing, this field does not correspond to a steady state.

There are obvious advantages to using the vortex method in architectural flow simulations. The problems of fine-gridding the large space surrounding buildings does not occur, and the computation (for the 2D problem) reduces to numerical solution of a line-integral equation taken over

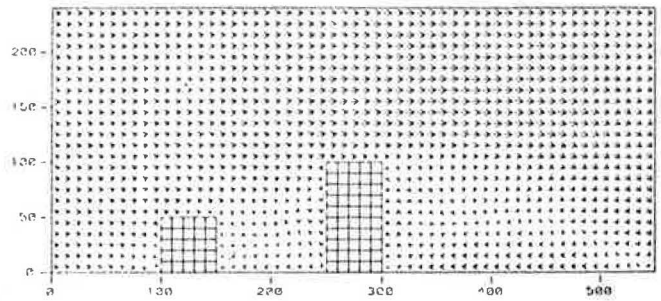

Figure 3. Solution of the two building problem (elevation) by control volume method

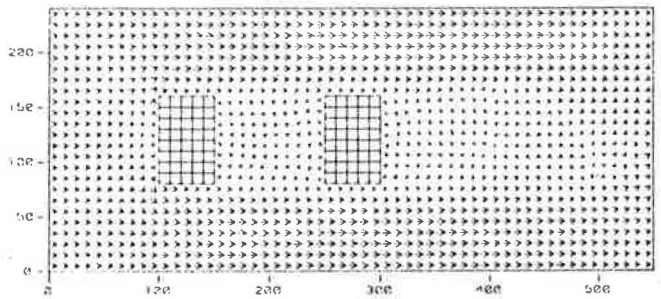

Figure 4. Solution of the two building problem (plan) by control volume method

the bounding surface of the building; thus buildings of arbitrary shape may be easily modelled, eg buildings with pitched roofs. In principle the method ought to be relatively economical of computer time, especially if the solution is required in a relatively confined region (eg along the surface of the building, or in its immediate wake, etc). The overriding disadvantage of the method is the great difficulty of extending the concept to 3D problems. It would appear this is still an area of active research by the originator of the method and his colleagues[12].

Control volume method

Recognition of the fundamental limitation of the vortex method (ie its restriction to 2D problems) led to the consideration of more conventional methods of solving the Navier–Stokes equation which would lend themselves to eventual 3D computation. The control volume method (see for example Roache[13]) derives its name from the fact that the solution domain is subdivided into regular rectangular cells (in the 2D case these are areas) through whose sides the momentum fluxes in the flow are computed to satisfy equation (1). The resulting solution, however, does not in general satisfy equation (2), the continuity condition. An adjustment is therefore made to the pressure distribution to correct this error and equation (1) is solved again for the momenta using the new pressure distribution to determine the pressure gradient term. The process is then continued until (1) and (2) are simultaneously satisfied.

There is no need to retain the time derivatives in this method so that the results obtained are effectively time-averaged. Figure 3 illustrates this for the two building model problem. It may be seen that the cells (shown by the squares forming the building shapes) are 10 m square and thus much larger than the boundary layers. Nevertheless the large scale features of the flow (cf Figure 1) are quite well reproduced.

To emphasize the need for 3D modelling, a plan version of the two building problem was also solved (see Figure 4). These results might be roughly representative of the velocity components in the flow around two very tall buildings in cross-section far above ground or below roof level. It is clear that it is not possible to reproduce all the features of the complete 3D flow simply by combining the plan and elevation results.

Extension of the control volume method to 3D is relatively straightforward. Before this, however, there are other questions to be resolved which can be tackled more economically with a 2D program. The most important of these concerns the selection of a turbulence model, for, unlike the vortex method, the control volume does not have 'built-in' turbulence and therefore requires the incorporation of an empirical model of turbulence in order to provide an accurate representation of the flow. Turbulence models generally employ the concept of turbulent viscosity in order to augment the molecular transport properties of the fluid to the levels associated with turbulent flow. Several levels of sophistication are available but each involves more complex computation (see Launder and Spalding[14]). The most primitive model, which we have used in the preceding examples, is simply to multiply the molecular viscosity by a constant factor, increasing it by two to three orders of magnitude. Further work is required to determine whether a more sophisticated method would yield a worthwhile improvement in the solutions without an excessive increase in computation cost.

CONCLUSIONS

Present criteria for estimating the effects of wind on the thermal performance of buildings are somewhat crude and reflect the fact that detailed knowledge concerning wind velocity and pressure distributions for different forms and groupings of buildings is not generally available. Provision of such information should lead to more accuracy in decision-making and to better design.

Computer simulation offers an attractive alternative to wind-tunnel testing provided that accurate prediction of large scale flow features can be economically obtained.

Encouraging results have been produced for 2D problems, but extension to 3D is essential so that realistic simulation can be attempted.

ACKNOWLEDGEMENTS

This work is supported by the UK Science and Engineering Research Council under Grant Number GR/A/8632.9. The authors thank Jim Nash and Fernando Pereira of EdCAAD Studies, University of Edinburgh, for their assistance with graphics.

REFERENCES

1 Smith, F and Wilson, C B 'The shading of the ground by buildings' *Build. Environ.* Vol 11 No 3 (September 1976) pp 187–195

2 Ne'eman, E and Light, W 'Availability of sunshine' *Build. Environ.* Vol 11 No 2 (June 1976) pp 102–130

3 Rogers, G G and Souster, C G S 'The development of an interactive computer program for the calculation

of solar irradiances and daily irradiations on horizontal surfaces on cloudless days for given conditions of sky clarity and atmospheric water content' Dept. Building Science Report BS28, University of Sheffield, UK (1976)

4 Markus, T A and Morris, E N *Buildings, climate and energy* Pitman Publishing Ltd, London (1980)

5 Jones, P M and Wilson, C B 'Wind flow in an urban area: a comparison of full scale and model flows' *Build. Sci.* Vol 3 No 1 (May 1968)

6 Penwarden, A D and Wise, A F E *Wind environment around buildings* BRE Report, HMSO, London (1975)

7 Hassan, S E S *The morphology of wind flow and built form* PhD thesis, University of Edinburgh, UK (1974)

8 Smith, F and Wilson, C B 'A parametric study of air-flow within rectangular walled enclosures' *Build. Environ.* Vol 12 No 4 (December 1977) pp 223–230

9 Soliman, B F *A study of the wind pressure forces acting on groups of buildings* PhD thesis, Department of Building Science, University of Sheffield, UK (1976)

10 Hussain, M and Lee, B E 'An investigation of wind forces on three dimensional roughness elements in a simulated atmospheric boundary layer' Department of Building Science Reports BS55–57, University of Sheffield, UK (1980)

11 Chorin, A J 'Numerical study of slightly viscous flow' *J. Fluid Mech.* Vol 57 Par 4 (1973) pp 785–796

12 Leonard, A 'Numerical simulation of interacting, three-dimensional vortex filaments' in *Proceedings of the Fourth International Conference on numerical methods in fluid mechanics* Springer Verlag, Berlin (1975)

13 Roache, P J *Computational fluid mechanics* Hermosa Publishers, Albuquerque, NM, USA (1976)

14 Launder, B E and Spalding, D B *Mathematical models of turbulence* Academic Press, New York, NY, USA (1972)