6352

Building Serv. Eng. Res. Technol. 13(2) 107–111 (1992) Printed in Great Britain

Technical Note

Summary Wind-induced pressure coefficients are required for evaluating the energy performance of existing buildings, validating new building designs and, most recently, for multizone air flow computer models as input data. However, the traditional method of obtaining pres sure coefficients, by experiment, is time consuming, expensive and not amenable to parametric changes. Work with an existing computational fluid dynamics (CFD) code, FLUENT, has been carried out for the new application of predicting building pressure coefficients. The equations describing three-dimensional, turbulent flow fields around a building are solved using a finite difference technique. Comparison between experimental data and those of the CFD program showed good agreement. In addition, it is much more efficient and economical. To demonstrate the flexibility of the CFD model, a new design concept was also examined; this shows that building a through slot in the middle of a high-rise building reduces the wind induced pressure and thus the rate of air infiltration.

Building pressure coefficients: Application of three-dimensional CFD methods to prediction

L Shao BSc PhD, **S Sharples** BSc(Hons) PhD CEng MCIBSE and **I C Ward** BSc
School of Architectural Studies, Arts Tower, University of Sheffield, Sheffield S10 2UJ, UK

Received 9 August 1991, in final form 4 October 1991

List of symbols

K	Turbulence kinetic energy (J)
ε	Turbulence dissipation (J s^{-1})
C_p	Pressure coefficient (Non-dimensional)
p	Building surface air pressure (N m^{-2})
p_{ref}	Free stream static pressure, used as a reference (N m^{-2})
V_{ref}	Free stream velocity, used as a reference (m s^{-1})

1 Introduction

Wind-induced pressure distributions around a building envelope are of fundamental importance to the magnitude of air infiltration into the building, since the wind pressure is usually the major driving force for air infiltration. In engineering practice, the pressure distribution characteristics of a building are described in terms of pressure coefficients, information on which is required not only for building air tightness and structural strength designs, but also as input data for computer software packages such as COMIS[1].

Traditionally, pressure coefficients have been obtained by means either of direct measurements under real wind conditions or measurements on model buildings in wind tunnels. This note investigates whether they may be obtained using the CFD (computational fluid dynamics) method which has advanced greatly in the last two decades with developments in computer technology.

The CFD method has several advantages over the traditional methods. The real wind measurements are known to be inefficient or unreliable because of the difficulty in getting stable wind conditions. However, this is not a problem with the CFD method, which has the additional benefit of being easier to set up, less costly and taking less time to run. Compared with the wind tunnel measurement method, the CFD method has the advantage of simulating the boundary layer wind profile more accurately and more easily, having no building size limitations, as well as being cheaper to set up. It is also easier to investigate parametrically the changes induced in the pressure field by small changes in a building's design. Another attractive general feature of the CFD method is that it provides, in addition to pressure coefficients, valuable by-products such as the detailed air flow pattern around the building. Thus, as well as a knowledge of pressure coefficients there are clues to why certain values of pressure coefficient are present and, if required, ways of reducing them.

Work has been carried out in the Building Science Unit of Sheffield University on the application of the CFD method to the calculation of wind induced pressure coefficients on buildings. Firstly, the calculation procedure is described, then the results are presented in section 3, which is divided into two parts. The first part presents the work on the validation of the CFD code and the second presents a hypothetical case study of its application which not only demonstrates the capability of the code but also an interesting conclusion from the case study.

2 Calculation procedures

In this work, the flow simulation software FLUENT was used, which solves the full Navier–Stokes equations in their three-dimensional form. Actually, since the flows under consideration in this study are turbulent, FLUENT solves the averaged Navier–Stokes equations, with additional Reynolds stress terms.

2.1 Turbulence model

The unknown Reynolds stress terms were modelled using the standard $K-\varepsilon$ model, which links these terms, via two equations, to other flow parameters. This model is considered adequate for calculating the relatively simple flow in the study, and the model has been widely used in CFD applications in the area of air infiltration and ventilation, as in Reference 2.

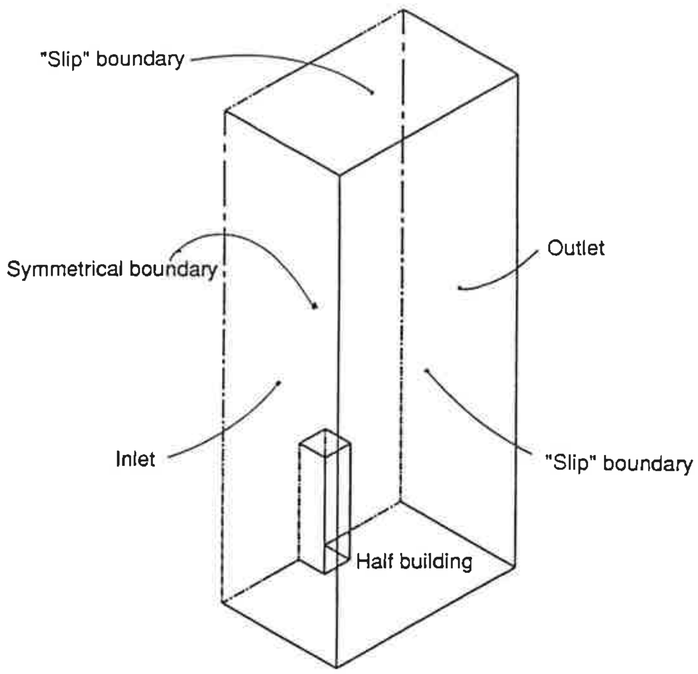

Figure 1 The computation domain

2.2 Boundary conditions

The computation domain is shown in Figure 1. Since the building only affects the air flow in its close surroundings, only the flow in the area close to the building was calculated. However, this causes the boundary of the calculation domain to be too close to the building. Since these boundaries were treated as walls in the code, unrealistic extra shear stress was added into the flow surrounding the building, causing modelling inaccuracy. This problem is solved by giving 'slip wall' boundary conditions to the side and top boundaries of the calculation domain. The buildings in this study are all symmetrical about their central symmetrical plane. This fact was utilised to reduce the amount of computer storage and calculation required by including only one half of the building in the computation domain. As a consequence, the central symmetrical plane becomes one of the boundaries and the symmetrical boundary condition has to be imposed, which assumes that the velocity component perpendicular to the plain is zero and all scalar gradients are zero. The velocity profile and turbulence intensity at the inlet of the computation domain were set to be those of the appropriate boundary layer wind.

2.3 Numerical Scheme

The Navier–Stokes equations were discretised into finite-difference equations, using the quadratic upwind discretization scheme (or QUICK). This scheme has a higher interpolation (between nodes) accuracy than the usual power law scheme for three-dimensional flows, such as that studied in this work. A multi-dimensional Cartesian grid system was employed and the choice of the density of the grid was based on two considerations. On the one hand, the grid has to be dense enough so that gradients can be accurately presented in the finite-difference equations and the calculated flow is detailed and accurate. On the other hand, the grid has to be coarse enough so that the number of cells is within the range that the storage of a given computer can cope with and the solution converges at reasonable speed. To cope with this dilemma, in this work, denser grids were used near the building where the gradients were likely to be high and coarser grids away from the building to reduce the total number of cells. In addition, to promote numerical stability, the ratios between the lengths of the sides of a cell were kept around unity for the cells close to the building, but the cells were allowed to take an increasingly longitudinal shape the further away they were from the building. The ratio of expansion, in the direction of away from the building, of the distance between parallel grid lines is about 1.2.

The finite-difference equations resulting from the discretisation process described above were solved using the SIMPLE[3] algorithm. The solution convergence speed was increased by means of the over-relaxation technique after a certain number of iterations, during which the solution had shown steady convergence.

3 Results and discussion

3.1 Code validation

Code validation can be carried out by comparing the code's prediction of pressure coefficients with data from corresponding experimental measurements either in the real wind or in a wind tunnel. However, the latter is preferred to the former, which suffers from much higher levels of uncertainty of data accuracy, as explained in the introduction.

Although a significant number of wind tunnel tests of building pressure coefficients have been reported[4,5] only a small number of them actually provide all the information that is needed for input to the CFD code, so that the CFD prediction can be computed and the comparison made. The best set of tests so far available, as far as this comparison is concerned, is that by Hussain and Lee[6]. Detailed information on wind velocity and turbulence intensity[7] profiles, measurement method, as well as positions of reference pressure and reference velocity were given. Moreover, a pressure coefficient profile along the vertical centre line of the windward facade, consisting of more than 10 data points, was measured, making detailed and reliable comparison with the CFD prediction possible.

The wind tunnel Hussain and Lee used was fan driven, with a honeycomb section to straighten the flow. The working length, measuring 7.2 m, and the working section, measuring 1.2 m × 1.2 m, can be divided into two parts. In the front, a castellated fence, a row of spires and a regular array of floor roughness were installed so that the air flow, when passing through, would develop a velocity profile of the same form as that of a typical urban wind—a power law in shape with an exponent index of 0.28. In the lower part of the profile, the measured velocity is larger than that corresponding to the 0.28 power law and the true value was incorporated into the CFD calculation. The second part of the working section contained a turntable with the model building at the centre. All the models used by Hussain and Lee were rectangular in shape with one side perpendicular to the wind direction. However, they differed from each other in height, frontal ratio and side ratio. The model chosen for the CFD calculation was among the taller ones and thus the pressure coefficients measured on the model are likely to be more accurate. The model measured 36 mm × 36 mm × 61.2 mm and had 12 pressure tappings (made of hypodermic tubing) in a vertical line in the centre of its windward facade. The measured pressure profile was reduced into a pressure coefficient profile using the following formula

$$C_p = \frac{p - p_{\text{ref}}}{\frac{1}{2}\rho V_{\text{ref}^2}} \quad (1)$$

where C_p, ρ and p are the external pressure coefficient, air density and pressure, respectively. p_{ref} is the free stream static pressure measured at a point above the model and V_{ref} is the free stream velocity or the maximum velocity at the top of the boundary layer.

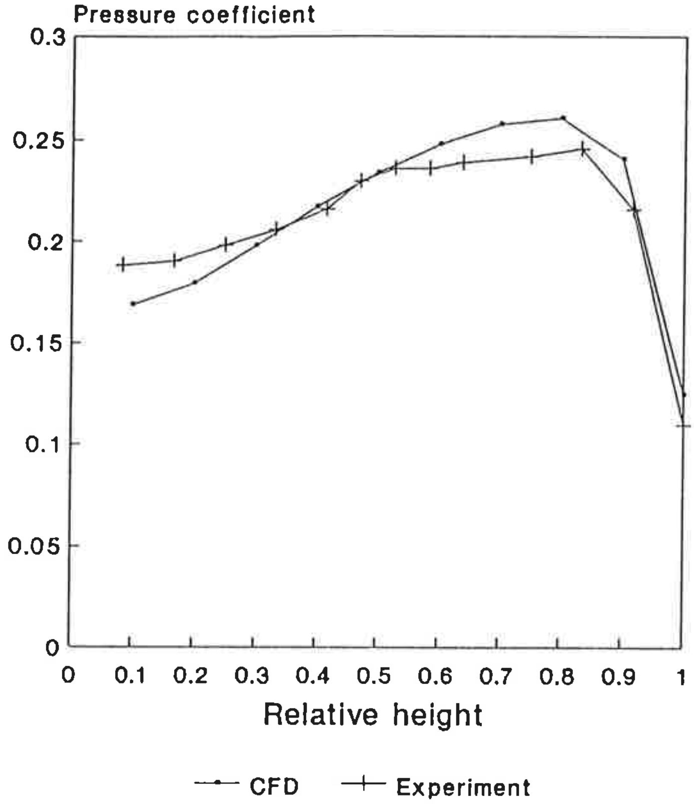

Figure 2 Windward pressure coefficient profiles: comparison between CFD and experiment

Figure 2 shows the CFD predicted windward pressure coefficient profile along the normalised building height together with that from Hussain and Lee's measurement. The point on the latter curve corresponding to full building height was not measured at the top of the windward facade but at the point 3 mm back along the roof central line. This is acceptable, since pressures at these two points are usually very close to each other. It can be seen from Figure 2 that the CFD prediction is in good agreement with the experimental result. Both curves rise with the relative height, to the peak at around the relative height of 0.8. This increase in pressure coefficient is largely due to the higher values of air flow velocity at higher vertical planes. Both curves then drop sharply. This reduction in pressure coefficient is because the building causes a contraction in the streamlines and thus causes the air to be accelerated over the top of the building.

Figure 3 shows the comparison between the leeward pressure distributions obtained from the CFD calculation and the experiments of Hussain and Lee. The agreement between the two curves is not as good as that between the windward pressure curves. This is probably because the flow phenomena in the leeward side of the building are more complex (with three-dimensional vortical structures) than in the windward side. Consequently, the number of nodes in the leeward side (the same as in the windward side) is not large enough to simulate accurately the gradients of flow variables there and thus the simulation accuracy suffered. Another contributing factor is probably that the $K - \varepsilon$ model is not suitable for the complex flow regime in the leeward side of the building. To improve the accuracy of the CFD prediction of leeward pressures, more nodes (denser grids) could be used. However, they are currently not available because of computer storage limitations. In addition, other more accurate turbulence models should be tested against the $K - \varepsilon$ model, as they become incorporated into the current code.

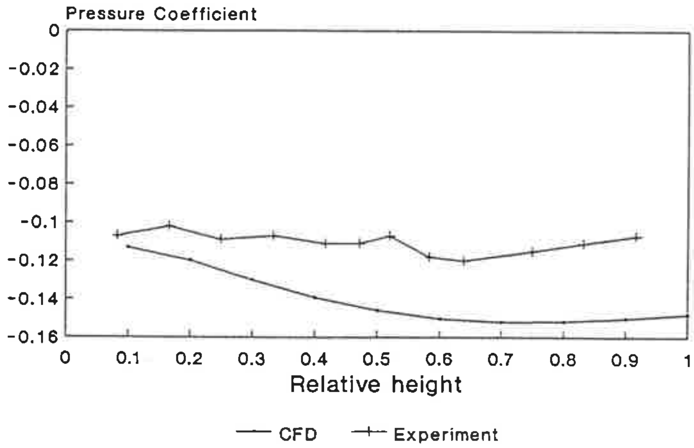

Figure 3 Leeward pressure coefficient profiles: comparison between CFD and experiment

To sum up: for the geometrically simple case of an isolated rectangular building model in a shear flow boundary layer the CFD results for windward pressure coefficients are in good agreement with the experimental data. Leeward pressure coefficient predictions may be improved when more accurate models are available and computers with larger storage are in use in the future.

3.2 Application

As mentioned in the introduction, the CFD code can also be used for investigating a proposed building design, in identifying the causes of air infiltration problems and consequently proposing a solution. This is demonstrated in the following hypothetical example.

It has been known[6] that among the parameters describing the configuration of a building, the building height is the most prominent in influencing the building surface pressures under wind. These pressures increase with height at a significant rate. Since high building surface pressures may lead to excessive air infiltration into the building, which in turn causes excessive energy consumption, it is desirable, especially for high-rise buildings, that the pressure be reduced. A reasonable proposal is to build a slot across the middle of a tall building, so that the tall building is reduced (Figure 4) to two much shorter ones (referred to below as upper and lower parts) in the hope that the building surface pressures are reduced. Moreover, this slot introduces air directly to the leeward side of the building, reducing the degree of vacuum and thus the negative pressure there. This design concept is tested using the CFD code, to demonstrate the flexibility of such a code.

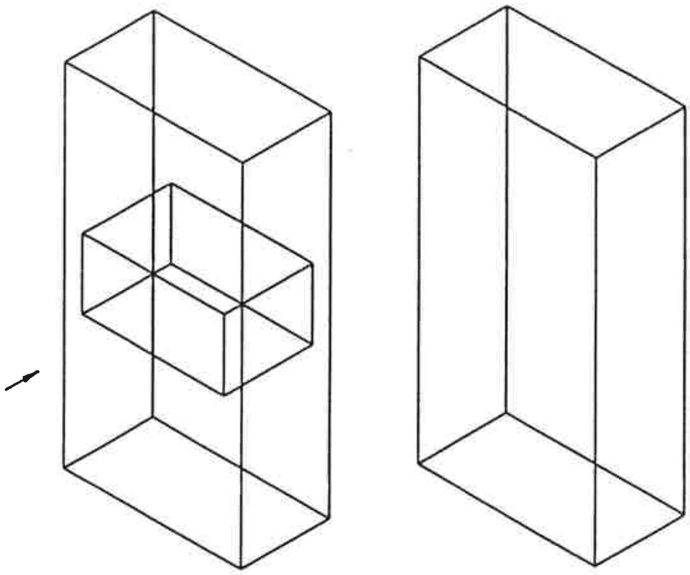

Figure 4 The two buildings used in the application case study

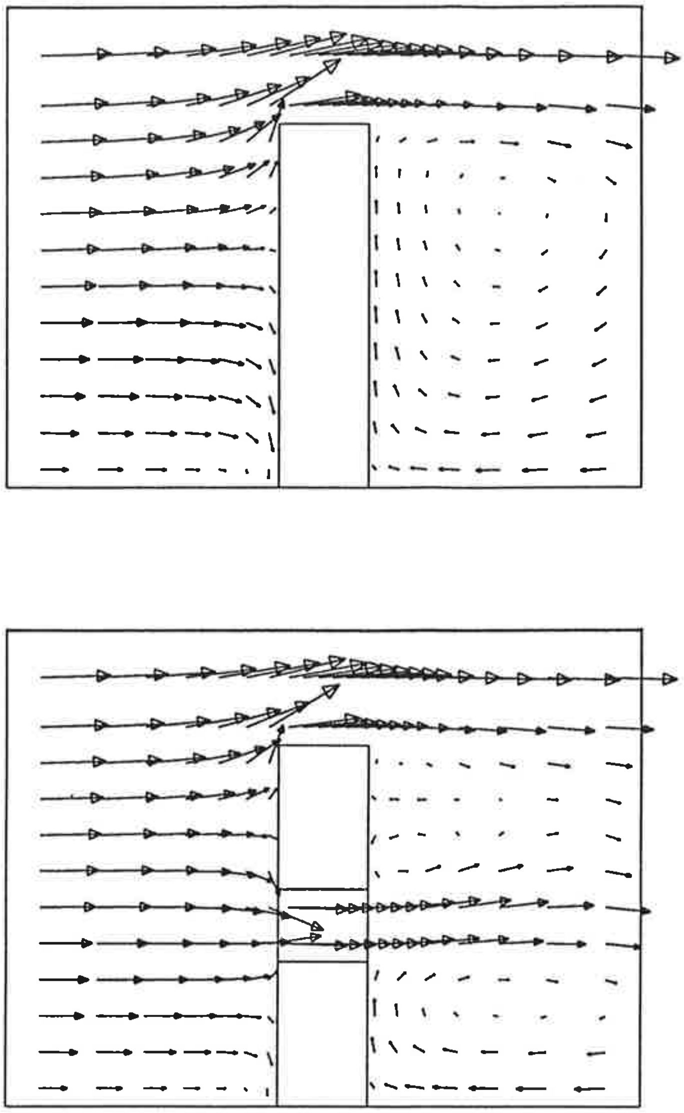

Figure 6 Velocity fields immediately surrounding the two buildings

Two buildings (Figure 4), one with and the other without the slot (otherwise identical), are used in the calculation. The simulated wind has a power-law velocity profile and is in the direction of the slot. The power law profile index is 0.28 which is the typical value for an urban wind[(6)].

Figure 5 shows the pressure profiles along the windward and leeward facade vertical centre lines of both buildings. Concentrating on the pressure profiles corresponding to the lower parts of both buildings, one would notice the improvement the slot introduces—both the windward pressure and leeward negative pressure have been reduced, the reduction of the latter being the more significant. However, the picture for the upper parts of the buildings is, unexpectedly, not so clear cut. On the windward facade the

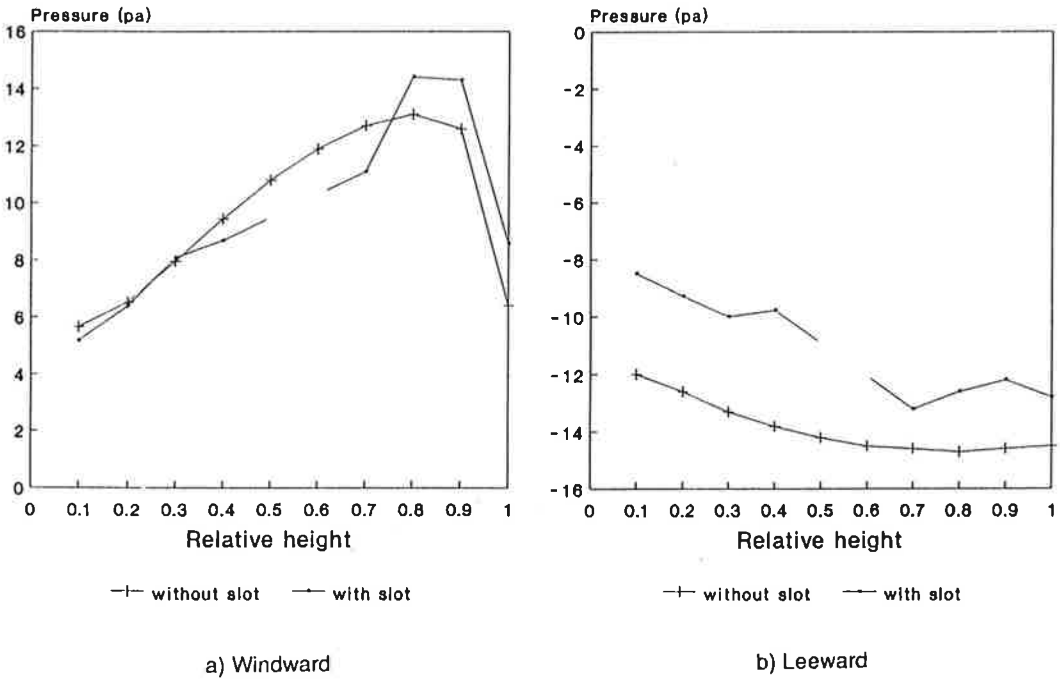

Figure 5 Comparison between the two buildings for (a) Windward and (b) Leeward surfaces

introduction of the slot actually increased the pressure. To find out the cause, information on the velocity field around the building, provided by the CFD code as a by-product, was looked at. It turned out that the slot diverted part of the flow which originally climbs up and then flows over the building (Figure 6). As a result, the speed of the flow climbing up the building reduces (see Table 1) and consequently the pressure increases as dictated by Bernoulli's law.

Table 1 The comparison of air velocities on the surfaces of the upper parts of the two buildings

Relative height	0.8	0.9	1.0	1.1
Velocity ($m\,s^{-1}$) without slot	0.649	1.4	3.46	6.55
Velocity ($m\,s^{-1}$) with slot	0.622	1.05	3.16	6.32

However, the leeward negative pressure of the upper part reduces due to the introduction of the slot and this reduction more than compensates for the increase of windward pressure (Figure 5). So the slot is beneficial to the upper part of the building too. Based on the above discussion, it can be concluded that the slot design is an effective way of reducing wind-induced pressure and thus the rate of air infiltration for high-rise buildings.

4 Conclusions and future work

The suitability of applying a CFD method to building pressure coefficient calculations has been assessed for a simple rectangular building model for a normal incidence wind direction. The three-dimensional, turbulent flow field equations were solved using the finite-difference technique. Comparison between experimental data and those of the CFD method, which involves 20 points on the building surfaces, showed good agreement. It is obviously necessary that other types of building of various geometries and houses subjected to wind of various directions be used to test the CFD method as suitable experimental data become available.

The CFD technique can be used in obtaining detailed pressure coefficient information for an existing building, validating a new design, probing an air infiltration problem and providing input data for multi-zone air flow models (e.g. COMIS). Compared with traditional experimental methods the CFD technique is much more efficient in terms of time, and more economical in terms of manpower and physical model construction cost.

An example of further applications of the CFD method is the examination of a new design concept using the CFD method which has been carried out. This shows that building a through-slot in the middle of a high rise building reduces the wind-induced pressure (and thus the rate of air infiltration).

The pressure that drives air infiltration is generated not only by wind, but also by temperature differences (between the inside and outside of a building or between two zones of one building). The CFD simulation of the latter phenomenon is currently being studied.

Acknowledgements

The authors would like to express their sincere thanks to Dr A T Howarth and H-G Kula for the many useful discussions and the valuable information they provided during this work. This study was part of an air infiltration research project funded by the Science and Engineering Research Council (Grant No. GR/F28397).

References

1 Feustel H E *et al.* *Fundamentals of the multizone air flow model—COMIS* AIVC Technical Note **TN-29**(Coventry: Air Infiltration and Ventilation Centre) (1990)
2 Awbi H B Application of computational fluid dynamics in room ventilation *Building and Environment* **24**(1) 73–84 (1989)
3 Patankar S V *Numerical heat transfer and fluid flow* (New York: McGraw–Hill) (1980)
4 Allen C Wind pressure data requirements for air infiltration calculations AIVC Technical Note **TN-13**(Coventry: Air Infiltration and Ventilation Centre) (1984)
5 Kula H-G and Feustel H E *Review of wind pressure distribution as input data for infiltration models* LBL Report **23886** (California: Lawrence Berkeley Laboratory) (1988)
6 Hussain M and Lee B E *Flow over isolated roughness elements and the influence of upstream fetch* Internal Report **BS-55** (Sheffield: Building Science Unit, Sheffield University) (1980)
7 Lee B E *The simulation of atmospheric boundary layers in the Sheffield University 1.2* m × *1.2* m *boundary layer wind tunnel* Internal Report **BS-38** (Sheffield: Building Science Unit, Sheffield University) (1977)