2496

#2496 AIC 1979

Journal of Wind Engineering and Industrial Aerodynamics, 25 (1987) 219-228 219
Elsevier Science Publishers B.V., Amsterdam — Printed in The Netherlands

PREDICTION OF THE WIND-GENERATED PRESSURE DISTRIBUTION AROUND BUILDINGS

E.H. MATHEWS

Department of Mechanical Engineering, University of Pretoria, Pretoria 0002 (South Africa)

(Received October 4, 1985; accepted in revised form September 3, 1986)

Summary

The use of computational methods to predict wind-generated pressure distributions around buildings is investigated. These pressure distributions were needed for the prediction of natural ventilation in the buildings. For the example considered, the accuracy of the predicted pressure distribution was found to be acceptable.

Notation

B	constant
C_1, C_2, C_μ, C_D	coefficients in turbulent transport equations
div	divergence
h_{ref}	height at which V_{ref} is measured (usually 10 m or equivalent height for scale model)
H	width or height of building
I	turbulence intensity
k	turbulent kinetic energy (time-averaged)
k_s	surface friction coefficient
L	length scale of turbulence
P	pressure
U_O	average inflow velocity
V	velocity (time averaged)
V_{ref}	velocity at 10 m or at equivalent height for scale model
x	x direction
y	y direction
z	turbulence property

Greek symbols

α	mean wind speed exponent
ϵ	dissipation rate of turbulent kinetic energy (time-averaged)

μ	effective viscosity (laminar plus turbulent viscosity)
ρ	density
$\sigma_k, \sigma_\epsilon$	coefficients in approximated turbulent transport equations

Subscripts

1	x direction
2	y direction
ref	reference
t	turbulent
x	main flow direction
y	perpendicular to main flow direction
z	turbulence property (k or ϵ)

Other

∇	del operator
$\frac{D()}{Dt}$	substantial derivative; that is, differentiation following the motion of a fluid particle
$\rightarrow$	vector quantity

1. Introduction

The prediction of airflows around buildings and the resulting pressure distributions around them are subjects of great importance. Velocity fields around buildings are for example needed when air pollution from, or the wind environment around, buildings is studied. The pressure distribution around a building is often needed for structural design. Furthermore, the wind-generated pressure distribution is one of the driving forces of natural ventilation. The present study was undertaken to demonstrate the possibility of calculating the wind-generated pressure distribution around a building for the purpose of calculating natural ventilation inside the building.

Many researchers and designers in the field of natural ventilation use boundary layer wind tunnels to acquire pressure data needed for the calculation of natural ventilation flow rates [1-4]. Although such wind tunnels are useful tools for obtaining wind-induced pressure distributions for design purposes, they have the following disadvantages:

(1) They are expensive in capital and running costs.

(2) Tests are time-consuming.

(3) It may be difficult to simulate the atmospheric boundary layer and turbulence for complex terrains.

(4) Many designers and researchers interested in natural ventilation do not have access to boundary layer wind tunnels.

Because of the disadvantages of wind tunnel studies and due to the recent prominence and rapid development of numerical techniques, the use of numerical methods to obtain pressure data was investigated [5]. Great effort is presently devoted to the development of computer hardware and software for the numerical solution of fluid flow problems [6, 7]. These methods have already been successfully used to calculate flow in and around buildings [8–10]. Possible advantages of numerical flow analyses are the following:

(1) Once a computer algorithm has been developed, a computational study may be performed with much more speed than experimental investigations [11].

(2) It may be easier to simulate the full-scale boundary conditions with computational methods than in the wind tunnel.

(3) The problems of scaling effect and building expensive models for wind tunnel testing are eliminated.

(4) Flow calculations may be less expensive than experiments. Pletcher and Patankar [11] point out that the cost of performing a given calculation decreases tenfold every eight years, while the cost of performing experiments is steadily increasing.

(5) Many designers and researchers interested in predicting natural ventilation have access to a computer on which flow can be simulated numerically.

Although numerical studies of wind flow and the resulting pressure distributions around buildings are important, it seems that little work has been done in this field. Researchers such as Frost and co-workers [12, 13] and Yeung and Kot [14] used the vorticity-stream function formulation of the Navier–Stokes equations to establish flowfields around two-dimensional rectangular blocks. However, they do not give the resulting pressure distributions around the blocks. Hanson et al. [10] used both the vorticity-stream function formulation and the primitive variable formulation of the Navier–Stokes equations to obtain the velocity fields, but not the pressure distributions around two-dimensional blocks. An excellent discussion on the use of computational methods in fluid flow for thermo-fluid problems related to buildings is given by Hammond [9]. However, calculated pressure distributions around buildings are not given.

Although the vorticity-stream function approach seems popular for obtaining flowfields around buildings, it has the disadvantage that pressures are not determined during the solution of the flow equations. An additional procedure is needed to solve for the pressures. Is is also difficult to extend this approach to three dimensions. The aim of this paper is to demonstrate that the primitive variable formulation of the Navier–Stokes equations can successfully be used to predict wind-generated pressure distributions around buildings for the purpose of calculating natural ventilation flow rates. A computer program which embodies a particular numerical technique has been developed [5] and is discussed. Although the potential use of this procedure for the calculation of natural ventilation is not fully exploited by the program, work in this field is initiated.

A two-dimensional flow approach is followed. Buildings in general are usually long and narrow for reasons of natural ventilation, thermal performance and natural lighting. In many cases therefore, buildings have one predominant length dimension. In a city most of the buildings tend to be tall, therefore also having one predominant length dimension. For long and for tall buildings, a two-dimensional flow approach could, as a first approximation, produce useful information. The computer code can however be extended to three dimensions with minimal modification to the basic code.

2. Basic outline of the numerical procedure

2.1. Partial differential equations

The partial differential equations that govern the movement of a viscous fluid are the Navier–Stokes equations and the continuity equation. In the case of incompressible flow the primitive variable formulation of the Navier–Stokes equations in vector notation are given by

$$\rho\frac{\mathrm{D}\vec{V}}{\mathrm{D}t} = -\operatorname{grad} P + \mu\nabla^2\vec{V} \tag{1}$$

while the continuity equation in vector notation is given by

$$\operatorname{div}\vec{V} = 0 \tag{2}$$

For the simulation of turbulence in the flow, the k–ϵ turbulent viscosity model is employed. The time-mean partial differential equation for the transport of the turbulence property z, where z can denote either k or ϵ, is given by [15]

$$\rho\frac{\mathrm{D}z}{\mathrm{D}t} = \operatorname{div}\left(\frac{\mu_t}{\sigma_z}\operatorname{grad} z\right) + \left[C_1\frac{\mu_t}{k}(\operatorname{div}\vec{V}^2 - BC_2\frac{\rho\epsilon}{k}\right] \tag{3}$$

where $\mu_t = C_\mu \rho k^2/\epsilon$ is the turbulent viscosity. If the turbulence property z in eqn. (3) is the kinetic energy k, the value of C_1 will be equal to unity while the value of B will equal $1/C_2$. By substituting the dissipation term ϵ into eqn. (3) the transport equation for ϵ is found. The value of B will then be unity. The values of the constants σ_ϵ, σ_k, C_μ, C_1 and C_2 depend on the particular flow being investigated and may therefore vary for different flow applications. The following set of turbulence constants for the atmospheric boundary layer, proposed by Yeung and Kot [14], were used for the purpose of this study: $\sigma_\epsilon = 1.0$, $\sigma_k = 1.0$, $C_\mu = 0.03$, $C_1 = 1.54$, $C_2 = 2.0$.

2.2. Finite difference equations and solution procedure

The finite difference equations for the numerical procedure are derived by integrating the partial differential equations over control volumes surrounding a grid point [5]. These equations are then solved by a line-by-line solution method. Unknown variables along each grid line are calculated by the application of a tri-diagonal matrix algorithm. To maintain stability, the variables are under-relaxed. The converged solution provides mean values for all the variables.

2.3. Boundary conditions

(a) Inflow boundaries

For boundary layer flow conditions, the power law is used to simulate the inflow V_1 velocities of the atmospheric boundary layer. The power law is given by [16]

$$\frac{V_1(y_1)}{V_1(y_2)}=\left[\frac{y_1}{y_2}\right]^{\alpha} \tag{4}$$

where $V_1(y_1)$ is the mean wind speed at a height y_1. The exponent α is the mean wind speed exponent and is dependent on upstream terrain roughness. Values of α for different terrain categories are widely published [16].

For modelling tall buildings, where only a two-dimensional section through the building is investigated, a uniform V_1 velocity profile is used at inflow.

In all cases the inflow values of V_2 velocities are set to zero, while the length scale of longitudinal turbulence L for natural wind at the inflow boundary is approximated from the following empirical equation [16]:

$$L(y)=151(y/10)^{\alpha}(h_{\mathrm{ref}}/10) \tag{5}$$

where $L(y)$ denotes the length scale of the velocity component V_1 in the x (or flow) direction at height y. The height at which the reference velocity is measured is given by h_{ref}. The length scale distribution at inflow can be prescribed via inflow values for $\epsilon(y)$. The relationship between $\epsilon(y)$ and $L(y)$ is defined by the following equation [15]:

$$\epsilon(y)=\left[C_{\mathrm{D}}\rho k(y)^{3/2}\right]/L(y) \tag{6}$$

where the value of C_{D} is assumed to be unity [17].

The turbulence intensity $I(y)$ of natural wind at height y at inflow can also be approximated by an empirical equation, which is given by [16]

$$I(y)=(6.7k_{\mathrm{s}})^{\frac{1}{2}}\,V_{\mathrm{ref}}/V_1(y) \tag{7}$$

where V_{ref} and $V_1(y)$ are the mean speeds at heights h_{ref} and y respectively and where k_{s} is a surface roughness parameter which is a measure of the surface friction coefficient of the terrain. As the value of $k(y)$ is a measure of turbulence intensity at height y, the turbulence intensity can be simulated at inflow by specifying appropriate values for $k(y)$. The relationship between $k(y)$ and $I(y)$ is given by [5]

$$k(y) = \tfrac{1}{2}[I(y)\,V_1(y)]^2 \tag{8}$$

(b) Outflow boundaries

Zero gradient outflow boundaries are used for all the variables.

(c) Solid boundaries

Both normal and tangential velocity values are set to zero at solid boundaries.

The boundary conditions for turbulence properties near solid walls are, however, more difficult to prescribe. The k–ϵ model is only valid for fully turbulent flows. A problem therefore arises near a solid boundary where the local Reynolds number becomes very small, resulting in laminar flow. This effect is built into the turbulence model by means of wall functions. These wall functions give a better description of the shear stresses near walls, resulting in more accurate values for k and for tangential velocities near a wall [15].

(d) Free stream boundaries

All the free stream boundaries are placed far enough from the obstructions so that simplifying assumptions concerning these boundaries have no serious effects upon the flow inside the flow domain.

Free stream boundaries can be treated in at least three different ways. Firstly, the pressures and the V_1 velocities (velocities parallel to the boundary) can be described at the boundary, while no flow is allowed to cross the boundary. Secondly, flow can be prohibited from crossing the boundary, while zero gradient boundaries can be employed for V_1 velocities and pressures. Thirdly, flow can be allowed to cross the boundary, while V_1 velocities and pressures are prescribed [18]. For such a treatment, the exchange processes at the free stream boundaries are incorporated in the mathematical model by prescribing free stream pressures adjacent to the boundary. Mathematically, this results in calculating the boundary velocities perpendicular to the free stream boundary.

During numerical experiments the second boundary treatment proved to be more accurate than the first mentioned. It was also shown [5] in program applications that the second procedure produced results that were of sufficient accuracy for the purpose of this study. The boundary specifications of ref. 18 were therefore not implemented in the computer code.

A zero gradient boundary was used for all the turbulence properties.

3. Numerical flow program and applications

3.1. Program

A computer program, based on the theory discussed was developed. This was written in Fortran 5 and implemented on a CDC 750 mainframe computer.

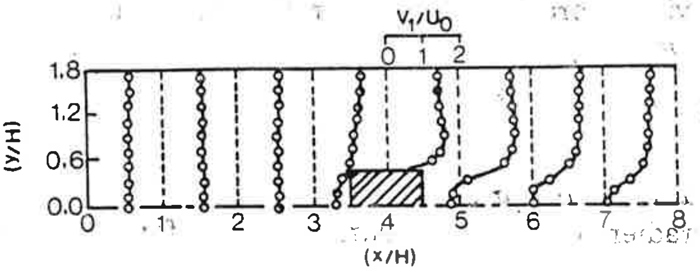

Fig. 1. Predicted V_1 velocity profiles around a model of a tall building. (Only half of flow domain shown.)

Tests were conducted for all the applications to determine the dependency of solutions upon grid size. Solutions were obtained with successive refinement of grid size, i.e. reduction in the spacing between grid nodes in the x and y directions. When solutions were observed to be unaffected by further refinements, they were presumed to be grid independent.

3.2. Turbulent flow around a tall building

Although flow around a tall building is sensitive to the vertical velocity profile, a two-dimensional analysis can present useful design information [19]. The following data were used to predict the flow around a model of a tall building, which was investigated experimentally in ref. 19:

(1) Building dimensions $= H \times H$
(2) Reynolds number of flow (Re_H) $= 2 \times 10^4$
(3) Turbulence intensity $= 10\%$
(4) Turbulence length scales (L/H) $= 0.67$
(5) Uniform grid $= 20 \times 30$

Predicted velocity profiles around the model are presented in Figs. 1 and 2. The predicted pressure coefficient distribution resulting from the flow around the model is compared with the measured distribution [19] in Fig. 3. A good agreement between measured and predicted pressure coefficients is found.

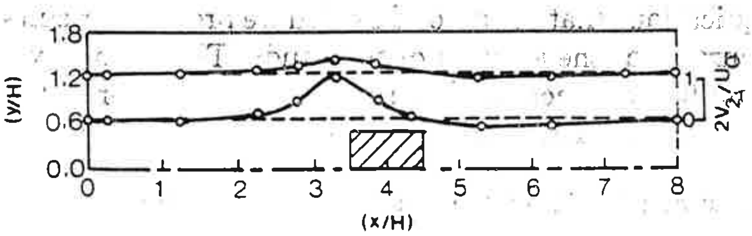

Fig. 2. Predicted V_2 velocity profiles around a model of a tall building. (Only half of flow domain shown.)

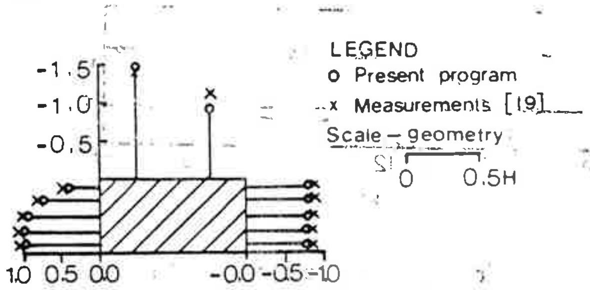

Fig. 3. Mean pressure coefficient (C_p) distribution around a model of a tall building. (Only half shown.)

3.3. *Turbulent flow over a long building*

Important data used for the computation of turbulent flow over a model of a long building were as follows:

(1) Buildings dimensions = $2H \times H$
(2) Velocity profile = eqn. (4)
(3) Reference velocity (V_{ref}) = $10\ ms^{-1}$
(4) Reference height (h_{ref}) = $2H$
(5) Mean wind speed exponent (α) = 0.3
(6) Reynolds number ($Re_{h_{ref}}$) = 7.5×10^4
(7) Values for k at inflow = eqns. (7, 8)
(8) Values for ϵ at inflow = eqns. (5, 6)
(9) Value for surface friction coefficient (k_s) = 0.03
(10) Uniform grid = 20×30

Predicted velocity profiles around the building are presented in Figs. 4 and 5. The re-attachment length was predicted at a distance of about $5H$ behind the building which is in fair agreement with that measured by Counihan et al. [20], namely approximately $6H$. The predicted pressure coefficient distribution resulting from the flow over the building is shown in Fig. 6. The pressure coefficients are normalized to the dynamic pressure at the eaves height of the building.

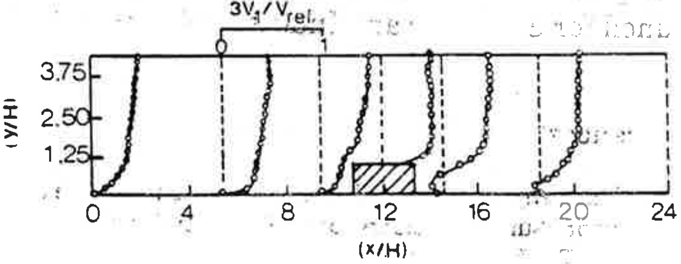

Fig. 4. Predicted V_1 velocity profile over a model of a long building.

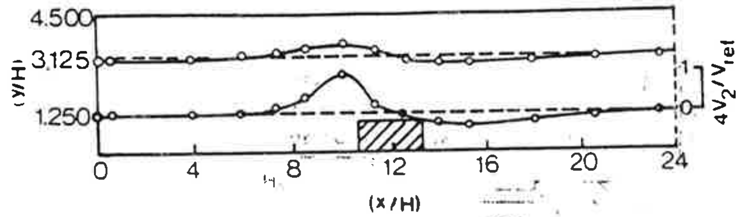

Fig. 5. Predicted V_2 velocity profile over a model of a long building.

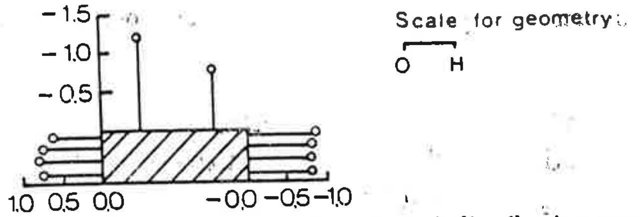

Fig. 6. Mean pressure coefficient (C_p) distribution over a model of a long building.

4. Conclusions

It was shown that a numerical technique can be used to predict wind-generated pressure distributions around buildings. These distributions can be used for the purpose of calculating natural ventilation flow rates. Although only very simple examples were given, the method is very powerful, and more complex flows can be investigated using it. Vast possibilities exist for useful extensions to the computer code, e.g. three-dimensional analyses of clusters of buildings.

It is evident from the literature that great effort is devoted to the development of hardware and software for the numerical solution of fluid flow codes. Numerical flow programs may therefore in future be used to supplement, or perhaps (in some cases) even to substitute pressure measurements in a wind tunnel!

Acknowledgement

The work described in this paper was carried out whilst the author was employed by the National Building Research Institute of the South African Council for Scientific and Industrial Research.

References

1 A.J. Bowen, A wind tunnel investigation using simple building models to obtain mean surface wind pressure coefficients for air infiltration estimates, National Research Council Canada, Rep. LTR-LA-209, Dec. 1976.
2 H.J. Banks et al., Wind-induced pressure distribution on a large grain storage shed: prediction of wind-driven ventilation rates, J. Stored Prod. Res., 19 (1983) 181–188.

3 B.E. Lee et al., Predicting natural ventilation forces upon low-rise buildings, ASHRAE J., (Feb. 1980) 35-39.

4 A.A. Field, What's new in Europe. Wind action on tall buildings, Heating, Piping and Air Cond., 42 (1970).

⑤ E.H. Mathews, The prediction of natural ventilation in buildings, D. Eng. dissertation, University of Potchefstroom, South Africa, 1985.

6 Final Report, Numerical Aerodynamic Simulation Facility, Preliminary Study, prepared under contract no. NAS2-9456 by Burroughs Corporation, Paoli, PA, Oct. 1977.

7 V.L. Peterson, Impact of computers on aerodynamics research and development, Proc. of the IEEE, Special issue on supercomputers — their impact on science and technology, 72 (1984) 68-79.

8 N.C. Markatos, Computer analysis of building-ventilation problems, CHAM Report PDR/CHAM UK/21, CHAM, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, Oct. 1983.

9 G.P. Hammond, Appropriate calculation methods for thermo-fluid problems related to buildings, Seminar on: What next in energy in buildings research? SAUS, Rodney Lodge, Bristol University, Sept. 1982.

10 T. Hanson et al., Computer simulation of wind flow around buildings, Comput. Aided Des., 14 (1982) 27-31.

11 R.H. Pletcher and S.V. Patankar, Computers in analysis and design, Mech. Eng., 73 (1983) 73-79.

12 C.F. Shieh and W. Frost, Application of a numerical model to wind energy conversion systems siting relative to two-dimensional terrain features, Proc. 5th. Conf. on Wind. Eng., Colorado State University, Fort Collins, CO, Vol. II, 1979, Pergamon Press, Oxford, 1980, IX-5-1-IX-5-10.

13 J. Bittle and W. Frost, Atmospheric flow over two-dimensional bluff surface obstructions, NASA CR 2750, 1976.

14 P.K. Yeung and S.C. Kot, Computation of turbulent flows past arbitrary two-dimensional surface-mounted obstructions, J. Wind Eng. Ind. Aerodyn., 18 (1985) 177-190.

15 B.E. Launder and D.B. Spalding, Mathematical Models of Turbulence, Academic Press, London, 1972.

16 C. Scruton, An Introduction to Wind Effects on Structures, Oxford University Press, Oxford, 1981.

17 A.D. Gosman and F.J.K. Ideriah, TEACH-T: A general computer program for two-dimensional, turbulent, recirculating flows, Lecture Notes, Department of Mechanical Engineering, Imperial College, London, 1976.

18 D.Sharma et al., Practical developments in the modelling of turbulent heat and mass transfer, Numerical Methods in Laminar and Turbulent Flow, Proc. 1st Int. Conf., Swansea, 1978, pp. 569-584.

19 A.R. Barriga et al., Pressure distribution on a square cylinder at a small angle of attack in a turbulent cross flow, Proc. 4th Int. Conf. on Wind Effects on Buildings and Structures, Heathrow, 1975, Cambridge University Press, Cambridge, 1977, pp. 89-93.

20 J. Counihan, J.C.R. Hunt and P.S. Jackson, Wakes behind two-dimensional surface obstacles in turbulent boundary layers, J. Fluid Mech., 64 (1974) 529-563.