AIR MOVEMENT – A Numerical Prediction Technique

G.E. Whittle
Building Services Research and Information Association, Bracknell, U.K.

#4913

Introduction

Air movement within enclosures, which may result from a combination of infiltration, mechanical ventilation and convective heat transfer effects, is important for considerations of thermal comfort, ventilation efficiency and energy conservation.

Until quite recently the prediction of air movement relied extensively on empirically obtained data which defined, for example, the performance of air jets, and convective heat transfer from surfaces. Techniques are now being introduced from the field of computational fluid dynamics which allow a 'whole-field' prediction of air movement based on the solution of the conservation equations.

It is intended here to briefly review a predictive technique which is based on the numerical solution of the partial differential equations using a finite volume formulation and which can be readily applied to this problem area. The use of the method is illustrated by example.

The Conservation Equations

The equations which govern air flow and convective heat transfer are the conservation equations of momentum, energy and mass. These, together with hydrodynamic and thermal boundary conditions and a turbulence model, form the basis of any rigorous approach to predicting air movement within enclosures.

The momentum equations, which are often referred to as the Navier-Stokes equations[1], comprise up to three component equations, one for each co-ordinate direction. For example, in a two-dimensional flow field, that is one which can be fully described on a plane through the flow domain where the component of fluid velocity and gradients of fluid properties normal to the solution plane are zero, only two components of the momentum equations would be applied.

Having manipulated the equations to represent turbulent flow in terms of mean-flow velocity components, temperature and density, etc., the conservation equations are shown below for the special case of two-dimensional flow in a cartesian co-ordinate system (x,y).[2, 3]

Momentum – U component

$$\underbrace{\frac{\partial(\rho U)}{\partial t}}_{\text{transient term}} + \underbrace{\frac{\partial}{\partial x}(\rho UU) + \frac{\partial}{\partial y}(\rho VU)}_{\text{convection}} =$$

$$\underbrace{-\frac{\partial p}{\partial x}}_{\text{pressure gradient}} + \underbrace{\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial U}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial U}{\partial y}\right)}_{\text{diffusion}} + \underbrace{S_U}_{\text{additional 'source' term}} \qquad (1)$$

– V component

$$\frac{\partial(\rho V)}{\partial t} + \frac{\partial}{\partial x}(\rho UV) + \frac{\partial}{\partial y}(\rho VV) =$$

$$-\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial V}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial V}{\partial y}\right) - \underbrace{(\rho - \rho\infty)\, g}_{\text{buoyancy term}} + S_V \qquad (2)$$

Energy

$$\frac{\partial(\rho T)}{\partial t} + \frac{\partial}{\partial x}(\rho UT) + \frac{\partial}{\partial y}(\rho VT) =$$

$$\frac{\partial}{\partial x}\left(\Gamma_{eff}\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_{eff}\frac{\partial T}{\partial y}\right) + S_T \qquad (3)$$

Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho U) + \frac{\partial}{\partial y}(\rho V) = 0 \qquad (4)$$

where
- U, V = velocity components of mean flow
- t = time
- ρ = density
- p = pressure
- g = gravity
- T = temperature
- μ_{eff} = diffusion coefficient for momentum
- Γ_{eff} = diffusion coefficient for temperature

S_U and S_V are additional viscous terms whose influence is small except where changes in fluid properties are considerable. Similarly the term S_T, which represents heat generation by viscous dissipation, is small and can be neglected in the processes being considered.

For isothermal flows where viscous dissipation is zero, where the boundaries of the flow domain are adiabatic and where there are no heat sources (or sinks) within the flow field, then the energy equation would become redundant.

In the above equations the laminar viscosity from the Navier-Stokes equations is replaced by an effective viscosity (diffusion coefficient) which will generally be orders of magnitude larger to account for the enhanced diffusional effects in turbulent flow.

In many situations the steady-state version of the equations are appropriate to define a flow field; in this case the transient terms would be zero. Transient effects are generally only of significance when studying changes which take place on a time scale of the order of seconds or minutes. An example of where this is important is in the study of smoke movement in an enclosure. Smoke or contaminant concentrations can be predicted using a convection-diffusion equation of similar form to Equation 3.

The diffusion coefficients are predicted in references 2 and 3 from a two-equation ($k - \epsilon$) model of turbulence, where k is the kinetic energy of turbulence and ϵ its dissipation rate (see also reference 4).

Both k and ϵ are predicted within the flow field using an equation similiar to Equation 3.

The $k - \epsilon$ model is the most widely used model of turbulence. Its value is in predicting effective diffusion coefficients and in quantifying the energy in the turbulent fluctuations within the flow field. It does, however, still rely on the use of empirically obtained constants.

The Solution of the Equations

A feature of Equations 1 – 3 is the similarity of form. This is used to advantage in the method of expressing the equations in 'discretised' or 'difference' form and in the numerical solution sequence.

The conservation equations express velocity components and fluid properties as continuous functions throughout the flow field. A procedure for solving the equations is to integrate them over finite control volumes and represent them in numerical, discretised form with the variables expressed at specific locations on a mesh of grid nodes within the flow domain. Each of the conservation equations are thus replaced by a set of linear algebraic equations (one equation for each grid node) which describe variables at grid nodes in terms of the variables at surrounding nodes. In obtaining a solution each set of equations is solved in turn. Because of non-linearities in the partial differential equations an iterative solution scheme must be adopted.

In representing the momentum and energy equations in numerical form, a 'hybrid' formulation is used to express the convective and diffusive fluxes across the boundaries of each control volume.[3] A typical control volume is shown in Figure 1.

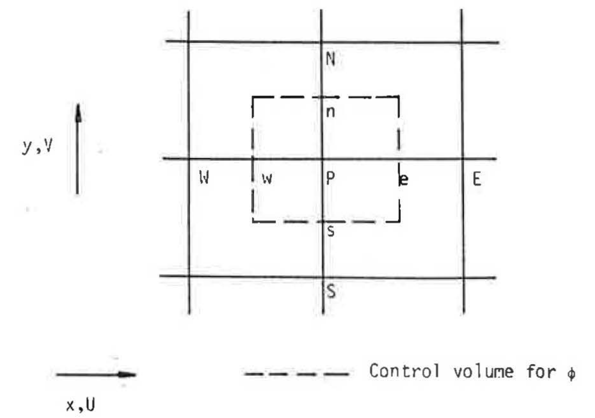

Figure 1. A control volume

It is claimed[2, 3] that the 'hybrid' formulation provides both accuracy and numerical stability of the solution scheme. In practice it is convenient to define a staggered grid system, as shown in Figure 2. Here the grids for U and V are displaced from the grid for the remaining solution variables. The reasons for adopting this approach are for simplicity in applying the pressure gradient terms in the momentum equations, and in specifying the special form of the mass continuity equation.

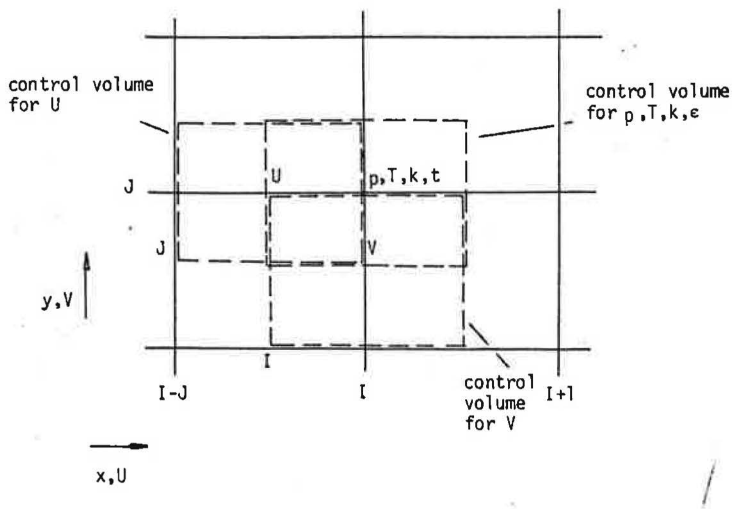

Figure 2. Staggered grid system

In solving each set of equations (one set for each variable, U, V, T etc.,) it is convenient and economical in computing requirements to use the Tri-Diagonal Matrix Algorithm (TDMA). The application of the TDMA, which here is the procedure of integrating along a grid line, is repeated in a line-by-line manner until the whole flow domain is swept. Stability of the sequence is encouraged by damping the changes in variable from one iteration to the next (under-relaxation).

The conservation equations (Equations 1 – 4) do not provide a ready means of calculating the pressure field. The way forward is to derive relationships for the rates of change of velocity components with pressure difference from the momentum equations and to combine these into a special form of the mass continuity equation. The resulting equation being solved for pressure correction at each grid node using the TDMA. The pressure and velocity fields are then updated.

The solution procedure outlined above is known as the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm.[3] A number of enhancements to this algorithm have also been developed.[5]

Boundary Conditions

In order to obtain a solution to the equations, boundary values of all variables must be applied either directly or through a boundary flux. If infiltration or mechanical ventilation effects are present then the velocity components and air temperature at the inlet to, and outlet from, the flow domain must be prescribed.

In the case of turbulence modelling, a wall function approach is appropriate. These are special formulae for dealing economically with the steep variations in flow properties near surfaces.

When solving the energy equation it is necessary to prescribe surface temperatures or surface fluxes, as appropriate. Radiant energy transfer from surfaces can be modelled either by an external calculation or through an extension to the above model.

An Application of the Method

As an example of the general approach described above, a prediction has been undertaken of the air movement and temperature distribution in a perimeter module of a building. The enclosure considered is a rectangular shaped air conditioned office space in which the influence of winter heat losses is examined. The office space is sized 5m x 2.8m height and has one external fully glazed wall. Supply air at a temperature of 27°C enters the enclosure through a diffuser located in the ceiling adjacent to the glazing. The supply air flow rate is 50 litre/s per metre run of diffuser and the discharge velocity is 2.5m/s. The flow has been computed on a two-dimensional plane through the flow field.

Figures 3 and 4 show a velocity vector plot and temperature contours. The air movement pattern is characterised by a down-wash of air from the cold glazing. The supply jet under the influence of buoyancy forces was not able to counteract the down-wash. In this case the vertical temperature gradient combined with the cool air from the glazing results in an unacceptable thermal environment. The solution was obtained on a Prime minicomputer, requiring approximately ten minutes of cpu time. Other applications exist in the field of ventilation efficiency for contaminant control, and in the study of smoke movement within buildings.[6]

A further application of numerical models relates to external air flows around buildings. Here, there exists the potential to use a model such as is described above as a numerical wind tunnel to predict pressure distributions over building surfaces.[7]

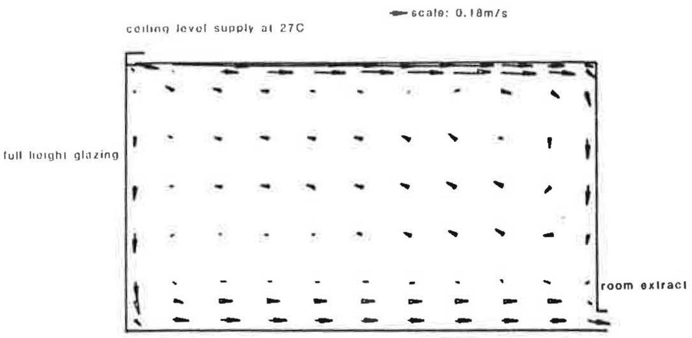

Figure 3. Velocity vector plot: winter heating

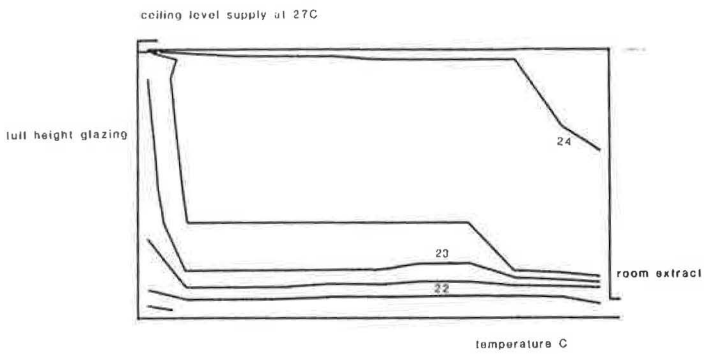

Figure 4. Temperature contours: winter heating

Concluding Remarks

The numerical procedures which have been briefly reviewed above form the basis of many fluid dynamics computer programs. They offer the ability to compute full three-dimensional steady-state and transient flow regimes and they provide a powerful and comprehensive predictive tool in air movement studies. However, the numerical techniques are still at a fairly early stage in development and so it is vital that proper engineering judgement be exercised in applying and interpreting the output from such codes.

References

1. Schlichting, H.
'Boundary Layer Theory'.
McGraw-Hill, New York, 1968.

2. Gosman, A.D. and Ideriah, F.J.K. 'Teach.
A General Computer Program for Two-Dimensional, Recirculating Turbulent Flows'.
Dept. of Mech. Eng., Imperial College, University of London, 1976.

3. Patankar, S.V.
'Numerical Prediction of Three-Dimensional Flows'.
Published in 'Studies in Convection', Edited by B.E. Launder. Academic Press, London and New York, 1975.

4. Launder, B.E. and Spalding, D.B.
'Mathematical Models of Turbulence'.
Academic Press, London and New York, 1972.

5. Patankar, S.V.
'Numerical Heat Transfer'.
Hemisphere Publishing, Washington DC, 1980.

6. Markatos, N.C. and Cox, G.
'Hydrodynamics and Heat Transfer in Enclosure Containing a Fire Source'.
Physico Chemical Hydrodynamics, Vol. 5, No. 1, pp 53–66, 1984.

7. Hanson, T., Summers, D.M. and Wilson, C.B.
'A Three-Dimensional Simulation of Wind Flow Around Buildings'.
Int. Journal of Numerical Methods in Fluids, Vol. 6, pp 113–127, 1986.

Visit to AIC by Government Minister

The AIC recently welcomed Sir George Young, Bt., MP, UK Parliamentary Under Secretary for the Department of the Environment, accompanied by Mr Ian Macpherson and Mr John Tory of the Department of the Environment.

They were given a brief review of the structure of the AIC and shown the functioning of the information service and some of the technical work undertaken by the AIC staff.