International Journal of Ambient Energy, Volume 7, Number 3 July 1986

Computation of air movement and convective heat transfer within buildings

G. E. Whittle*

SYNOPSIS
A review has been undertaken of the numerical computation of air movement and convective heat transfer within buildings. The fundamental conservation equations together with a turbulence model are described, and a numerical procedure for solving the elliptic partial differential equations is outlined. The literature on the subject, spanning the past twelve years, is reviewed. Attention is briefly drawn to examples of the use of numerical air movement codes.

INTRODUCTION

The prediction of air speed and temperature distribution in a proposed new or refurbished building is of vital interest to engineers and architects particularly when a novel heating/cooling system variant is to be used, or, when the building is of unconventional or critical design such as large atrium spaces. The information can be important for a number of reasons.

1. Thermal comfort, i.e., whether the air speeds and temperature gradients generated within a building result in a thermally acceptable working or living environment.

2. The effectiveness of the ventilation system in removing contaminants.

3. Providing specified standards of air cleanliness in a process area or operating theatre.

Where the usual design procedures may be inappropriate for a particular application then physical modelling in the laboratory can provide a means of assessing a design before it is finalized. The procedure here is to construct either a full or part scale mock-up of a module (a room or zone) of the proposed building. With careful design of the test facility, the appropriate thermal boundary conditions can be imposed, with plant and equipment provided to represent and model the performance of the air supply/extract system. A detailed programme of measurements of air velocity and temperature can then be undertaken and the design optimized against specified criteria. For the critical application this is an established way forward though sometimes time consuming and moderately expensive. The investment in carrying out such an exercise may well be insignificant when compared to the cost of rectifying problems after a building has been completed.

The harnessing of fast and flexible computer power over the past 15 to 20 years and developments in numercial computation techniques have provided a stimulus to research leading to the solution of complex, recirculating

* G. E. Whittle, MSc, PhD, CEng, MIMechE, Head of Air Distribution, Building Services Research and Information Association, Bracknell, England.

turbulent fluid flow problems. Methods now exist which are able to solve the governing non-linear elliptic partial differential equations and predict both the steady and transient velocity and temperature fields within two and three dimensional flow domains. These methods, applied to the study of the thermal environment within buildings, may now provide a supplement to existing design methods or, in certain cases, offer an alternative to existing practice.

In this paper it is the intention to outline the formulation of numerically based procedures which are centred around a solution of the fundamental conservation equations of fluid flow in a form suitable for predicting air movement, convective heat transfer and temperature distributions within building enclosures. The current status of such methods are discussed and examples of typical applications are presented.

THE CONSERVATION EQUATIONS

The equations which govern fluid flow and convective heat transfer are the conservation equations of momentum, energy and mass. These are the fundamental relationships which form the basis of any rigorous method for predicting air velocity and temperature distribution within a room or enclosure together with thermodynamic equations, a turbulence model and boundary conditions. The conservation equations in a general form can be stated as follows [1]:

Momentum

$$\frac{\partial \bar{U}}{\partial t} + (\bar{U}.\nabla)\bar{U} = -\frac{1}{\rho}\nabla p + \bar{g} + \frac{1}{\rho}\nabla.\Phi \qquad (1)$$

Energy

$$\frac{\partial T}{\partial t} + (\bar{U}.\nabla)T = \alpha\nabla^2 T + \frac{1}{\rho Cp}\left(\frac{dp}{dt} + D\right) \qquad (2)$$

Mass

$$\frac{\partial \rho}{\partial t} + (\bar{U}.\nabla)\rho = 0 \qquad (3)$$

where $\bar{U}$ = velocity
t = time
ρ = density
p = pressure
$\bar{g}$ = gravity
Φ = stress tensor due to rate of deformation of fluid
T = temperature
α = thermal diffusivity of fluid
Cp = specific heat of fluid
D = viscous dissipation

The momentum equations (1) comprise up to three component equations, one for each direction of the co-ordinate system. For example, in two-dimensional flows where the fluid movement is defined on a flat plane (i.e., where the velocity component and gradients of fluid properties normal to the plane are zero) only two components of the momentum equation need to be solved. Additionally, for isothermal flow where viscous dissipation is zero, the boundaries of the flow domain are adiabatic and where there are no heat sources (or sinks) within the flow field then the fluid temperature everywhere is constant and therefore the energy equation (2) becomes redundant.

The equations shown (1–3) need to be rearranged in a more manageable form for solution. The procedure is to decompose the instantaneous values of the velocity components and scalar quantities into time averaged and fluctuating components. The influence of the unknown Reynolds or apparent stresses and scalar fluxes, which are responsible for turbulent diffusion and which in turbulent flow dominate the diffusional processes, are eliminated by proposing a Boussinesq type "effective viscosity" model [2]. Here the unknown diffusional fluxes are assumed proportional to the gradient of mean flow properties where the coefficients of proportionality are

μ_t and $\mu_t/\sigma_{\phi t}$.

where μ_t = turbulent viscosity
$\sigma_{\phi t}$ = turbulent Prandtl/Schmidt Number for velocity component or scalar property ϕ

In the case of two-dimensional flow in a cartesian co-ordinate system (x,y) the conservation equations can be represented as follows:

Momentum

U-component

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

$$\underbrace{-\frac{\partial p}{\partial x}}_{\text{pressure gradient}} + \underbrace{\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial U}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial U}{\partial y}\right)}_{\text{diffusion}}$$

$$+ \underbrace{S_U}_{\text{additional "source" term}} \qquad (4$$

V-component

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

$$-\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial V}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial V}{\partial y}\right)$$

$$\underbrace{-(\rho-\rho_{\infty})\ g}_{\text{buoyancy term}} + S_V \quad (5)$$

Energy

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

$$\frac{\partial}{\partial x}\left(\Gamma_{eff}\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_{eff}\frac{\partial T}{\partial y}\right) + S_T \quad (6)$$

Mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho U) + \frac{\partial}{\partial y}(\rho V) = 0 \quad (7)$$

An interesting feature of (4), (5) and (6) is the similarity of form. This is used to advantage in the method of expressing the equation in "difference" form and in the numerical solution procedure [3].

Equations (4) to (7) contain eleven unknowns: 2 components of velocity, pressure, 2 density variables, temperature, 2 diffusion coefficients and 3 source terms. Clearly, more information is required before these equations can be solved. Specifically, it is necessary to identify additional relationships so that the number of unique equations matches the number of unknowns. Although the equations are shown in two-dimensional form, identical reasoning applies to the three-dimensional equations [4].

An additional relationship arises from a thermodynamic equation of state,

$$p = f(\rho, T) \quad (8)$$

Also, the density of the surroundings ρ_{∞} is a function of the density field.

$$\rho_{\infty} = f(\rho) \quad (9)$$

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

The diffusion coefficients μ_{eff} and Γ_{eff} are calculated as follows:

$$\mu_{eff} = \mu + \mu_t \quad (10)$$

where μ_{eff} = effective viscosity/diffusion coefficient
μ = laminar viscosity
μ_t = turbulent viscosity

and,

$$\Gamma_{eff} = \frac{\mu}{\sigma_T} + \frac{\mu_t}{\sigma_{T,t}} \quad (11)$$

where Γ_{eff} = effective diffusion coefficient
σ_T = laminar and turbulent Prandtl/Schmidt numbers

The turbulent viscosity μ_t is predicted in References [3] and [4] from a two equation $(k-\epsilon)$ model of turbulence, where k is the kinetic energy of turbulent fluctuations and ϵ is the dissipation rate of kinetic energy. The turbulent viscosity is related to k and ϵ by the following relationships, applicable for high Reynolds number flows.

$$\mu_t = \frac{\rho k^{1/2}}{l} \quad (12)$$

where l is a length scale representing the size of the turbulent eddies, and

$$\epsilon = \frac{C_{\mu} k^{3/2}}{l}$$

where C_{μ} is an empirically based constant.

Both k and ϵ are predicted within the flow field using an equation similar to (4), (5) and (6), i.e.,

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

$$\frac{\partial}{\partial x}\left(\Gamma_{\phi}\frac{\partial \phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_{\phi}\frac{\partial \phi}{\partial y}\right) + S_{\phi} \quad (13)$$

where $\phi = k, \epsilon$

The "sources" S_{ϕ} contain velocity gradient terms and empirical constants [3].

The $(k-\epsilon)$ model is probably the most widely used model of turbulence. However, it is still semi-empirically based. Its value is in predicting effective diffusion coefficients and in quantifying the energy in the turbulent fluctuations in the flow field.

The early development of mathematical models of turbulence is very clearly presented by Launder and Spalding [5].

NUMERICAL SOLUTION OF THE EQUATIONS

The conservation equations express the 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 so that the equations are represented in discretized form, and the velocity components and fluid properties computed at specific locations on a mesh of grid nodes within the flow domain. Each of the conservation equations are represented by a set of linear algebraic equations, one for each grid node. The procedure to solve the fluid flow and heat transfer problem is to successively solve each conservation equation in turn. Because of the non-linearities in the fundamental differential equations the process requires an iterative approach whereby each equation is linearized and solved sequentially until the solution for the numerical values of each variable is attained.

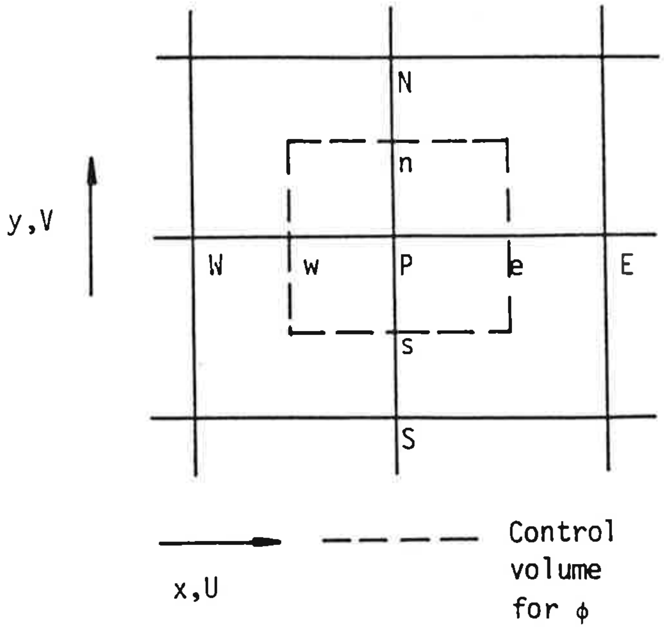

Figure 2 A control volume.

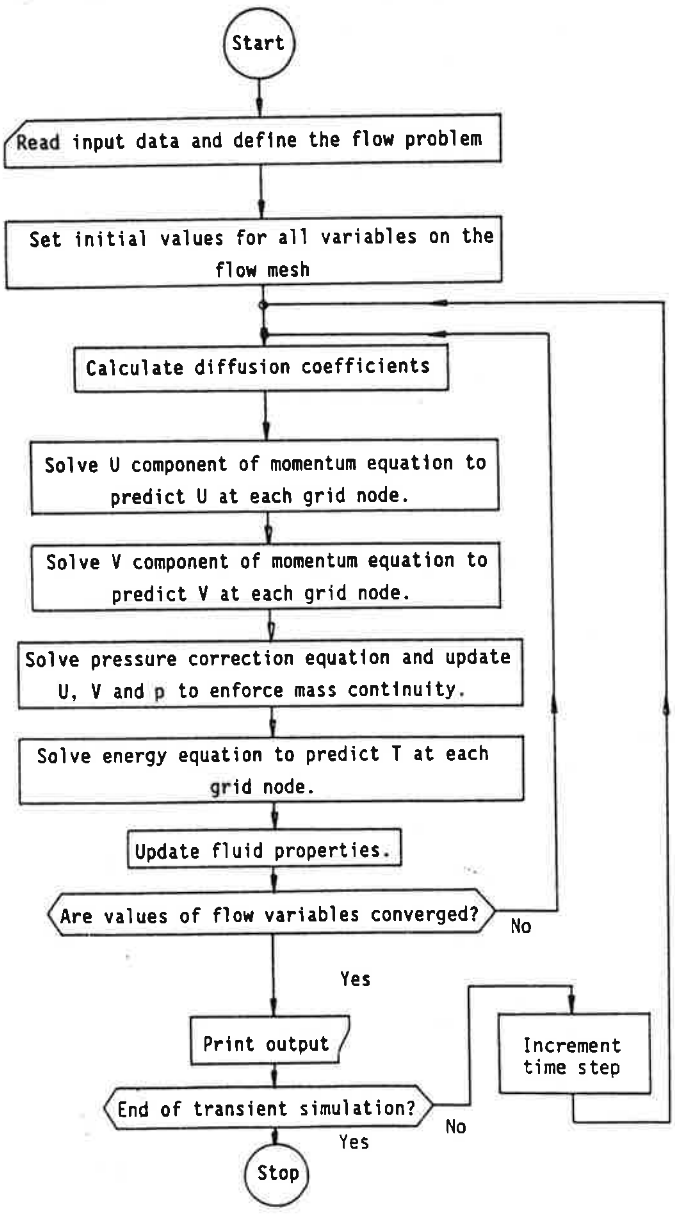

Figure 1 Outline flow chart of 2-D solution sequence.

Figure 1 shows this process in flow chart form for a two-dimensional transient simulation.

In representing the momentum and energy equations in numerical form a hybrid formulation is used to express the convection and diffusion fluxes across the boundaries of each control volume [4]. Considering the control volume shown in Figure 2, the convection and diffusion fluxes across the east face (e) are:

for $|P_e| < 2$

$$C_e = (\rho U)_e A_e \left(\frac{\phi_P + \phi_E}{2}\right) - (\Gamma\phi)_e A_e \left(\frac{\phi_E - \phi_P}{X_E - X_P}\right) \quad (14)$$

for $Pe \geqslant 2$

$$C_e = (\rho U)_e A_e \phi_P \quad (15)$$

for $Pe \leqslant -2$

$$C_e = (\rho U)_e A_e \phi_E \quad (16)$$

where C_e = combined convection/diffusion flux

Pe = grid Peclet number

$$= (\rho U)_e \left(\frac{X_E - X_P}{(\Gamma_\phi)_e}\right) \quad (17)$$

A_e = area of control volume east boundary

ϕ = U, V, T

X_E = x co-ordinate dimension of node E

Y_p = y co-ordinate dimension of node p

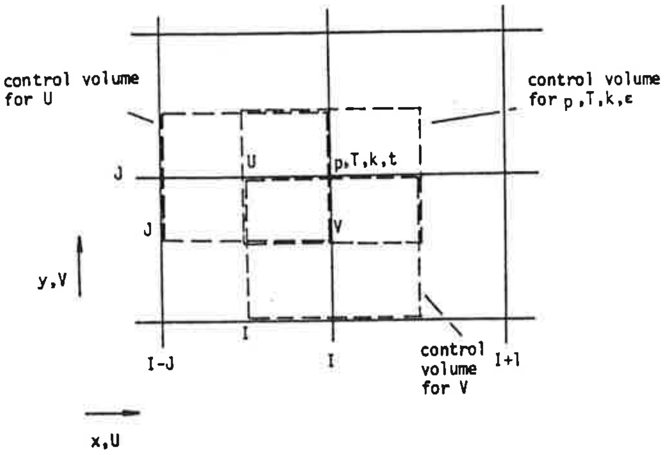

Figure 3 Staggered grid system.

The fluxes for the remaining boundaries of the control volume are defined in a similar manner.

The hybrid scheme assumes that at $|Pe|>2$ the upwind ϕ is convected downstream and diffusion is supressed. At $|Pe| < 2$ diffusion takes place normally. It is claimed [3] and [4] that this hybrid formulation provides both accuracy and stability of the solution scheme.

The transient term (see (4) and (5)) is expressed in finite difference form as:

$$(\rho_p\phi_p - \rho_p{}^0\phi_p{}^0)\ \Delta x.\Delta y/\Delta t \qquad (18)$$

where $\Delta x.\Delta y$ represent the dimensions of the control volume, and Δt a time interval during which $\rho_p{}^0\rho_p{}^0$ changes to $\rho_p\phi_p$.

In practice, it is convenient to define a staggered grid system, as shown in Figure 3. Here, the grids for U and V are displaced from the grid for the remaining variables. The reasons for adopting the staggered grid are for simplicity in applying the pressure gradient terms in the momentum equations and in specifying the special form of the mass conservation equation.

The pressure gradient in the U component of the momentum equation is identified from Figure 3 as

$$-(p_{I,J} - p_{I-1,J})\ A_{I,J} \qquad (19)$$

where $p_{I,J}$ = pressure at grid node I,J
$p_{I-1,J}$ = pressure at grid node I−1,J
$A_{I,J}$ = control volume flow area normal to the U velocity component.

When the finite difference form of all the terms in the equations have been identified they are arranged as follows:

$$a_p\phi_p = a_E\phi_E + a_W\phi_W + a_N\phi_N + a_S\phi_S + b \qquad (20)$$

where "a" are coefficients and "ϕ" the variable at each grid position.

The expressions for the pressure gradients in the momentum equations are absorbed into "b" on the right hand side of the equation, as are any additional "source" terms and values of ϕ_p at time step, t. The variable, ϕ_p, in Equation (20) is that at time step, $(t + \Delta t)$.

Thus the value of ϕ (i.e., U, V, T, etc.) is computed as a weighted function of surrounding values (see Figure 2). For example in the case of the U-component of momentum then ϕ will equal U and there will be an equation of the form of (20) for U at each grid node.

For solving the set of equations it is convenient to use the Tri-Diagonal Matrix Algorithm (TDMA) [6], where the equations can be expressed, for example, as follows:

$$a_S\phi_S + a_p\phi_p - a_N\phi_N = a_E\phi_E + a_W\phi_W + b \qquad (21)$$

The application of the TDMA procedure is the process of integrating the equation across the flow from an initial value on the south boundary to the north boundary. The whole flow domain is swept in a line-by-line manner. To ensure stability of the calculation a damping factor (df) is used such that

$$df < 1.0$$

This limits the changes in variable value from one iteration to the next.

Mass Continuity – the Pressure Correction Equation. Unfortunately, the conservation equations as shown in (4–7) do not provide a means of calculating the pressure field. The way around this difficulty is to recognize that the velocity components are partially "driven" by pressure gradients, i.e., $\partial U/\partial\Delta p$ from (4) and $\partial V/\partial\Delta p$ from (5) are both non zero. Combining this into a special form of the mass conservation equation allows a pressure correction to be computed for each grid node. The equation shown for simplicity in steady-state form is,

$$\left[U_w + \left(\frac{\partial U}{\partial(\Delta p)}\right)_w d(\Delta p)_w\right] - \left[U_e + \left(\frac{\partial U}{\partial(\Delta p)}\right)_e d(\Delta p)_e\right] + \left[V_s + \left(\frac{\partial V}{\partial(\Delta p)}\right)_s d(\Delta p)_s\right] - \left[V_n + \left(\frac{\partial V}{\partial(\Delta p)}\right)_n d(\Delta p)_n\right] = 0 \qquad (23)$$

where U and V are current estimates of the velocity components.

The terms $\partial\phi/\partial(\Delta p)$ are obtained from (20) by separating out the pressure gradient from b, so that,

$$\frac{\partial\phi}{\partial\Delta p} = \frac{A}{a_p} \qquad (24)$$

Equation (23) is solved for P at each grid node using, for example, the TDMA method. The pressure field and velocity components are updated by

$$p_{new} = p_{old} + \Delta p \qquad (25)$$

where $\phi_{new} = \phi_{old} + \dfrac{\partial \phi}{\partial(\Delta p)}.d(\Delta p)$

The solution procedure outlined above is known as the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm.

Boundary Conditions. The conservation equations are applied to control volumes within the flow domain (see Figure 4). In order to solve the equations, boundary values of all flow variables must be prescribed, either directly or through a boundary flux.

At the inlet to the flow domain, values of velocity components, temperature and other fluid properties are known. At the outlet it is usually appropriate to set normal gradients to zero. Regarding the pressure field, it is only relative pressure within the flow domain which is important. It is therefore usual to fix an arbitrary reference pressure within the flow field and compute pressures elsewhere based on the pressure gradients.

In the case of turbulence modelling adjacent to a wall, the approach used is dependent on the local Reynolds number and the distance of the adjacent grid line from the wall. Although the flow in a wall region comprises three zones (the viscous laminar sublayer, a transition zone and a fully turbulent layer) the approach is to assume that the flow can be represented by a laminar region and a turbulent region. The junction of these layers is where the linear velocity profile in the laminar region meets the logarithmic velocity profile in the turbulent sublayer [3] and [4]. Wall functions, which are special formulae for local treatment of the flow characteristics, are necessary because of the steep variations in fluid properties adjacent to a wall and limitations in the general turbulence model.

A similar philosophy of approach is used in dealing with boundary conditions in the energy equation. It is necessary either to predict heat transfer based on surface and fluid temperature or to impose a fixed heat transfer rate to represent, for example, a known heat release rate from lighting, equipment or occupants. It is important to account for both radiative and convective heat transfer. For example, heat transfer to cold glazing will occur due to a combination of convection from the air within the room and radiation from the surrounding surfaces of the room. The temperatures of surfaces, which are boundary conditions to the flow model, depend on conduction through the boundaries, convection from the surface and radiant interchange with other surfaces within the enclosure.

Other Numerical Developments. Markatos and Pericleous [6] have described the use of a refinement to the SIMPLE algorithm (SIMPLEST) which is claimed to significantly improve the rate of convergence of the iterative scheme. The modifications involve the separation of the diffusion and the convection terms in the momentum equations so that the left hand side of the discretized equations contain only the

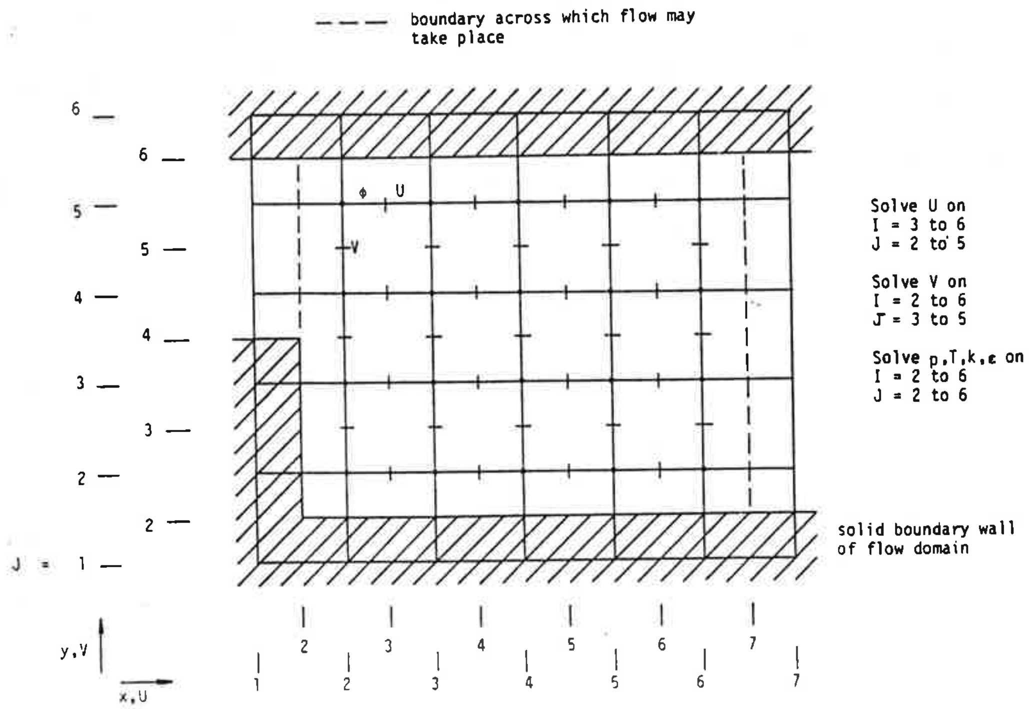

Figure 4
An example of a 2-D flow domain.

diffusion contributions; the convection terms are absorbed into the "source" term on the right hand side. In the absence of diffusion the equations are solved by an indirect Jacobi method. A further modification, which is responsible for reducing computer time requirements, is the whole-field pressure solver used in the mass continuity-pressure correction equation.

The form of the pressure correction equation in SIMPLE is identical to the basic linearized equation set in the BSRIA "LEAKS" air infiltration and mechanical ventilation prediction program. Here the equation set is solved using a sparse matrix implementation of the Choleski factorization method [6, 8]. The method takes advantage of the symmetric form and the sparseness of the matrix by storing (and therefore operating only upon) the elements which occur, inclusively, between the first non-zero element and diagonal element of each row. The routine is particularly fast and efficient.

A full implementation of a direct solution method to an equation set comprising the momentum and the mass continuity equations has been described by Vanka [9]. The solution algorithm is based on solving a block of equations for each grid node using a sparse matrix procedure. Each block of equations comprises the momentum equations and a mass continuity equation in primitive form (i.e., in terms of U, V, P). The procedure makes use of Newton's method. The advantage of retaining coupling between the momentum and mass continuity equations provides for rapid convergence of the solution sequence. A disadvantage is in the main memory requirements to execute the elimination procedure. Vanka has reduced the inpact of this disadvantage by employing domain splitting techniques so that matrix inversion is undertaken sequentially on sub-domains within the flow field.

Although each iteration takes longer than a solution method using the SIMPLE algorithm, it is claimed that the reduced number of iterations required to obtain convergence results in a significant saving in central processor (cpu) time.

Further enhancements of the SIMPLE method have been described by Van Doormaal and Raithby [10]. A number of detailed modifications have been made, notably in implementation of damping in the equations, in deriving the pressure correction relationships and in the treatment of boundary conditions in the pressure correction equation. The value of the enhancements are demonstrated by an application comparing the speed of convergence of the revised method (SIMPLEC) with the original SIMPLE and with SIMPLER (a further derivative of SIMPLE – see Patanker [11]). Significant savings in computational effort were achieved.

ROOM AIR MOVEMENT STUDIES

Thermal Range of Application. Air movement and heat transfer in rooms can be characterized as occurring within a limited range of air velocities, temperatures and heat fluxes.

The supply velocity from grilles and diffusers may be as high as 10 m/s, but air movement in the occupied part of a room should not exceed spatial mean speeds of 0.15 to 0.25 m/s [12] in most cases. Mechanical ventilation rates in air conditioned buildings are usually in the range up to 20 air changes per hour, except for clean room or process areas where very much higher ventilation rates are utilized.

A range of temperatures from 12°C to 70°C would be sufficient to encompass the operating range of supply air systems. Exposed surface temperatures of radiators on low temperature hot water (LTHW) circuits would not exceed 80°C, although high temperature radiant emitters used in factory heating could operate at a temperature of up to 1000°C.

The mean temperature within the occupied zone of a room should be in the range 16°C to 26°C depending on activity and clothing level, and temperature gradients should normally not exceed 3K [12, 13].

Combined radiative and convective heat transfer from/to room surfaces such as windows could be up to 120 W/m² (single glazing – UK winter design) and from LTHW radiators up to 3500 W/m² (expressed in terms of wall area occupied by the radiator).

Radiative heat transfer is a function of the difference in surface absolute temperature raised to the power four, together with shape factor and surface emissivities. Heat emission from plane surfaces by radiation and convection for a range of surfaces and surroundings, mean radiant temperatures and surface emissivities are shown in Section C3 of the CIBSE Guide [14]. Convection is represented by an equation of the form:

$$\dot{q}_c = C\,(T_s - T_a)^n \qquad (27)$$

where $\dot{q}_c$ = heat transfer rate W/m²
n,C = functions of surface orientation and air speed
T_s = surface temperature K
T_a = air temperature K

The effect of surface orientation is shown to increase (surface looking down) or decrease (surface looking up) the convective heat transfer rate to the surface, for equivalent surface-fluid temperature differences, by a factor of about three. Again, for equivalent surface-fluid temperature differences, air speed over the surface in the range zero to 0.5 m/s will enhance heat transfer by about 30%.

At small temperature differences the ratio of radiative to convective heat transfer from a surface can range from about 0.7 to about five depending on surface emissivity and orientation.

In the subject under discussion, namely fluid flow and convective heat transfer modelling, the equations presented at the beginning of this paper replace equations of the form shown above. Many of the dimensionless groups, familiar from fluid mechanics and heat transfer studies, can be derived from the conservation equations.

Alamdari and Hammond [15] present a detailed analysis of heat transfer data correlations for buoyancy driven laminar, transition and turbulent flows specifically for applications in computer based building thermal models. The correlations are expressed in terms of Nusselt v. Rayleigh number relationships and also for direct use as convective heat transfer coefficient v. temperature difference.

Numerical Studies. Nielsen [16] was one of the first workers to apply numerical models to the study of air movement and heat transfer within rooms. At the time of the study (1970–73) numerical predictions of turbulent recirculating flow were generally restricted to a two-dimensional analysis. This is the type of flow which can be fully represented on a plane surface, where the velocity components and gradients of fluid properties normal to the plane are zero.

The numerical approach taken by Nielsen was based on work at the Imperial College of Science and Technology, University of London, reported by Gosman [17]. In this analysis, the conservation equations for momentum and mass are manipulated to eliminate the pressure gradient terms and to define the isothermal flow field in terms of stream function and vorticity where stream function contours define stream lines, and vorticity is equal to twice the local rotation vector of the fluid. Turbulence effects were represented by using the $(k - \epsilon)$ kinetic energy and dissipation turbulence model, where the predicted velocities were those of the mean flow. The air movement in a ventilated room at low Archimedes number* was computed neglecting the influence of the buoyancy term (this is responsible for inducing flows due to temperature/density variation). Comparisons were made of the vertical velocity profile in the room and the decay of the jet showing good agreement with experimental data. A detailed comparison with experimental data was not made for flows where buoyancy was influential, although a number of resulting flow fields were presented.

In further work, Nielsen [19] has also used the model from [16] to predict the distribution of water vapour through a room using equations in the non-buoyant form. The relationship for concentration of water vapour is of the form shown in Equation (13), i.e., a convection/diffusion equation. Comparisons with test data showed the method to be appropriate for predicting humidity distribution in air conditioned rooms and cold stores.

Nielsen *et al*. [20] used a later formulation of the conservation equations in numerical form which solve the flow field in primitive variable form, i.e., velocity components and pressure. The formulation is as described earlier. An advantage of the primitive variable form of the equations is that they can readily be extended to compute three-dimensional flow configurations. This feature is not available to the stream function and vorticity formulation since stream function cannot be defined in a three-dimensional flow. The authors limited the area of the computational flow domain to exclude the initial jet development zone. The characteristics of a spreading wall jet were imposed as a boundary to the flow domain located downstream of the jet source. The reason for this was to limit the number of grid lines required for the numerical prediction. Computed results were compared, with good agreement, to measurements made using laser-dopler anemometry. The $(k - \epsilon)$ turbulence model was employed.

In [20], Nielsen *et al*. studied the influence of buoyancy in ventilated rooms. A good comparison with the experimental data, sufficient for design purposes, was achieved. An interesting feature reproduced using the numerical model was the hysteresis effect where, at certain combinations of supply air flow and heat transfer rate, two different flow patterns can be observed dependent on the start-up conditions of the system or the previous air movement pattern established. The numerical model was able to predict two different flow fields under the influence of the same boundary conditions by imposing initial conditions appropriate to each solution. In accordance with reality, this dual solution characteristic would not be manifested under most operating conditions.

Timmons *et al*. [22] have solved flow equations in stream function and vorticity form in an inviscid flow model. The approach combines use of the fundamental equations with

* Archimedes number is the ratio of buoyancy to inertia forces in the room (see Mullejans [18]. The dimensionless group is more usually defined in slightly different form as Froude number.

semi-empirical relationships for vorticity distribution. An advantage claimed for the formulation is one of rapid execution speed. The accuracy of the scheme was found to be acceptable except at near-wall regions where the inviscid approximation, not surprisingly, resulted in errors.

In [23], Gosman *et al*. used a three-dimensional numerical model to quantify the dependence of the velocity characteristics on geometrical arrangements. An interesting observation was that although differential equations were solved for the prediction of k and ϵ for use in turbulence modelling, the resulting uniformity of turbulent kinetic energy and length scale suggested that simpler algebraic relationships may be feasible thereby reducing computer time requirements.

In [24] Nielsen, using a two-dimensional numerical model, studied contaminant distributions in factory buildings, where the air flow conditions were isothermal. Some comparison was made with experimental mleasurements, and comments were made on system design practice to minimize contaminant concentrations in the occupied zone.

Holmes [25] has used the PHOENICS code to model a number of air flow configurations within enclosures. PHOENICS (Parabolic, Hyperbolic or Elliptic Numerical Code Series)* is a general purpose computer code for simulating fluid flow, heat transfer and chemical reaction processes in engineering. The basis of the method is similar to the control volume approach described earlier but with the incorporation of further developments and enhancements. Holmes considered the behaviour of vertical buoyant jets, the behaviour of wall jets when confronted by obstructions such as ceiling beams, and a full numerical prediction of air movement in a theatre. A promising correspondence with experimental data was achieved, sufficient for the author to recommend further investigatory work.

Broyd *et al*. [26] have used the CAFE (Computer Aided Flow Evaluation) code† to study air movement and temperature distributions in a number of environmental problems. The model solves the equations of motion in incompressible flow form ($\nabla.U = 0$) and uses the ($k-\epsilon$) turbulence model. Predictions of two-dimensional flow in a clean room and in three large industrial enclosures were made. The examples given showed both steady-state and transient simulations and described air velocity, temperature and contaminant concentrations. It was stated that numerical procedures are now able to predict complex turbulent air flows to useful levels of accuracy. However, caution must be exercised in the use of these techniques, and skilled engineering judgement will continue to play an important role in the formulation of the problem and in interpretation of output.

An exercise of evaluation of ventilation systems through numerical computation has been described by Ishizu and Kaneki [27]. The two-dimensional equations, in stream function and vorticity form, were solved to provide transient simulations of contaminant concentrations in an enclosure. A simplified model of ventilation efficiency was derived based on the main numerical predictions.

An analysis of three-dimensional air flow and heat transfer in a large building enclosure representing a television studio is reported by Markatos [28]. The method used is that embodied in the PHOENICS computer code. Markatos has described the formulation of the equations and identified the numerical techniques used in attaining a solution. In the paper, Markatos demonstrated the process of refining an air conditioning system design using a numerical prediction procedure.

The detailed calculation of air speeds and temperature distribution in a room has been described by Reinartz and Renz [29] in work at Aachen Technical University. The flow from a radial diffuser was considered by computing the flow field on an axisymmetric plane using a two-dimensional model which is similar to that described earlier. In reality the flow was only approximately two-dimensional although a good correspondence was achieved with measured data. The calculation procedure modelled in detail the air flow through the supply diffuser and out into the room. A grid resolution of 50 × 50 was used, and a typical run time was up to 4000 s on a Cyber 175.

Alamdari *et al.* [30] at Cranfield Institute of Technology have described methods of calculating convective heat transfer from building surfaces using lower-level analytical solutions and data correlations, intermediate-level computer codes and higher-level codes of the type being discussed in this paper. The emphasis was on deriving appropriate tools for use with building thermal models. For this purpose intermediate level codes were preferred as a means of transferring the information provided by other models in a form appropriate for the requirements of building thermal models.

The modelling of convective heat transfer in rooms has been studied by Howarth [31] at Sheffield City Polytechnic. The approach is based on combining empirically obtained expressions for convective heat transfer from heat emitters and surfaces with appropriate boundary conditions in terms of surface temperatures and outside air infiltration.

* Concentration Heat and Momentum Ltd., Wimbledon, UK.
† W. S. Atkins and Partners, Epsom, UK.

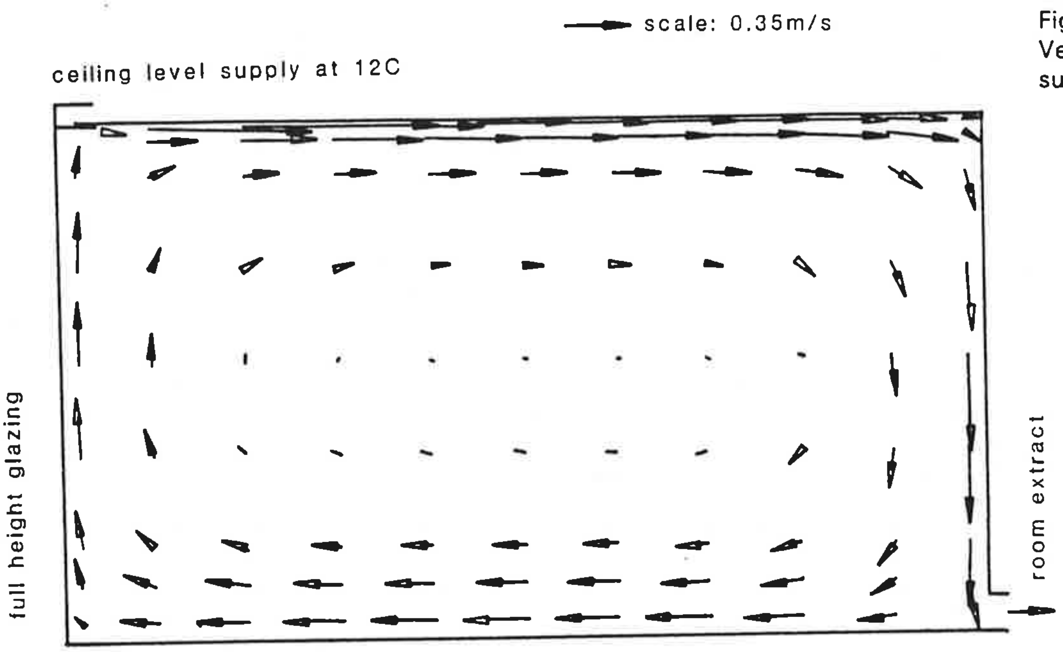

Figure 5
Velocity vector plot:
summer cooling.

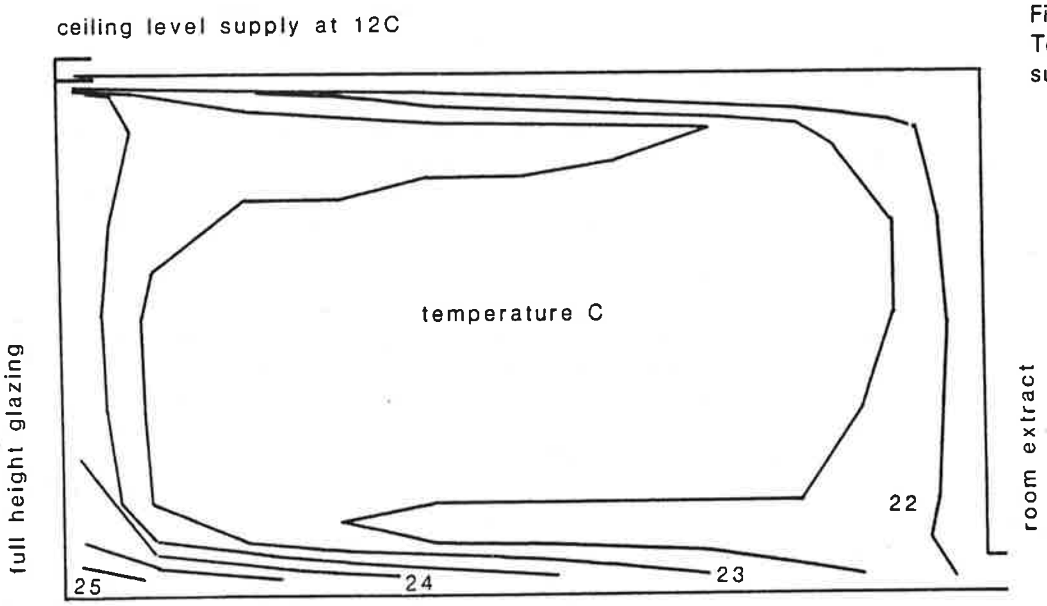

Figure 6
Temperature contours:
summer cooling.

Predicted temperature gradients are compared, with good agreement, to experimental data obtained by others.

Jones and O'Sullivan [32] at the School of Architecture, UWIST are using the PHOENICS code to predict air movement and temperature gradients in large factory buildings. The work is combined with an extensive programme of measurements, so the opportunity exists for useful model validation work. The emphasis has been in modelling flows in buildings heated by warm air systems, but also radiant heating systems are now being considered. An extension to the code to predict radiative exchange is being implemented. A body fitted co-ordinate system is employed to allow more convenient representation of pitched roofs and other non-regular building geometries.

The numerical solution of the ventilation air jet has been studied by Awbi and Setrak [33] at Napier College. The TEACH code [3] has been used to predict the velocity profiles in wall jets and the influence of the far-wall on the unconstrained projection of a jet beneath the ceiling. The effect of ceiling obstructions has been investigated and compared successfully with earlier experimental data obtained by others.

Waters [34] at Ove Arup and Partners is making extensive use of the PHOENICS code in predicting air flows and temperature gradients within a range of buildings.

Examples of Use. An example is presented below of a typical application where a numerical modelling approach can provide useful design information. The enclosure considered is a rectangular shaped air conditioned office space in which the influence of summer heat gains and winter heat losses on the air movement and temperature distribution is examined. Although design information is readily available for this type of air conditioning application there are times when modelling, either physical or numerical, can increase the level of confidence.

The numerical model is based on a development, at BSRIA, of the TEACH code [3], and the equations used are those outlined previously.

The office space is sized 5.0 m wide × 2.8 m high and is considered to be a perimeter module of a building. For demonstation purposes the flow can be represented on a vertical two-dimensional plane through the room. The perimeter wall is specified as a single glazed window of full wall height. A supply air diffuser is located in the ceiling adjacent to the glazing and discharges air along the underside of the ceiling. A return air grille is positioned at the base of the internal wall opposite the glazing. The supply air flowrate is 50 litre/s per metre run of diffuser and the discharge velocity is 2.5 m/s. Under summer conditions the thermal load on the room resulting mainly from solar and occupancy gains is 600 W/m, which is deemed to be distributed uniformly over the floor. This is offset by the supply air which is introduced into the room at 12°C. The remaining surfaces are adiabatic.

Under design winter heating the heat loss through the glazing is offset by supplying air at 27°C.

Figures 5 and 6 show the air movement and temperature conditions in the room during summer. The higher room velocities occur, as expected, during summer cooling when the buoyancy forces act with the momentum of the supply air to enhance the movement of air upwards at the glazed wall and downwards at the internal wall. During winter heating (Figures 7 and 8) the air movement pattern was characterized by a down-wash of air at the cold glazing which moves over the floor and is partially induced by the flow through the return-air grille. The supply jet under the influence of buoyancy forces was not able to counteract the down-wash from the glazing. The vertical temperature gradient in the room was approximately 3K which, combined with the down-wash of air from the glazing results in an unacceptable thermal environment.

Although no detailed comparison has been made with experimental data the predictions do demonstrate a qualitative realism.

The basic solution method can be used to compute transient flow regimes. Markatos and Cox [35] demonstrated this using a modified PHOENICS code JASMINE to model the development of a fire within a shopping mall and to predict smoke concentration levels. A key question in the event of a fire is the time interval between the fire starting and the build-up of smoke and toxic fumes which impede or prevent the full evacuation of the building. In the simulation the fire was represented by a heat source and a specified mass injection rate of contaminant. Figure 9 shows a "snapshot" of the smoke concentration level at a time instant 70 s into the fire. The influence of the strongly buoyant flow in the area of the fire caused the smoke to convect upwards to beneath the ceiling and then along the length of the building. The numerical model would provide important design information on the development of smoke concentration levels at ground level and therefore occupant evacuation times. Additionally, the need for, and the performance of, specific fire safety measures such as smoke ventilators could be readily studied.

Accuracy of Prediction. The accuracy of a numerical procedure is a function of the number of factors, particularly the convergence criterion and the grid resolution (the number of grid lines). The convergence criterion is a preset variable which is used to assess whether the iterative calculation is considered close enough to a true solution to stop the sequence and output the results. Convergence variables, which may be based on the largest residuals in the equation for U, V, P, T etc. or may reflect the change in the value of solution variables from one iteration to the next, are compared to a convergence criterion at each iteration. Selecting the value of the convergence criterion is a trade-off between accuracy and computing time.

A significant source of error associated with the grid resolution follows from the interpolation procedures used to estimate variable values between adjacent grid lines. This type of error can be minimized by increasing the number of grid lines where steep gradients in flow variables are expected. A further source of error again associated with the grid is numerical diffusion. This is a false enhancing of the diffusion processes and results from the numerical nature of the differencing equations. It can be minimized by arranging the streamwise grid lines to be near parallel to the main flow direction, and also by ensuring that the grid Peclet numbers are small. The latter requirement implies adopting a fine grid.

Other alternatives to the hybrid differencing scheme have been proposed which claim to minimize numerical diffusion effects [36].

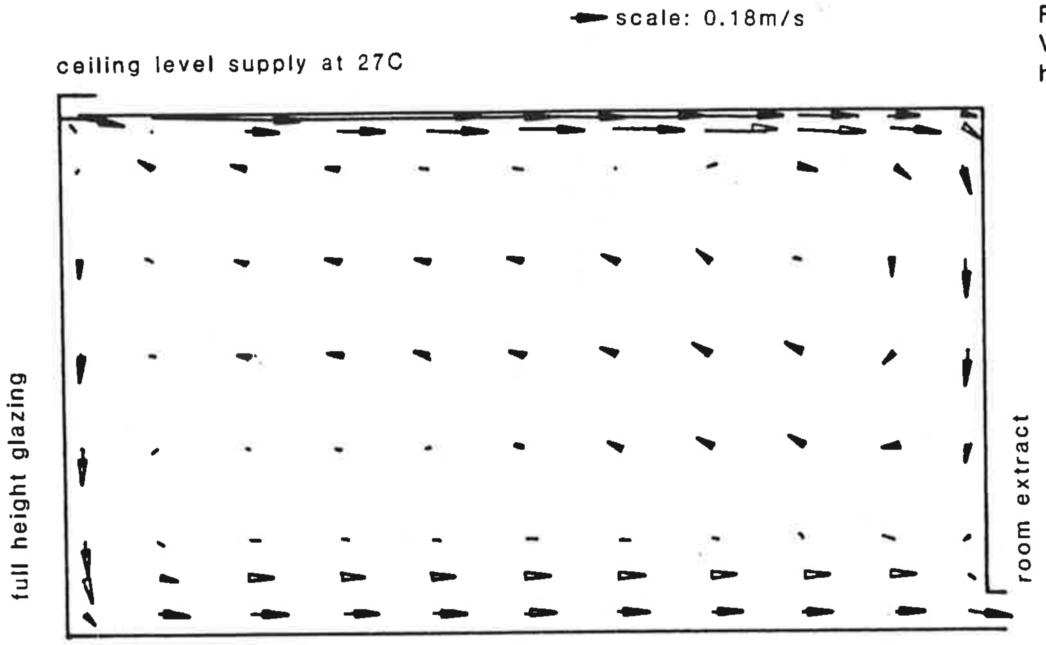

Figure 7
Velocity vector plot: winter heating.

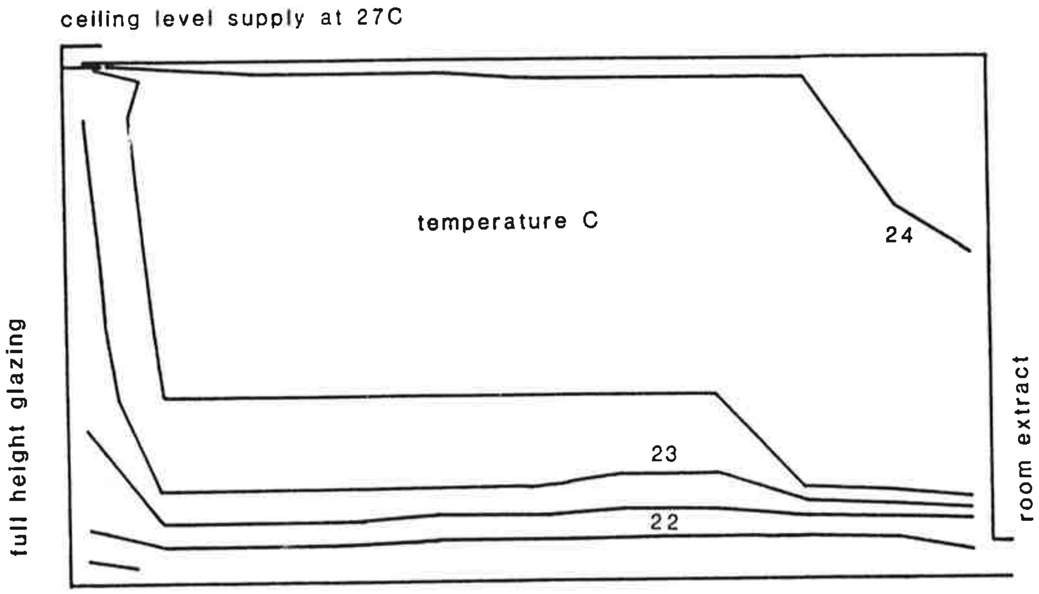

Figure 8
Temperature contours: winter heating.

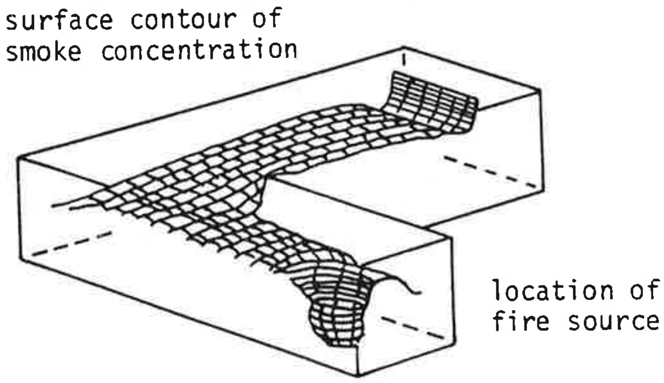

Figure 9 Smoke movement in a shopping mall [35].

CLOSING REMARKS

A considerable effort has been invested in the development and refining of numerical prediction methods for complex, recirculating, turbulent fluid flows. The spin-off from this effort is already making an impact in the field of air flows within buildings. It is certain that developments will continue and that numerical air flow codes will play an increasing part in the design of buildings and the analysis of thermal performance, ventilation efficiency and safety. It must be emphasized though that we are still at an early state of development and it is therefore vital that proper engineering judgement be exercised in applying and in interpreting the output from such codes.

NOMENCLATURE

a_P, a_W, a_E	coefficients is finite difference equations referring to grid nodes P, W, E etc.
b	RHS term is finite difference equations, part of linearized source term.
df	damping factor used to limit changes in variables between iterations.
g, g	gravitational force.
k	kinetic energy of turbulence.
l	length scale of turbulence.
n	exponent used in Equation 27.
p, $p_{I,J}$	pressure.
$\dot{q}$	heat flow rate.
t	time.
x, y	dimensions in co-ordinate system.
A_e, $A_{I,J}$	area of control volume face.
C	convection coefficient in Equation 27.
C_e	convection/diffusion flux across control volume face e.
C_p	specific heat at constant pressure.
C	turbulence constant.
D	viscous dissipation.
I, J	grid line identification.
Pe	grid Peclet number.
S_U, S_V, S_T, S	source terms in RHS of equations.
T, T_e, T_a	temperature.
U, U	velocity component and vector.
V	velocity component.
α	thermal diffusivity.
ϵ	dissipation of turbulence energy.
μ, μ_{eff}, μ_t	viscosity.
Φ	stress tensor.
ρ, ρ_x, ρ_p	density.
σ, σ_T, $\sigma_{T,t}$, $\sigma_{\phi,t}$	turbulent Prandtl/Schmidt number.
ϕ	variable U, V, T, K, ϵ.
Γ_{eff}	diffusion coefficient.

REFERENCES

1. Eskinazi, S. Vector Mechanics of Fluids and Magnetofluids, Academic Press, London and New York, 1967.
2. Schlichting, H. Boundary Layer Theory, McGraw-Hill, New York, 1968.
3. Gosman, A. D. and Ideriah, F. J. K. "TEACH – A General Computer Program for Two-Dimensional, Turbulent, Recirculating Flows" Department of Mechanical Engineering, Imperial College, University of London, 1976.
4. 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.
5. Launder, B. E. and Spalding, D. B. Mathematical Models of Turbulence, Academic Press, London and New York, 1972.
6. Williams, P. W. Numerical Computation, Thomas Nelson and Sons Ltd., London, 1972.
7. Markatos, N. C. and Pericleous, K. A. "Laminar and Trubulent Natural Convection in an Enclosed Cavity" Int. Journal of Heat and Mass Transfer Vol. 27 No. 5 1984 pp. 755–722.
8. Jennings, A. and Tuff, A. D. "A Direct Method for the Solution of Large Sparse Symmetric Simultaneous Equations" published in "Large Sparse Sets of Linear Equations", edited by J. K Reid, Acadamic Press, London and New York, 1971.
9. Vanka, S. P. "Block Implicit Calculation of Steady Turbulent Recirculating Flows" Int. Journal of Heat and Mass Transfer Vol. 28 No. 11 1985 pp. 2093–2103.
10. Van Doormaal, J. P. and Raithby, G. D. "Enhancements of the SIMPLE method for predicting incompressible fluid flow" Numerical Heat Transfer Vol. 7 1984 pp. 147–163.
11. Patankar, S. V. Numerical Heat Transfer, Hemisphere Publishing, Washington DC, USA, 1980.
12. Ashrae Standard 55-1981. "Thermal Environmental Conditions for Human Occupancy" American Society for Heating Refrigeration and Air Conditioning Engineering, Atlanta, USA, 1981.
13. CIBSE Guide. Section A1, Environmental Conditions for Design, Chartered Institution of Building Services Engineers, UK, 1978.
14. CIBSE Guide. Section C3, Heat Transfer, Chartered Institution of Building Services Engineers, UK, 1976.
15. Alamdari, F. and Hammond, G. P. "Improved Data Correlations for Buoyancy Driven Convection in Rooms" BSER & T Vol. 4 No. 3 CIBSE 1983.
16. Nielsen, P. V. "Flow in Air Conditioned Rooms" PhD Thesis, Technical University of Denmark, Copenhagen, 1974.
17. Gosman, A. D., Pun, W. K., Runchal, A. K., Spalding, D. B. and Wolfshtein, M. Heat and Mass Transfer in Recirculating Flows, Academic Press, London and New York, 1969.
18. Mullejans, H. The Similarity between Non-Isothermal Flow and Heat Transfer in Mechanically Ventilated Rooms (in German), Westdentscher Verlag, Koln and Opladen, 1966.

19. Nielsen, P. V. "Moisture Transfer in Air Conditioned Rooms and Cold Stores" Paper No. 1.2.1, Proc. of the 2nd Int. CIB/RILEM Symposium on Moisture Problems in Buildings, Rotterdam, 1974.
20. Nielsen, P. V., Restivo, A. and Whitelaw, J. H. "The Velocity Characteristics of Ventilated Rooms" Trans ASME Journal of Fluids Engineering Vol. 100 1978 pp. 291–298.
21. Nielsen, P. V., Restivo, A. and Whitelaw, J. H. "Buoyancy Affected Flows in Ventilated Rooms" Numerical Heat Transfer Vol. 2 1979 pp. 115–127.
22. Timmons, M. B., Albright, L. D., Furry, R. B. and Torrance, K. E. "Experimental and Numerical Study of Air Movement in Slot-Ventilated Enclosures" ASHRAE Trans 1980 pp. 221–240.
23. Gosman, A. D., Nielsen, P. V., Restivo, A. and Whitelaw, J. H. "The Flow Properties of Rooms with Small Ventilation Openings" Trans ASME Journal of Fluids Engineering Vol. 102 1980 pp. 316–323 .
24. Nielsen, P. V. "Contaminant Distribution in Industrial Areas with Forced Ventilation and Two-Dimensional Flow" IIR-Joint Meeting, Commission E1, Essen, West Germany, Sept. 1981.
25. Holmes, M. J. The Application of Fluid Mechanics Simulation Program PHOENICS to a few Typical HVAC Problems, Ove Arup and Partners, London, UK, 1982.
26. Broyd, T. W., Dean, R. B., Oldfield, S. G. and Moult, A. "The Use of a Computational Method to Assess the Safety and Quality of Ventilation in Industrial Buildings" Conference on Heat and Fluid Flow in Nuclear and Process Plant Safety, London, UK, 1983, pp. 65–74.
27. Ishizu, Y. and Kaneki, K. "Evaluation of Ventilation Systems through Numerical Computation and Presentation of a New Ventilation Model" Trans SHASE No. 24 1984, pp. 47–57.
28. Markatos, N. C. "The Computer Analysis of Building Ventilation and Heating Problems" Passive and Low Energy Architecture, 1983, pp. 667–675.
29. Reinartz, A. and Renz, U. "Calculation of the Temperature and Flow Field in a Room Ventilated by a Radial Air Distributor" Int. Journal of Refrigeration Vol. 7 No. 5 1984 pp. 308–312.
30. Alamdari, F., Hammond, G. P. and Melo, C. "Appropriate Calculation Methods for Convective Heat Transfer from Building Surfaces" Proc. 1st UK National Heat Transfer Conf., Leeds, 1984. Published in I.Chem.I Symp. Series No. 86 Vol. 2 1984 pp. 1201–1211.
31. Howarth, A. T. "The Prediction of Air Temperature Variations in Naturally Ventilated Rooms with Convective Heating" BSER & T Vol. 6 No. 4 CIBSE 1985.
32. Jones, P. and O'Sullivan, P. "Modelling of Air Flow Patterns in Large Single Volume Spaces" SERC Workshop: Developments in Building Simulation Programs, Loughborough University, October 1985.
33. Awbi, H. B. and Setrak, A. A. "Numerical Solution of Ventilation Air Jet". 5th Symposium on the use of Computers for Environmental Engineering Related to Buildings, CIBSE, Bath, UK, 1986.
34. Waters, R. Ove Arup and Partners, private communication, 1986.
35. Markatos, N. C. and Cox, G. "Hydrodynamics and Heat Transfer in Enclosures containing a Fire Source" PhysicoChemical Hydrodynamics Vol. 5 No. 1 1984 pp. 53–66.
36. Patel, M. K., Markatos, N. C. and Cross, M. "A Critical Evaluation of Seven Discretisation Schemes for Convection Diffusion Equations" Int. Journal for Numerical Methods in Fluids Vol. 5 1985 pp. 225–244.