5809

IMechE Seminar 'ATRIUM ENGINEERING' 26 June 1990.

AIR FLOW MODELLING IN ATRIA

G E Whittle
Arup Research and Development, 13 Fitzroy Street, London W1P 6BQ.

1. Introduction

The design of atria and other large spaces poses a considerable challenge to architects, services engineers and others concerned with environmental performance and energy use. Major areas of interest encompass air flow, temperature distribution and energy efficiency during both summer and winter operation.

Air flow is particularly important. Firstly, it provides a mechanism for relatively large scale energy transfer processes, and secondly, it has a strong relevance to thermal comfort.

In summer, overheating and temperature stratification are important factors. The need is to ensure that any overheating due to solar gain is minimised; and, for an energy-efficient design, that an opportunity exists to naturally ventilate and hence cool the occupied part of the space using outside air.

In winter, large areas of glazing in spaces of this type may cause major problems of cold discomfort. The risk is of heat loss through high level glazing generating a strong downdraught resulting in high air velocities and low local temperatures in the occupied zone.

With careful design, however, use can be made of winter and mid-season solar gain to provide an acceptable thermal environment for transient occupation, and to pre-heat ventilation air for the remainder (and main part) of the building.

The fluid dynamic and heat transfer processes which take place in these large spaces are very complex. Scale physical modelling and mathematical modelling both provide means of analysis of the flow fields, a successful analysis, however, will usually demand the complimentary use of both thermal and air flow models.

In this paper, air flow analysis methods are described which are based on reduced-scale physical modelling, single- and multi-zone mathematical models and complex computational fluid dynamics (CFD) applications.

An example of the CFD approach is presented.

2. Physical Models

Physical models offer the potential of a 'real world' analogue of a building. But because of size (and hence cost), reduced-scale modelling must be employed where the scale model maintains geometrical similarity with the building but is very much smaller. A model may employ the same 'working fluid' as that of the building, ie. air, or may use another fluid such as water. The need, though, is to ensure that the physical processes occurring in the scale model represent those in the building to an acceptable degree of accuracy. Exact correspondence is achieved by maintaining constant the important dimensionless parameter groups which characterise fluid flow and heat transfer in enclosures. Some of these groups are outlined below.

Reynolds number (Re) is the ratio of inertia and viscous forces and in an isothermal flow indicates whether the flow is laminar, turbulent or in transition. Reynolds number is:

$$Re = \rho V d / \mu \qquad (1)$$

where ρ = density
V = velocity
d = characteristic dimension
μ = dynamic viscosity

In buoyant flow it is Rayleigh number (Ra) which indicates whether the flow is laminar, turbulent or in transition. Rayleigh number is the product of Grashof number and Prandtl number. Grashof number is the ratio of buoyancy and viscous forces. Rayleigh number is:

$$Ra = \beta\, g\, \rho^2\, d^3\, \Delta T\, Cp / \mu\, k \qquad (2)$$

where β = expansion coefficient
g = gravitational acceleration
ΔT = temperature difference
Cp = specific heat
k = thermal conductivity

Archimedes number (Ar) is the ratio of buoyancy and inertia forces in a non-isothermal flow. It indicates the significance of buoyancy in defining flow patterns and can be used to quantify the trajectory of a heated or cooled jets. Archimedes number is:

$$Ar = g\, \Delta T\, d / T\, V^2 \qquad (3)$$

where T = absolute temperature

In practice, of course, it is not possible to maintain constant all these groups, and within certain ranges of flow conditions it is not necessary. Mullejans[1] has carried out non-isothermal tests in mechanically ventilated enclosures at scale factors of 1/1, 1/3 and 1/9 using air as the working fluid where Archimedes number was used as the basis of similarity. The procedure was to operate the models at the same temperature difference and adjust the velocity scale to maintain Archimedes number. The flow patterns in the three sizes of enclosure were compared and found to agree well, and to be largely independent of Reynolds number.

Baturin[2] states that in a mechanically ventilated space the requirements are:

- the Reynolds number at the inlet should indicate turbulent flow (ie. $Re \gg 2,320$);
- the Archimedes number should be strictly maintained; and
- for turbulent naturally driven convective flows the Rayleigh number should exceed 2×10^7, although Baturin added that recent work had indicated that this limit could well be significantly lower.

Using this approach the velocities in the building (the prototype) are related to the model by:

$$V_p = Vm\ (d_p/d_m)^{0.5}\ (\Delta T_p/\Delta T_m)^{0.5} \qquad (4)$$

where V = velocity in prototype (V_p) and in model (V_m)

d = characteristic length in prototype (d_p) and in model (d_m)

ΔT = temperature difference in prototype (ΔT_p) and in model (ΔT_m)

Parczewski and Renzi[3] have discussed scale modelling of air movement in enclosures and considered the criteria for thermal similarity. Experiments were carried out in a 1/4 scale air model with emphasis on the similarity of the temperature fields related to the heat transfer from the enclosures. The conclusion was that thermal similarity can be achieved at the scale considered although some limitations result due to poor scaling of radiant and convective heat transfer from surfaces within the enclosures due to the dissimilar mechanisms involved.

Reynolds[4] has considered the scaling of flows of energy and mass through stairwells and has defined the relevant scaling laws based on physical and dimensional arguments. Experimental results in a 1/2 scale model using air were found to preserve the essential features of the building flow, although some doubts were expressed about the value of 1/10 scale modelling.

Others have described the use of scale modelling using water, where a brine solution is used to represent the influence of temperature induced buoyancy (Curd[5], Linden et al[6]). This form of modelling has been carried out at a ratio of 1/30 or even 1/100.

Similarly, large scale-factors have been used (up to 1/100) in air models (eg. Shoda and Tsuchia[7]). Apart from issues of accuracy and spatial resolution one of the difficulties at these very large scale factor is in accurately measuring the resulting velocity field.

Scale modelling provides a potentially useful tool but care is needed in its application. Supplementary boundary-condition information is often needed (such as solar gain and fabric heat transfers) which usually requires the supplementary operation of a computer-based thermal model; and, as with mathematical models, testing and validation is still an important area.

3. Mathematical Models

Attention is now turned to mathematical models, which are invariably computer based.

3.1 Single-zone

Probably the simplest air flow model is that based on a single-zone analysis of heat transfer and fluid flow, where the whole of the space is deemed to be at a uniform temperature.

In this approach, which may be embodied within a dynamic thermal model[8,9], the air flow rate through the atrium is calculated based on a combination of the driving forces of buoyancy (stack), wind pressures and any mechanical ventilation. The buoyancy force is generated by the difference between the temperature in the space and that outdoors, where the indoor temperature is determined by a combination of solar gain, fabric heat transfer, occupancy / lighting / equipment gains and the air flow rate through the space. The area of ventilation openings and their pressure loss characteristic also, of course, influence the resulting flow rate and hence the indoor temperature.

In a dynamic thermal model, radiation heat transfer between surfaces will probably be modelled separately from surface convection leading to increased accuracy. A simple model may well combine the effects of radiation and convection into an enhanced surface heat transfer coefficient.

The flow rate and temperature are calculated from mass and energy conservation equations and additional empirical equations describing, for example, the characteristics of flow rate and pressure loss through ventilation openings. These latter empirical relationships replace the fundamental momentum conservation equations employed in computational fluid dynamics codes (see Section 4).

For the simple atrium shown in outline form in Figure 1, statements of mass and energy conservation for steady-state conditions take the following form:

$$\rho_o Q_1 - \rho_1 Q_2 + m_{vent} = 0 \qquad (5)$$

$$\rho_o Q_1 Cp(T_o - T_1) + m_{vent} Cp(T_s - T_1) + q_{gain} = 0 \qquad (6)$$

where Cp = specific heat
m_{vent} = mechanical ventilation mass flow rate
q_{gain} = net heat flux gain to space
Q = volume flow rate through openings 1 and 2
T_o = outside air temperature
T_1 = inside air temperature
T_s = supply air temperature
ρ_o = outside air density
ρ_1 = inside air density

The empirical equations describing the pressure loss through ventilation openings are, for each opening:

$$Q = k\ A\ \Delta P^n \qquad (7)$$

where A = area of opening
k = a flow constant
= $(2/\rho)^{0.5} C_d$
C_d = discharge coefficient
ΔP = pressure drop
n = flow exponent (0.5 for fully turbulent flow, 1.0 for laminar flow through small cracks, usually taken as 0.6 for non-crack flow)

From Bernoulli's, the pressure drop, ΔP, is rewritten in its component form, where temperature replaces density using the Boussinesq approximation and wind speed appears in the term for dynamic head. Hence:

$$\Delta P_1 = (P_o + C_{press.1}\rho V^2/2 - P_{ref}gh_1/[R(T_o + 273)]) - (P_1 - P_{ref}gh_1/[R(T_1 + 273)]) \quad (8)$$

and, for opening 2,

$$\Delta P_2 = (P_1 - P_{ref}gh_2/[R(T_1 + 273)]) - (P_o + C_{press.2}\rho V^2/2 - P_{ref}gh_2/[R(T_o + 273)]) \quad (9)$$

where C_{press} = wind pressure coefficient for surfaces containing openings 1 and 2
g = gravitational acceleration (ie. 9.81 m/s^2)
h = height of openings 1 and 2
P_o = outside reference pressure at base of building (zero)
P_{ref} = absolute pressure (101.325 kPa)
R = gas constant for air (0.287 kJ/kg K)

The expressions for pressure drop, Equations 8 and 9, are substituted into Equation 5 and the resulting equation solved for P_1, the pressure inside the atrium. Equation 6 is then solved for T_1, the temperature in the atrium, taking flow rate figures determined from Equation 5. This is a segregated approach to solving the equations. Alternatively, a coupled solution is possible where the numerical method recognises the full inherent inter-term dependencies which exist[11].

Because of the non-linearities, iterative methods must be used to achieve a solution. There are a number of numerical schemes available such as the simple bi-section method, or the more complex Newton-Raphson approach which is applied after linearisation using a Taylor series expansion. As a general comment, bi-section can be slow to converge as the solution is approached but is stable. Newton-Raphson, in contrast, provides very rapid convergence from a good initial estimate but potentially erratic convergence from initial estimates remote from the solution. Under-relaxation of the Newton-Raphson method can promote stability.

3.2 Multi-zone

The above method can be extended to a consideration of a number of zones all within a single space. A multi-zone model will provide further and more accurate information concerning temperature distribution and air movement patterns. However, the fact that the equation for momentum conservation is relatively poorly represented and turbulence effects are not modelled means that some inaccuracies will be present. Accuracy will be of particular concern, for example:

- in modelling the trajectory of a jet of supply air to predict local air movement;
- in boundary layers where flow over a surface is being driven by momentum; and
- in representing turbulent diffusion (of velocity and temperature).

Essentially, the modelling approximations inherent mean that the approach is more capable of operation with a (dynamic) thermal model (Clarke and Hensen[10], Holmes[11]) than is a CFD code, although even with this (simplified) air flow model its operation is not insignificant in the computational demand that it places on the computer system. The advantage of the method, however, is that the equation set is more easily handled than that embodied in CFD and so more consistent with thermal models, which are energy and temperature oriented rather that air movement oriented. However, a CFD model provided with the right boundary conditions would be expected to be more accurate than a zone model particularly for providing comfort-related data on air movement.

4. Computational Fluid Dynamics (CFD)

Computational fluid dynamics is the representation of the fundamental conservation equations for momentum, energy and mass in mathematical form and their solution to predict fluid flow and convective heat transfer. Applied to buildings the approach can predict detailed air velocity and temperature fields, concentrations of any contaminant throughout the space, and the spread of smoke in the early stages of a fire. On the down-side, the computational task can be enormous and it can take considerable skill and experience in procuring meaningful results. However, despite the drawbacks there is a considerable and growing interest in the application of CFD (particularly) to large buildings.

The equations to be solved are the conservation equations which are based on the fundamental laws of physics, they are shown in differential form in Figure 2. The momentum and energy equations are known as convection-diffusion equations since they describe how the velocity (in component form) and enthalpy (usually as temperature) is convected with the flow and diffused throughout the field. To these must be added equations or relationships which define the magnitude of the diffusion characteristic in turbulent flow (a turbulence model). In the k-ε turbulence model this will involve solving additional convection-diffusion equations for the kinetic energy of turbulent fluctuations (k) and its dissipation rate (ε). A model of this type will predict the diffusion coefficient as a field variable rather than as a constant. In turbulent flow the diffusion coefficient will usually be two to three orders of magnitude greater than the laminar viscosity.

An additional convection-diffusion equation will be needed if it is required to solve for contaminant distribution. In buildings contaminants will usually be passive, not causing any distortion to the flow field by virtue of density changes. In which case the contaminant equation can be solved as a post-processing exercise following the solution of the flow equations.

In order to solve the differential equations they must first be represented in numerical form, finite difference or finite element methods are used to do this (Patankar[12], Shih[13], Baker[14]). The most usual method is that called finite volume which is a form of the finite difference approach. Most of the commercial CFD codes are finite volume based. Figure 3 shows as an example the finite volume discretised equations in a form ready for solving.

The strong non-linearities demand that an iterative method be used, where an initial estimate of the solution is assumed at the start of the calculation and this is improved upon at each iteration. In strongly buoyant flows in large buildings a thousand or more iterations may be required to achieve a solution. In some cases a steady-state solution may not exist and so a judgement will need to be made on whether to implement a transient simulation. Buoyant flows are not well dealt with by the majority of the commercial CFD codes, although the literature does point to areas where major improvements can be made (see, for example, Galpin and Raithby[15]).

In all of the finite-domain methods the equations are represented on a grid (mesh) of cells (elements) on which velocities, pressures and temperatures are calculated. Some codes calculate velocities on a 'staggered grid' and others use a cell-centred (non-staggered) representation. Figure 4 illustrates (in two dimensional form) the different approaches.

Some finite volume codes are restricted to cartesian (rectangular) meshes although others allow an I, J, K structured mesh to be distorted to fit irregular boundaries (body-fitted co-ordinates). Other finite volume codes give further degrees of mesh flexibility by allowing an unstructured mesh to be generated, characteristic of the finite element approach. The accuracy of CFD simulations is very sensitive to the mesh resolution (the number and disposition of cells).

The benefits of each numerical method are usually discussed and compared in terms of the accuracy of the solution and in terms of computational efficiency, which is the efficiency with which a solution is achieved and includes computation time and memory requirements. The computational requirements of CFD are extremely onerous, a three-dimensional simulation involving say 20,000 cells (which may be a relatively coarse calculation) demands the solution of a matrix of 100,000 equations up to or exceeding 1,000 times.

The vast amounts of data generated by a CFD simulation requires a good interactive graphics-based post-processor to interpret the results, and indeed a flexible input and mesh-generation system to set up the application in the first place.

As noted above, the analysis may form part of a dynamic thermal model where the temperature and flow rate are calculated as functions of time. However, the time constant of air movement in a building is very much shorter than that associated with heat transfers into and out from storage in fabric and it is often justified to take 'snap-shots' in time of the resulting air movement field. This may be done by operating a dynamic thermal model (which will probably have a simplified air movement model built into it) to generate the boundary conditions required by the CFD code. These boundary conditions will comprise surface temperatures and all convective heat fluxes. It is the convective gains which are of interest to the CFD code, radiation heat transfer should be modelled or accounted for in the thermal analysis.

A more rigorous approach would be to integrate the operation of the thermal model and the CFD code so that they solve the heat transfer and fluid flow equations in transient form. This is very time-consuming for the computer (and for the user) and is not recommended except as a research exercise. Apart from the time requirements of solving the CFD equations, a need to accurately resolve the evolution of the air movement field (in order, for example, to study control action) would dictate an integration time step of something like one second, which is two or three orders of magnitude more demanding than that required by the fabric calculation.

As an example of the computational demand of CFD in this context, it has been found that in carrying out a transient analysis of air movement and heat transfer in a perimeter office space, using a combined dynamic thermal code (R2D2) and CFD code (AIRFLO), that the CFD analysis was responsible for 99% of the computer time (Holmes et al[16]). The remainder of the computational time being devoted to weather data handling, finite difference fabric conduction, glazing analysis, inter-surface radiation heat transfer, and simulation of warm air heating plant and controls.

As a general observation on the use of commercial CFD codes in building applications there are a number of areas where further work is need, these are:

- modelling of turbulence;
- need for integration with surface-to-surface radiation model;
- need for interaction (not necessarily integration) with a dynamic thermal model;
- need for faster convergence for buoyant flows;
- need for improved mesh generation methods and user interfaces.

However, with the careful exercise of engineering judgement none of the above shortcomings preclude the use of any of the mainstream CFD codes in the buildings field.

5. Example of Use

As an example of the use of CFD, Figures 5 and 6 show air movement patterns within a symmetric half of a 6 floor atrium. Both winter and summer conditions are shown. The lower three floors of the building are open to the atrium and run off the figure to the right hand side; the upper three floors are partitioned off for reasons of fire safety, and in these simulations are not modelled.

The thermal conditions in this atrium, being air conditioned from the lower three floors and at the base, are not particularly demanding. Nevertheless, the value of the simulations is in demonstrated the extent of the cool downdraught in winter as generated by heat loss through the glazed roof, and its dilution at high level by upward moving warmer air. The year-round air change rate in the atrium is between 1 and 2 changes per hour from the floors and from the base of the atrium.

In summer, the air movement beneath the glazed roof has been suppressed due to thermal stratification (buoyancy). Corresponding plots of temperature distribution (not shown) demonstrate that the thermal stratification is contained at high level and that, as expected in this design, comfort conditions are maintained at the base of the atrium and at the level of the three lower floors.

The boundary conditions for the air flow simulation, comprising surface temperatures, were generated a priori using the Ove Arup Partnership dynamic thermal model ROOM[17].

6. Conclusions

This paper has reviewed both physical and mathematical air flow modelling methods applied to atria and large buildings.

Scale physical modelling can provide a means for evaluation of designs although care must be taken concerning the derivation and representation of boundary conditions, and testing and validation. Air-based and water-based models are in current use.

Simple mathematical models can be constructed based on single-zone analysis of heat transfer and air flow. Such models are useful at the design stage. In some cases they are integrated with dynamic thermal models.

Multi-zone models of a single space provide more information and are can also be integrated with thermal models.

Computational Fluid Dynamics models solve the fundamental physical equations of fluid flow and convective heat transfer. They can provide very detailed predictions of air velocity and temperature distribution but require substantial provision of computing resources and, in building applications, considerable skill and experience on behalf of their operator. Some technical shortcomings exist in most mainstream commercial CFD codes although none preclude their use in buildings.

Acknowledgments

The author wishes to acknowledge the benefit of many valuable discussions with his immediate colleagues in Arup Research and Development, namely, Mike Holmes, Joe Lam and Kevin Ruddick.

References

1. Mullejans, H. The similarity between non-isothermal flow and heat transfer in mechanically ventilated rooms. Westdentscher Verlag, Koln and Opladen, , 1966, (HVRA translation).

2. Baturin, V V. Fundamentals of Industrial ventilation. Pergamon Press, Oxford, 1972.

3. Parczewski, K I. and Renzi, P N. Scale model studies of temperature distributions in internally heated enclosures. ASHRAE Trans., Vol 69, pp 453-463, 1963.

4. Reynolds, A J. The scaling of flows of energy and mass through stairwells. Building and Environment, Vol 21, pp 149-153, 1986.

5. Curd, E F. Industrial ventilation design by hydraulic modelling. International ventilation and safety conference, Barbican Centre, London March 27th - 29th, 1984.

6. Linden, P F., Lane-Serff, G F. and Smeed, D A. Emptying and filling boxes: the fluid mechanics of natural ventilation. J. Fluid Mech., Vol 212, pp 309-335, 1990.

7. Shoda, T. and Tsuchiya, T. Scale model experiments on cooling and heating of a large air space. J.SHASE (Japan), No 16, pp 37-48, June, 1981.

8. Clarke, J A. Energy simulation in building design. Adam Hilger Ltd., Bristol and Boston, 1985.

9. Holmes, M J. The concept of the dynamic thermal model. Int. J. of Ambient Energy. Vol 7, No 3, 1986.

10. Clarke, J A. and Hensen, J L M. An approach to the simulation of heat

and mass flow in buildings. Energy Simulation Research Unit, University of Strathclyde, Scotland.

11. Holmes, M J. Private communication, Arup Research and Development, 1989.

12. Patankar, S V. Numerical heat transfer. Hemisphere Pub., Washington DC, USA, 1980.

13. Shih, T M. Numerical heat transfer. Hemisphere Pub., Washington DC, USA, 1984.

14. Baker, A J. Finite element computational fluid mechanics, Hemisphere Pub., Washington DC, USA, 1983.

15. Galpin, P F. and Raithby, G D. Numerical solution of problems in incompressible fluid flow: treatment of the temperature-velocity coupling. Numerical heat transfer, Vol 10, pp 105-129, Hemisphere Pub., 1986.

16. Holmes, M J., Lam, J K-W., Ruddick, K G. and Whittle, G E. Computation of conduction, convection and radiation in the perimeter zone of an office space. Roomvent 90, Oslo, Norway, June 1990.

17. ROOM - A computer model for the dynamic thermal analysis of rooms. OASYS, Ove Arup Partnership, 1988.

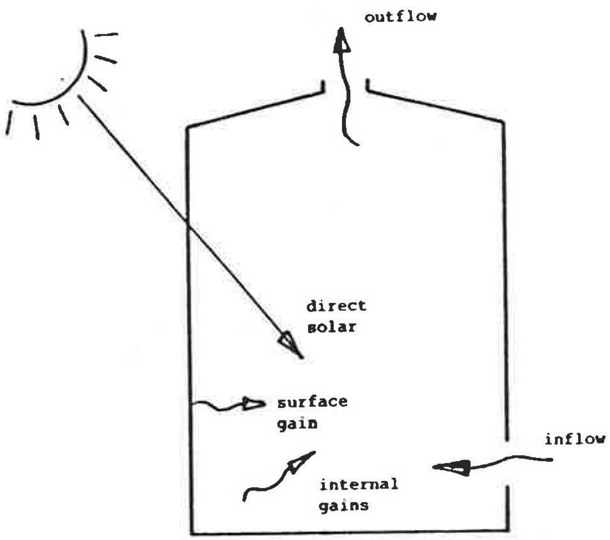


Figure 1 Schematic of atrium

Momentum Equation in X direction

$$\underbrace{\frac{\partial}{\partial t}(\rho U)}_{\text{transient term}} + \underbrace{\frac{\partial}{\partial x}(\rho UU) + \frac{\partial}{\partial y}(\rho VU) + \frac{\partial}{\partial z}(\rho WU)}_{\text{convection}} = \underbrace{-\frac{\partial P}{\partial X}}_{\text{pressure gradient}}$$

$$+ \underbrace{\frac{\partial}{\partial X}\left(\mu_{eff}\frac{\partial U}{\partial X}\right) + \frac{\partial}{\partial Y}\left(\mu_{eff}\frac{\partial U}{\partial y}\right) + \frac{\partial}{\partial Z}\left(\mu_{eff}\frac{\partial U}{\partial Z}\right)}_{\text{diffusion}}$$

$$+ \underbrace{(\rho - \rho_\infty)\, g_x}_{\text{buoyancy}} + \underbrace{S_u}_{\text{momentum source or sink}}$$

Similar equations to the above exist for the Y and Z directions.

Mass Conservation Equation

$$\underbrace{\frac{\partial \rho}{\partial t}}_{\text{transient term}} + \underbrace{\frac{\partial}{\partial X}(\rho U) + \frac{\partial}{\partial Y}(\rho V) + \frac{\partial}{\partial Z}(\rho W)}_{\text{convection}} = 0$$

Energy Conservation Equation

$$\underbrace{\frac{\partial}{\partial t}(\rho h)}_{\text{transient term}} + \underbrace{\frac{\partial}{\partial X}(\rho Uh) + \frac{\partial}{\partial Y}(\rho Vh) + \frac{\partial}{\partial Z}(\rho Wh)}_{\text{convection}} =$$

$$\underbrace{\frac{\partial}{\partial X}\left(\Gamma_{eff}\frac{\partial h}{\partial X}\right) + \frac{\partial}{\partial Y}\left(\Gamma_{eff}\frac{\partial h}{\partial Y}\right) + \frac{\partial}{\partial Z}\left(\Gamma_{eff}\frac{\partial h}{\partial Z}\right)}_{\text{diffusion}} + \underbrace{S_h}_{\text{energy source or sink}}$$

where:

c_p	=	specific heat
g_x	=	gravitation acceleration in X-direction
h	=	enthalphy (C_p.T)
P	=	pressure
S_U, S_h	=	sources (sinks) of momentum, energy
t	=	time
T	=	temperature
U, V, W	=	velocity components in X,Y,Z directions
X, Y, Z	=	co-ordinate directions
ρ, ρ_∞	=	density and reference density
μ_{eff}, Γ_{eff}	=	diffusion coefficients for momentum, energy.

Figure 2 The conservation equations

Discretised momentum equation in X direction

$$a_P U_P = a_N U_N + a_S U_S + a_E U_E + a_W U_W + a_H U_H + a_L U_L + b_U$$

where a = convection / diffusion fluxes
- a function of (C, D)

C = convection (mass) flux

D = diffusion flux

$$a_P = a_N + a_S + a_E + a_W + a_H + a_L + Vol.\rho_P / dt$$

$$b_U = \text{momentum source} + Vol.\rho_P^o.U_P^o / dt$$

Vol = cell volume

ρ = density

U = computed velocity

U^o = velocity at previous time step

dt = time step

Subscript P refers to the in-cell variable value, and subscripts N, S, E, W, H and L refer to surrounding north, south, east, west, high and low variable values.

Superscript 'o' refers to the variable value at the previous time step.

The form of the function defining a_N, etc, follows from the selection of the discretisation scheme, eg. upwind, central, hybrid differencing.

Discretised mass continuity equation

$$(\rho_P - \rho_P^o).Vol / dt + C_N + C_S + C_E + C_W + C_H + C_L + b_c = 0$$

where b_c = mass source

Discretised energy equation

$$a_P h_P = a_N h_N + a_S h_S + a_E h_E + a_W h_W + a_H h_H + a_L h_L + b_h$$

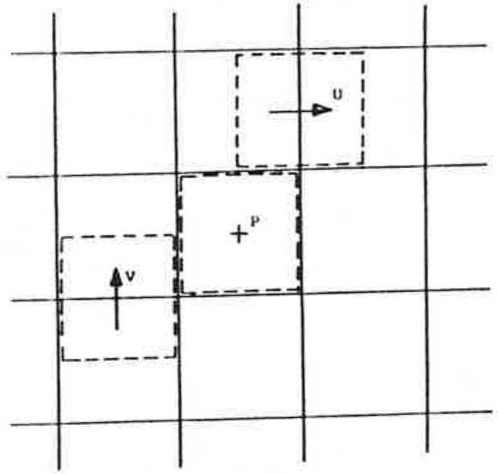
where h = enthalpy ($C_p T$)

Cp = specific heat

T = temperature

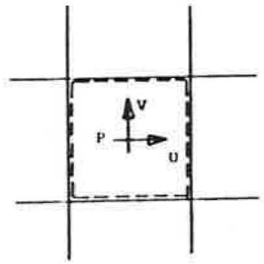
$$b_h = \text{energy source} + Vol.\rho_P^o.h_P^o / dt$$

Figure 3 The discretised conservation equations



staggered grid

- velocity components solved at cell faces



non-staggered grid

- velocity components solved at cell centres

Figure 4 Staggered and non-staggered grid

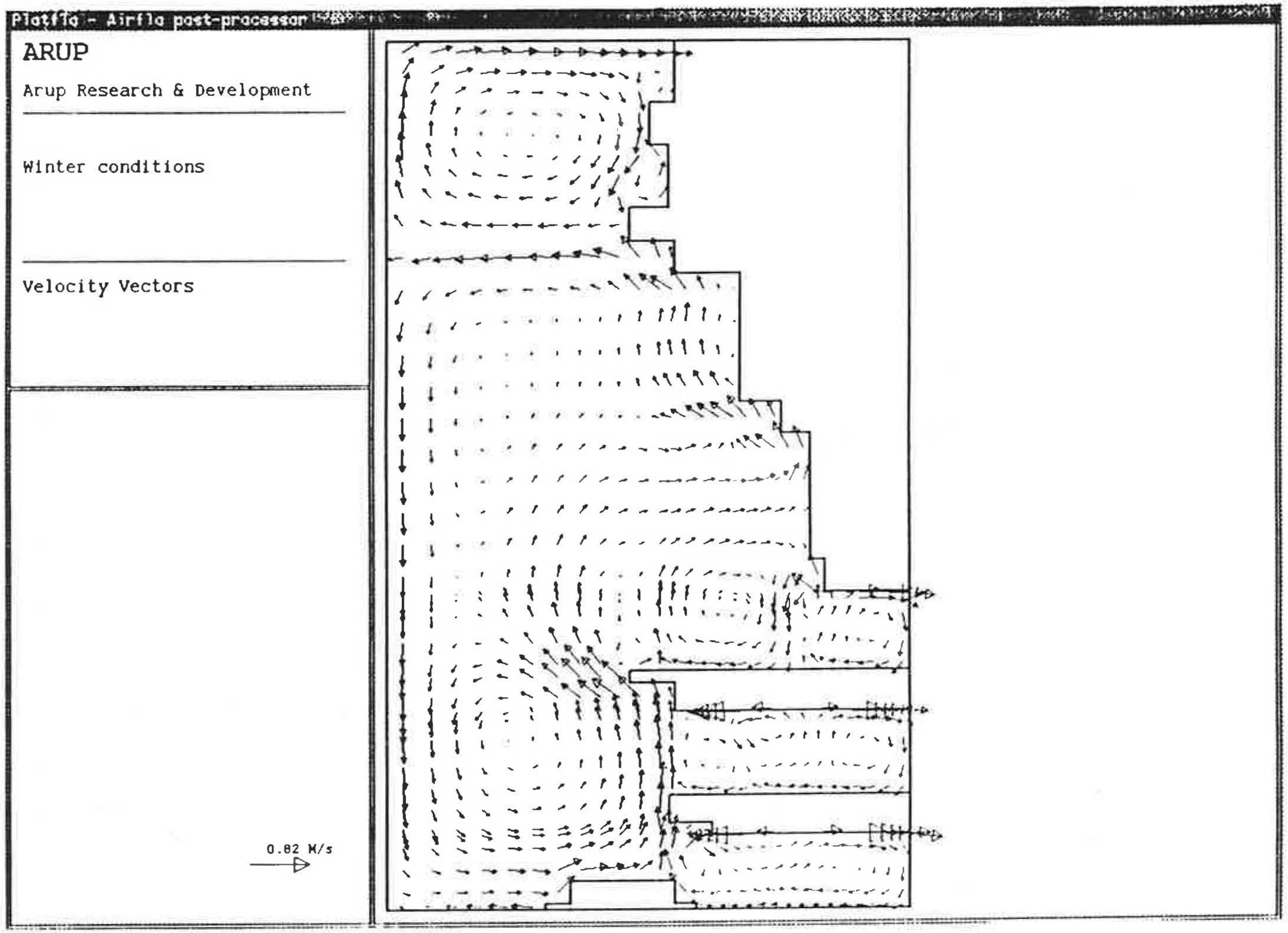


Figure 5 Atrium air flow in winter

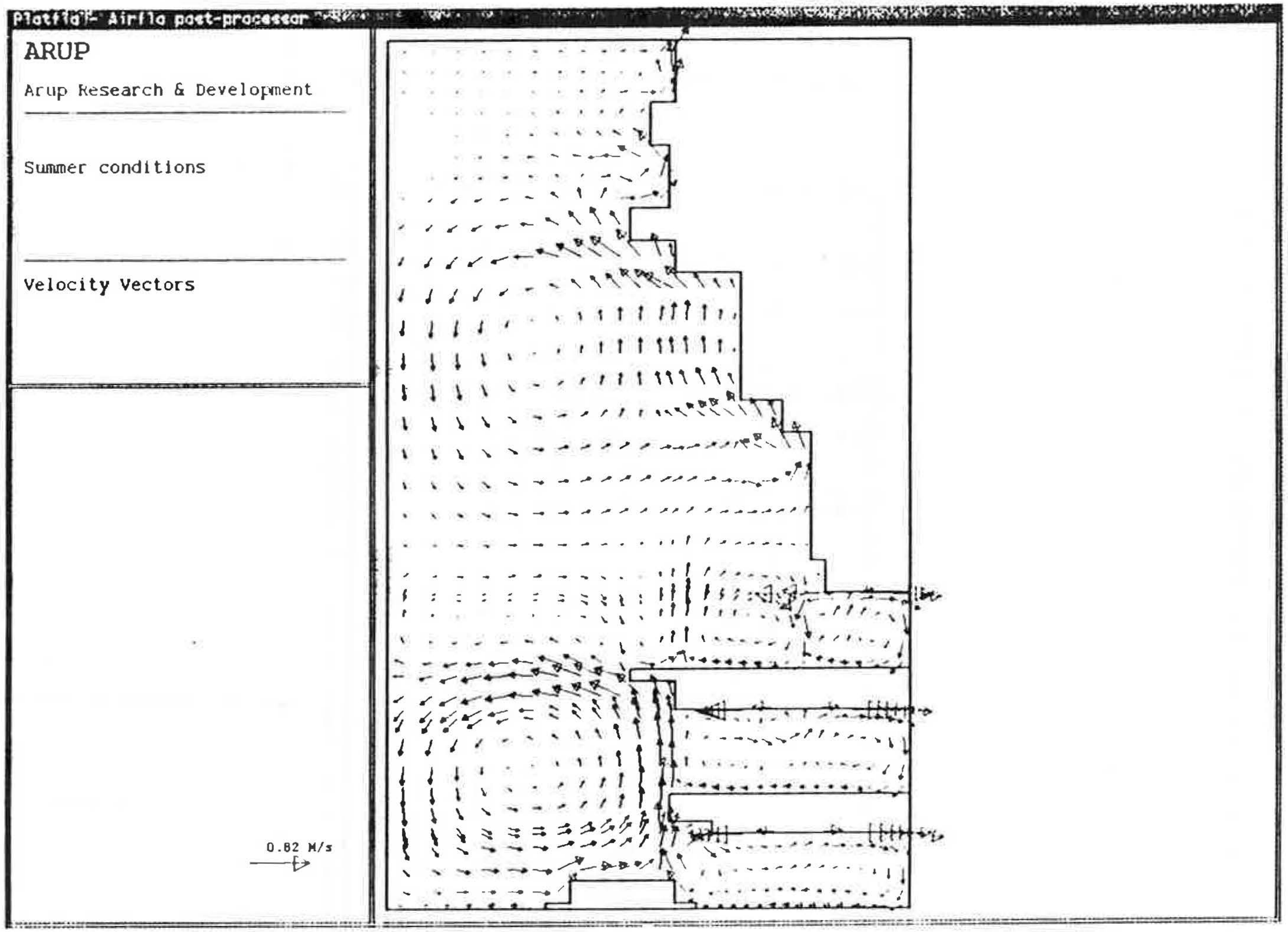


Figure 6 Atrium air flow in summer