BEPAC

(Building Environmental Performance Analysis Club)

PAPERS PRESENTED AT THE JOINT AIVC/BEPAC WORKSHOP:

*'BUILDING AIRFLOW SIMULATION:
USABILITY OF CFD & ZONAL MODELS IN PRACTICE'*

22 OCTOBER 1992

British Gas Midlands Research Station, Solihull

Contents:

Further copies of these papers can be obtained from: BEPAC Administration, BRE, Garston, Watford, WD2 7JR. Tel: 0923 664132. Fax: 0923 664780. Price: £5.00 (incl. p&p).

CFD and Network Flow Modelling at the University of Strathclyde

Joe Clarke
TF&EE and ESRU

- Work at the Department of Mechanical Engineering, University of Strathclyde.
- Network approach to fluid flow modelling in the context of ESP-r system.
- CFD developments underway and planned.
- Applications.
- Approaches contrasted (in context of buildings) and future needs.

Network approach:

- Within ESP-r the network approach to fluid (air or water) flow simulation is employed:
  - As a free-standing program for use in cases where buoyancy effects are time-invariant.
  - And as an encapsulation within the overall simulator, bps, for use in cases where buoyancy has a strong temporal dimension.
  - Many applications in a building design context:

    Building infiltration and natural ventilation.
    Mechanically driven air flow through HVAC network.
    Flow of plant working fluid.
    Estimation of convective flows within building spaces.

CFD approach:

- PHOENICS (finite volume), ANSYS/FLOTRAN (finite element) and in-house (finite difference) applications:

  Two phase flow.
  Impeller flow in turbomachinery.
  3-D flow in compressor cascade.
  Natural convection in solar panel.
  Forced convection in oil piping.
  IC combustion engine.
  Free surface flow.
  Metallic melting process.
  Non-Newtonian fluid flow in medical science.
  Building engineering.
  Flow around oil platform.
  Pollution monitoring.
  Incompressible flow around ships.
  Irrigation and flood control.
  Drainage system for road construction.
  Mass transfer through multi-layer media.

Network and CFD approaches:

• Nodes represent (parts of) spaces and (parts of) plant components possessing either known or unknown conditions.	• Nodes represent small volumes or elements of a domain.
• Inter-nodal connections represent distributed flow paths (cracks, doors, ducts, pipes, vents, etc.)	• Inter-nodal connections represent domain topology.
• Empirical relationships represents flow as a function of pressure difference across connection (eg $v = C\Delta P^n$).	• Mathematical models represent the physical flow phenomena at the cell level (eg eddy viscosity or Reynolds stress models of turbulence).
• Conservation of mass and energy leads to set of simultaneous, non-linear equations representing a non-linear problem.	• Conservation of mass, momentum, energy and species leads to set of simultaneous algebraic equations (after discretisation) representing a non-linear problem.
• Iterative technique employed to solve equations.	• Iterative technique employed to solve equations.

Network definition:

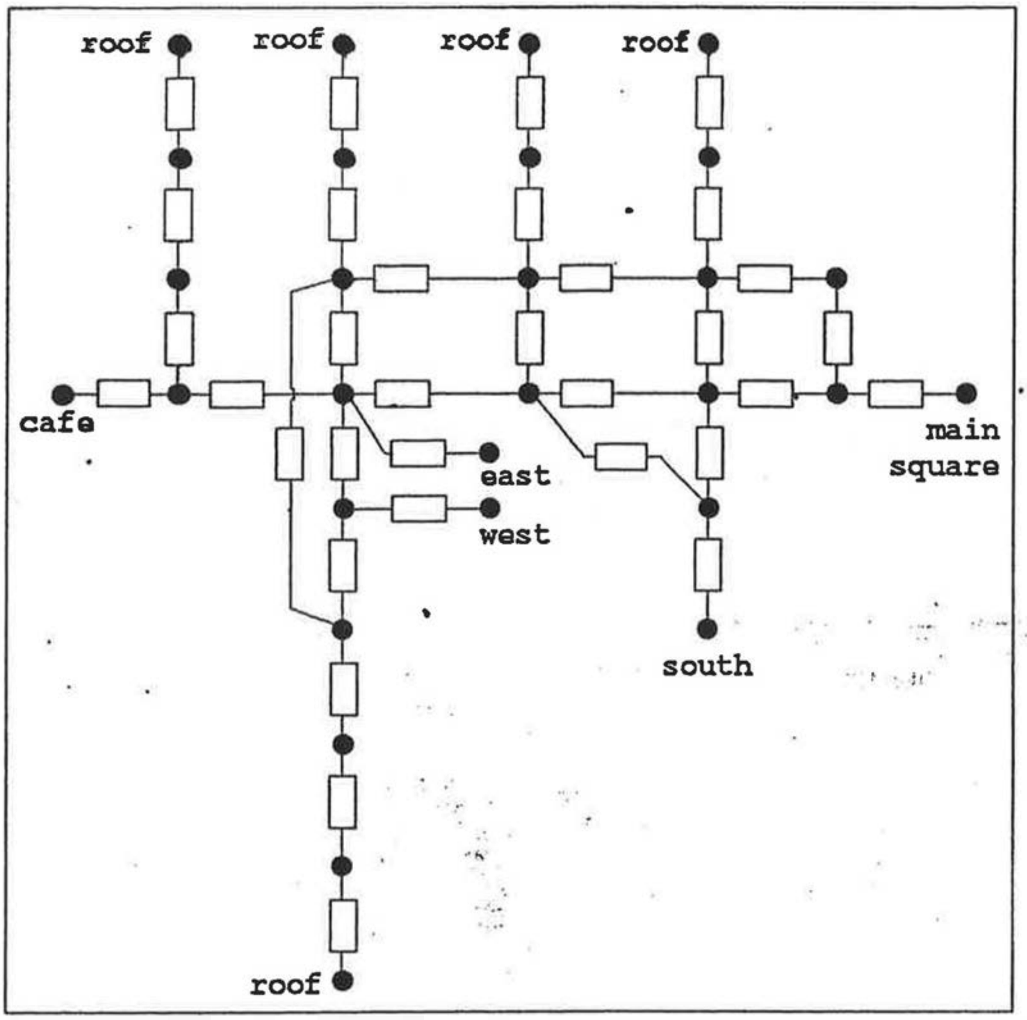
- Fluid type, node descriptions, flow component types, interconnection definition and boundary conditions. (cf multi-grid techniques of CFD.)

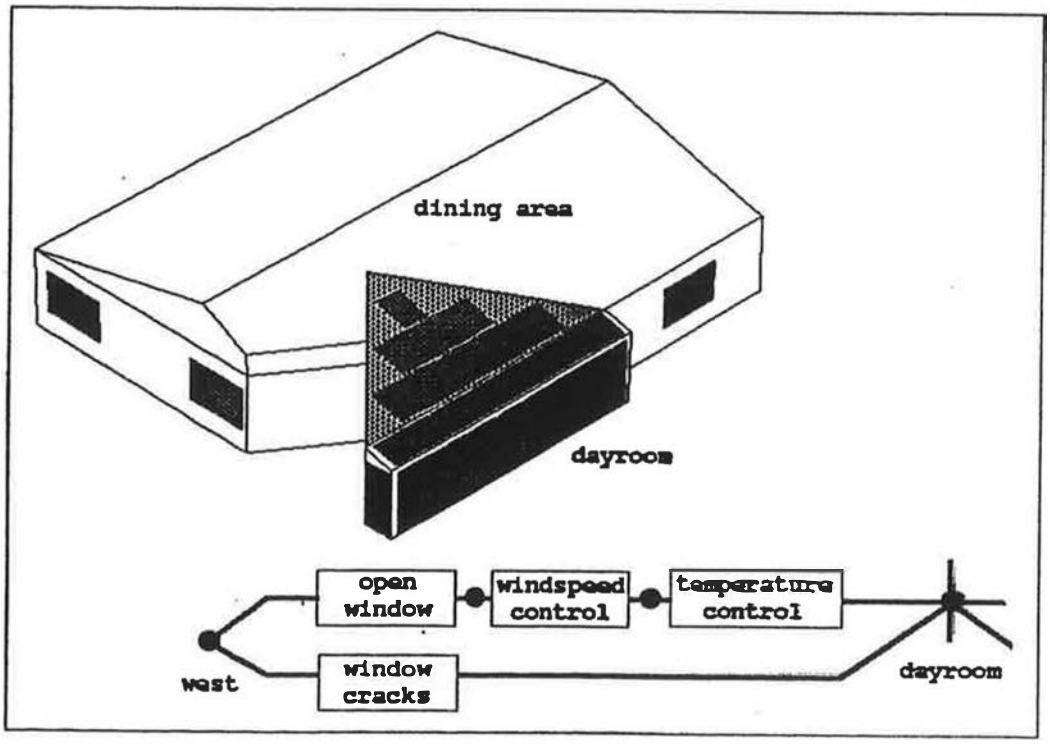
- Boundary conditions can be known pressures or pressure coefficient sets. (cf pressure and velocity field requirements of CFD).

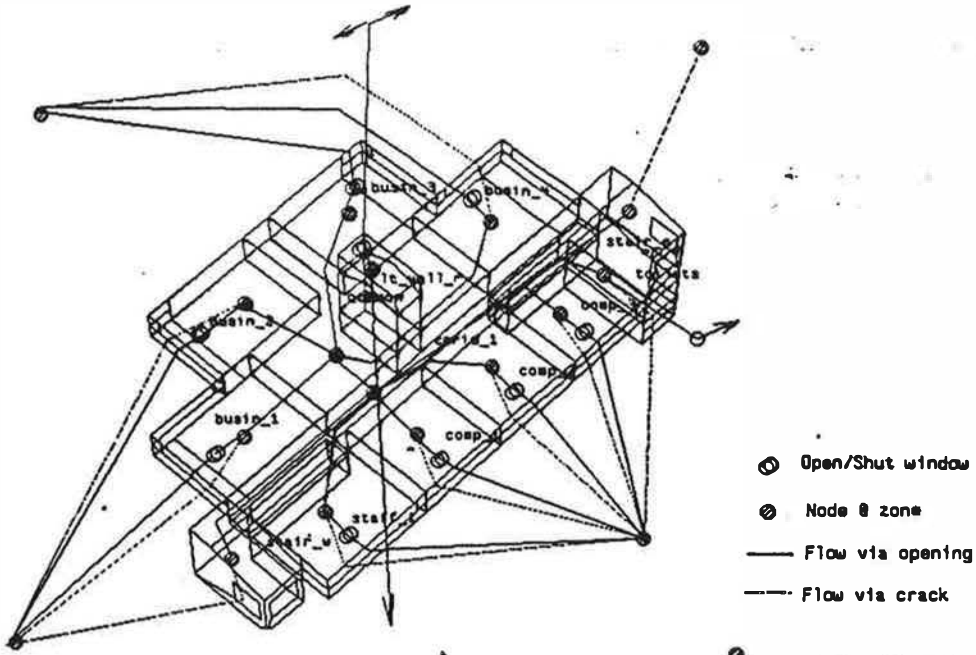
- Requirements for a soluble network: at least one of the node pressures must be known; all nodes must be linked, through some path, to a known pressure node. (cf CFD where solubility is dependent on experience.)

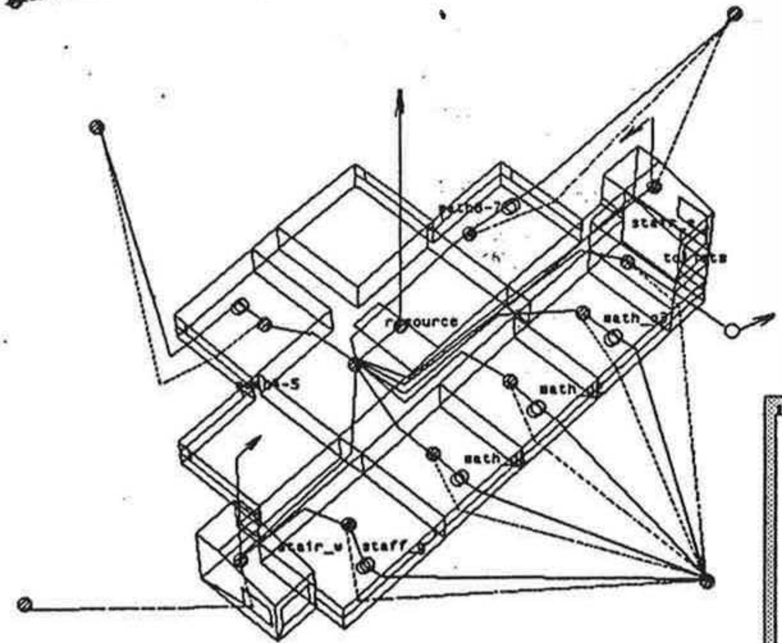
ESP-r flow components:

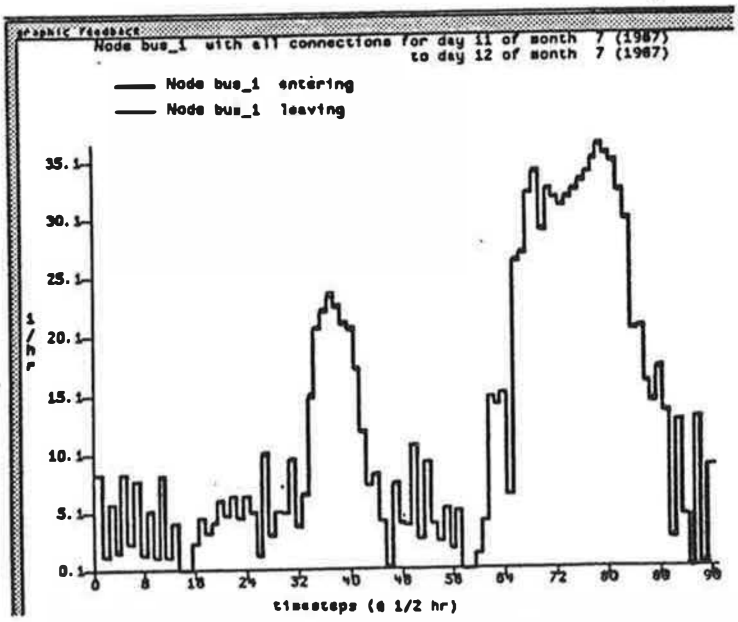
Component	Formula
Power law volume flow resistance	$\dot{m} = \rho\, a \Delta P^{b}$
Power law mass flow resistance	$\dot{m} = a \Delta P^{b}$
Power law mass flow resistance	$\dot{m} = a\sqrt{\rho}\, \Delta P^{b}$
Quadratic law volume flow resistance	$\Delta P = a\dot{m}/\rho + b(\dot{m}/\rho)^2$
Quadratic law mass flow resistance	$\Delta P = a\,\dot{m} + b\,\dot{m}^2$
Constant volume flow rate	$\dot{m} = \rho\, a$
Constant mass flow rate	$\dot{m} = a$
Common orifice flow	$\dot{m} = \rho\, f(Cd\, A\, \rho\, \Delta P)$
Laminar pipe flow	$\dot{m} = \rho\, f(L\, R\, \mu\, \Delta P)$
Specific air flow opening	$\dot{m} = \rho\, f(A\, \Delta P)$
Specific air flow crack	$\dot{m} = \rho\, f(W\, L\, \Delta P)$
Specific air flow door	$\dot{m} = \rho\, f(W\, H\, Hr\, Cd\, \Delta P)$
General flow conduit (duct or pipe)	$\dot{m} = \rho\, f(D_h\, A\, L\, k\, \Sigma C_i \nu\, \Delta P)$
Flow conduit ending in converging 3-leg junction	$\dot{m} = \rho\, f(D_h\, A\, L\, k\, \Sigma C_i \nu\, \Delta P)$ $C_{c,p} = f\,(a_{0\text{–}5}, \frac{q_p}{q_c}, \frac{q_{p'}}{q_c})$
Flow conduit starting in diverging 3-leg junction	$\dot{m} = \rho\, f(D_h\, A\, L\, k\, \Sigma C_i \nu\, \Delta P)$ $C_{c,p} = f\,(a_{0\text{–}5}, \frac{q_p}{q_c}, \frac{q_{p'}}{q_c})$
Flow conduit ending in converging 4-leg junction	$\dot{m} = \rho\, f(D_h\, A\, L\, k\, \Sigma C_i \nu\, \Delta P)$ $C_{c,p} = f\,(a_{0\text{–}9}, \frac{q_p}{q_c}, \frac{q_{p'}}{q_c}, \frac{q_{p''}}{q_c})$
Flow conduit starting in diverging 4-leg junction	$\dot{m} = \rho\, f(D_h\, A\, L\, k\, \Sigma C_i \nu\, \Delta P)$ $C_{c,p} = f\,(a_{0\text{–}9}, \frac{q_p}{q_c}, \frac{q_{p'}}{q_c}, \frac{q_{p''}}{q_c})$
General flow inducer (pump or fan)	$\Delta P = \sum_{i=0}^{3} a_i (\dot{m}/\rho)^i$ $\dot{q}_{min} \leq \dot{m}/\rho \leq \dot{q}_{max}$
General flow corrector	$\dot{m} = \rho\, f\,(\rho_0\, \Delta P_0\, k_{vs}\, k_{v0}\, k_{vr}\, H/H_{100})$ $H/H_{100} = f\,(day\ time\ S_l\, H_l\, S_u\, H_u)$
Flow corrector with polynomial flow resistance	$\dot{m} = f\,(\rho A\, \Delta P\, C)$ $C = \sum_{i=0}^{3} a_i (H/H_{100})^i$ $H/H_{100} = f\,(day\ time\ S_l\, H_l\, S_u\, H_u)$
Ideal (frictionless) open/shut flow controller	$\dot{m} = 0$ or $\Delta P = 0$


roof
roof
roof
roof
cafe
main
square
east
west
south
roof

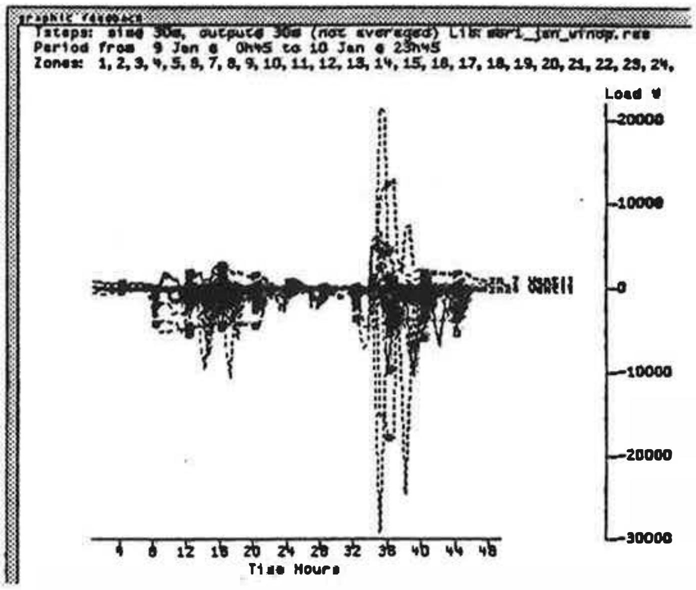

dining area
dayroom
open
window
windspeed
control
temperature
control
west
window
cracks
dayroom







July air flow rates in business classroom 1 based on window opening if room temperature > 23 deg C.



Energy content of infiltration (exterior) and ventilation (inter-zone) flows when window control is operational.

Pressure Coefficients:

- Pressure coefficient represents the mapping from free stream wind velocity to building surface.
- Mapping done as a function of vertical velocity profile selected from several in-built options:
  - Direct user input.
  - Power law wind profile.
  - Logarithmic wind profile.
  - LBL wind profile.
- ESP-r offers a database of pressure coefficient sets.

Buoyancy:

- Nodes have a reference height which defines the mean height of the associated building space or plant component.

- Inlet and outlet of a connecting component may be at different heights relative to each other and relative to these nodes.

- Pressure difference is given by:

$$\Delta P_i = (p_1 + \rho V_1{}^2/2) - (p_2 + \rho V_2{}^2/2) + \rho g(z_1 - z_2) \quad (Pa)$$

where ΔP_i is sum of friction and dynamic losses, p_1, p_2 are entry and exit static pressures, V_1, V_2 are entry and exit velocities, ρ is density of component fluid, g is acceleration of gravity and z_1, z_2 are entry and exit elevations.

- In PHOENICS buoyancy is explicitly requested as a momentum equation source term.

Network Solution:

- Flow component, i, relates mass flow rate, m_i, through component to pressure drop, ΔP_i, across it.
- Solution requires iterative processing of set of simultaneous non-linear equations when subjected to set of boundary conditions.
- Internal nodes are assigned arbitrary pressure to enable calculation of connection flow from the appropriate connection equation.
- Internal node mass flow residuals are computed.
- Nodal pressures are iteratively corrected and mass balance at each internal node re-evaluated until convergence criterion is met.

Iterative Procedure:

New estimate of node pressure vector, $\mathbf{P}^*$, is computed from the current pressure field, $\mathbf{P}$, via:

$$\mathbf{P}^* = \mathbf{P} - \mathbf{C} \qquad (1)$$

where pressure correction vector, $\mathbf{C}$, is determined on basis of simultaneous solution of Jacobian matrix representing nodal pressure corrections in terms of branch flow partial derivatives:

$$\mathbf{C} = \mathbf{R}\,\mathbf{J}^{-1}$$

where $\mathbf{R}$ is vector of nodal mass flow residuals and $\mathbf{J}^{-1}$ is inverse of square Jacobian matrix whose diagonal elements are:

$$J_{n,n} = \sum_{k=1}^{K_{n,n}} \left(\frac{\partial \dot{m}}{\partial \Delta P} \right)_k \quad [kg/s\,Pa]$$

where $K_{n,n}$ is number of connections linked to node n and ΔP_k is pressure difference across the kth link. Off-diagonal elements of $\mathbf{J}$ given by:

$$J_{n,m} = \sum_{k=1}^{K_{n,m}} - \left(\frac{\partial \dot{m}}{\partial \Delta P} \right)_k \quad [kg/s\,Pa]$$

where $K_{n,m}$ is number of connections between node n and node m.

In PHOENICS, eq 1 is also applied for U, θ and τ. Alternative direction interation (ADI) method used to simplify Jacobian which is solved by TDMA method.

Steffensen Device:

- For case of oscillating pressure corrections ESP-r assumes that each successive pressure correction is constant ratio of previous correction so that it is possible to extrapolate corrections to an assumed solution:

$$P_i^* = P_i - C_i / (1-r) \quad (Pa)$$

where r is ratio of C_i for current iteration to its value in previous iteration. $1/(1-r)$ is a relaxation factor.

- Extrapolated value of node pressure may be used in next iteration. If it is, then r is not evaluated for that node in the following iteration i.e. r only evaluated with unrelaxed pressure correction values.

- Method gives a variable and node-dependent relaxation factor.

Matrix Processing:

- Solution of matrix equation **J C = R** based on LU decomposition with implicit pivoting (Crout's method with partial pivoting).

- Matrix **J** decomposed to a lower triangular matrix **L** and an upper triangular matrix **U**, such that **L U = J**. Decomposition is used to solve the linear set:

$$\mathbf{J\,C} = (\mathbf{L\,U})\,\mathbf{C} = \mathbf{L}\,(\mathbf{U\,C}) = \mathbf{R}$$

  by first solving for the vector **Y** such that **L Y = R** and then solving **U C = Y**.

- Advantage is that both substitutions are trivial. Pivoting used to make method numerically stable.

Convergence:

- Assumed when either:
  - Largest percentage residual flow error (ERRMAX) falls below user-specified value:

$$\left[\frac{|\sum \dot{m}_k|}{\sum|\dot{m}_k|}\right]_{max} \leq \frac{\text{ERRMAX}}{100} \quad (-)$$

  - Largest absolute residual flow error (FLOMAX) falls below user-specified value:

$$(|\sum \dot{m}_k|)_{max} \leq \text{FLOMAX} \quad (kg/s)$$

  PHOENICS criteria.

- Occasional instances of slow convergence with oscillating pressure corrections on successive iterations is avoided through use of a "Steffensen iteration" device.

ESP-r Combined Heat and Mass Flow:

- Energy conservation equations, corresponding to plant-side finite volumes (discretised components and distribution network) established on basis of latest values of building-side state variables and plant component/ network mass flows.
- Plant matrix equation solved by sparse matrix technique taking into account control action. Since plant time constants usually less than building, plant equations may be integrated more frequently.
- Energy conservation equations corresponding to finite volumes representing building-side discretised constructions, surfaces and air volumes then established on basis of latest values of plant flux inputs and building-side air flows.
- Building matrix equation solved by customised matrix inversion technique which employs partitioning and ordering to ensure that only non-zero matrix entries are processed. Control system characteristics fully integrated within the solution process.
- Whole-system, fluid flow equations solved, iteratively, utilising newly established building and plant-side state variables to estimate buoyancy effects.
- Time-step control may be activated to prevent evolution of time in cases where newly computed state-variables differ markedly from latest values assumed when matrix equations were established.
- Finally, the simulation clock is incremented and process repeats.
- Overall procedure equivalent to modular simultaneous with mixed-frequency, variable time-stepping permitting possibility of sub-matrix exclusion depending on the problem.

Advantages of network approach:

- Number of nodes - ~500 in a moderately building network - is low and so the additional CPU burden is acceptable.

- Strong relationship between heat and air flow networks. Information demands of the heat and air flow conservation equations can be directly satisfied.

- Technique can be applied to combined multi-zone buildings and multi-component, multi-network plant systems.

CFD techniques:

Finite Difference	Finite Volume	Finite Element
• Grid of discrete points	• Mesh of discrete control volumes	• Mesh of discrete elements
• Computational molecule around point	• Computational volume around point	• Nodal points on element boundary
• One equation for each point	• One equation for each point	• Matrix of incomplete equations
• Computational molecules do overlap	• Control volumes are disjoint	• Finite elements are disjoint
• Approximation in each point zero	• Integral over whole domain zero	• Integral over whole domain minimal
• Local approximation	• Local approximation	• Global approximation
• Iterative solution procedures	• Iterative solution procedures	• Direct solution methods ("Gauss")
• Orthogonal grids as a bottleneck	• Orthogonal grids as bottleneck	• Arbitrary complex geometries
• Complicated differential equations	• Complicated differential equations	• Variational principle as bottleneck
• Structured grid	• Structured grid	• Unstructured grid

Problems:

- Network approach:
  - Limited application scope of models (by type and resolution).
  - Flow field constrained to steady flow (possibly bi-directional) of incompressible fluid.

- CFD approach:
  - Convergence difficulties (well-posed problems require a specialist).
  - Computing constraints (especially with 3D, non-steady problems).
  - Estimation of boundary conditions.
  - Low Re turbulence modelling (Reynolds stress modelling very CPU intensive).
  - Building applications restricted to steady-state.

Network method future work:

- Additional component models of extended scope.
- Knowledge based initialisation.
- Improved numerics.
- Eliminate prefectly mixed zone concept.
- Validation.

CFD research topics:

- Characteristics of turbulence modelling:
  - Eddy viscosity models.
  - Reynolds stress models.
  - Wall function performance.

- Addtional body-force manipulation:
  - Centrifugal and Corioli force.
  - Bouyancy force.

- Improved numerical schemes:
  - False diffusion effect.
  - Incorporation of experimental data in numerical scheme.

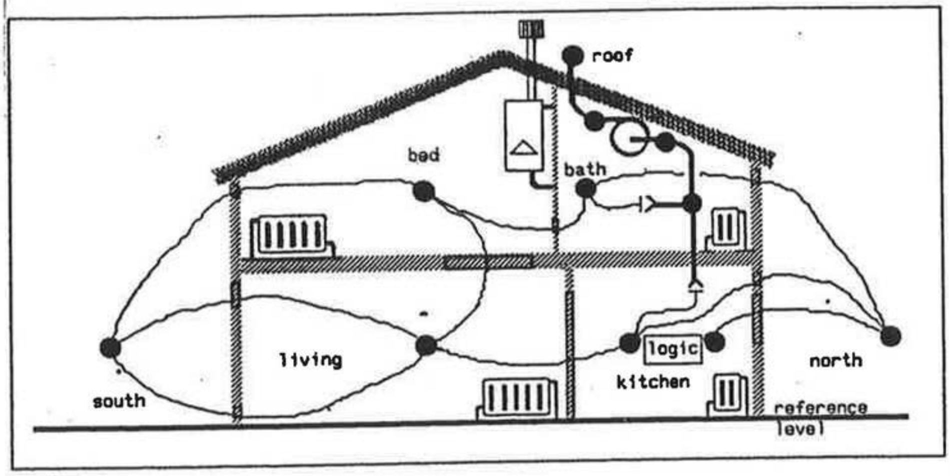
CFD/ Network Method conflation:

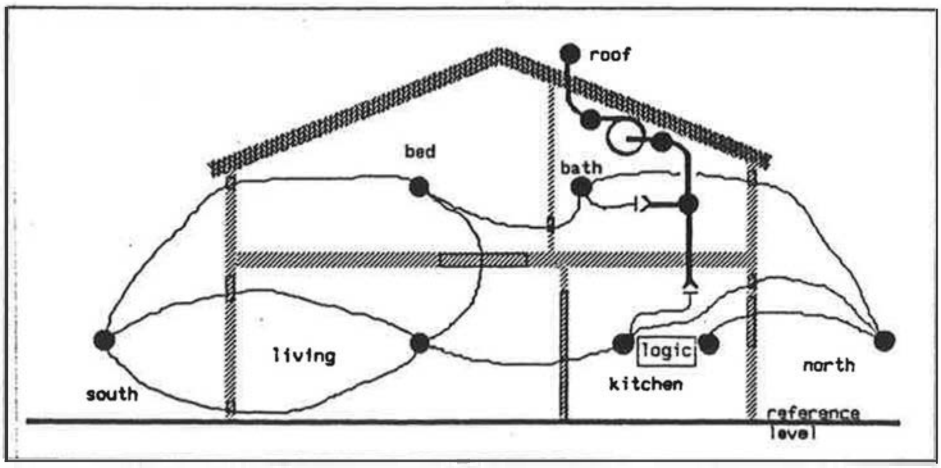
- Domain calculation by CFD followed by transient simulation by CTD on basis of small time-step and coarse spatial division.

- Multiple gridding schemes and numerical conflation required.

- Macro-system simulation to indicate "correct" turbulence model coefficients, wall-functions and boundary conditions.

- Conflation required for "integrated building design systems" of the future.


roof
bed
bath
living
logic
north
south
kitchen
reference
level


roof
bed
bath
living
logic
north
south
kitchen
reference
level

THE USE AND APPLIABILITY OF AIRFLOW MODELS

S J Irving; Oscar Faber Applied Research Ltd., St Albans, Herts, AL1 3UT

1. Introduction

Thermal and airflow models can be applied in a number of different ways during the process of designing and operating a building. In assessing the suitability of different modelling techniques, it is important that the various approaches to analysing building performance are set into the proper context of applications needs. The following sections identify a number of applications areas where models are used, and seeks to identify key factors which influence the selection of model complexity.

Air movement effects building thermal performance in a number of inter-related ways. As a result of increasing thermal insulation standards, ventilation is becoming the dominant factor in the energy balance of a building. Air movement around and within buildings also effects thermal performance by influencing the rate of heat transfer from the building fabric to the air. Air movement is also a primary mechanism for conditioning the thermal environment of a space, either directly by the mechanical injection of conditioned air into the space, or indirectly by natural convection flows generated by the heated (or cooled) surfaces such as radiators or chilled ceilings. Air movement is also the primary mechanism for pollutant transport within buildings.

2. Applications Areas for Models

From the above, it can be seen that air movement has a profound effect on energy, comfort and air quality. It is therefore essential that building designers have a full understanding of how air movement interacts with the overall building thermal and pollutant transport processes; the following paragraphs give more detail on each of these interactions, and highlights the need for better combined heat transport and air flow modelling.

2.1 HVAC System Design.

Traditionally, the main application for thermal models has been to help size HVAC system components. This will include

- room units (radiators, air flow terminals)
- central plant (boilers, chillers)

The techniques used for such exercises vary from simple steady state methods, right through to full dynamic simulation. It must be recognised that sizing calculations are to do with providing adequate plant capacity to meet a particular design condition, which is associated with a degree of acceptable risk. Therefore simplified assessments of ventilation rate are probably adequate in actively conditioned buildings, where mechanical ventilation will probably pressurise the building.

The main need for enhanced ventilation algorithms comes in the context of designing more passive systems such as using atria or sunspaces to pre-heat ventilation air in winter. There is also increasing interest in using natural ventilation as a means of controlling summer overheating. These examples relate more to the design of the building envelope than to the HVAC systems. The design approach demands that the required air change rates are delivered at the right time to the right place. This involves an interaction between envelope leakage distribution, wind pressures and temperature variations. Typical thermal models will assume the air change rates, and predict the resultant temperature distribution. Multi-zone ventilation models will assume a temperature distribution and predict the air change rates. The problem is that of course these parameters are not independent, a problem which is further exacerbated by the thermal inertia of the building fabric. Combined modelling of air flow and heat transport can address these interactions directly.

2.2 Ventilation Energy Losses

By definition, analysis of energy use is primarily concerned with long time sequences, usually a full year, or at least a heating or cooling season. There are a range of methods which can be applied to predict energy consumption. These range from steady state models using period average data, right through to simulations of building, plant and controls at time steps of minutes.

In well insulated buildings, the ventilation energy loss can often represent up to 50% of the total heating energy demand. It is therefore self evident that the estimation of the ventilation energy loss be as precise as possible. The relationship between natural ventilation rate and inside-outside temperature difference is non-linear, and so the seasonal energy loss is not directly proportional to the product of average flow rate and average temperature difference. Because the flow rate is itself a function of the temperature difference, and the internal temperature is influenced by the rate of room energy loss, there is a strong interaction between the two parameters, especially at low wind speeds. This interaction is complicated by the user influence, since the permeability of the external envelope may also be temperature dependant, as occupants open windows in warmer weather.

2.3 Stratification

In most thermal models there is an implicit assumption that the air in any one zone is fully mixed, i.e. there are no variations in air temperature within the zone. Such assumptions are reasonable in many situations, but there is an increasing tendency to design to achieve stratification in the space. Two examples are displacement ventilation and atrium design where the intent is to capture the heat at the top of the space to promote stack ventilation in the summer and avoid excessive temperatures in the occupied zone. Some atria are also used to capture and pre-heat ventilation air in winter.

In most atria, the pattern of air movement is strongly influenced by, if not entirely due, to natural convection, and so some approaches to modelling these convection flows is necessary for credible design assessments. Solar radiation falling on the walls or floor of the atrium will generate warm buoyant plumes. In winter and mid season, areas of the atrium glazing may be cold, resulting in down draughts and recirculation. By definition, atria are coupled to the rest of the building, and so the distribution of temperature with height up the atrium will influence the conduction and ventilation energy exchanges with different floors of the main building. It is clear that a full understanding of all the energy flow paths is needed to achieve an effective design. This is one of the most challenging areas for thermal modelling.

2.4 Pollutant Transport

In the past, the design emphasis has been on thermal comfort, but with the advent of new materials and a concern to achieve energy efficient ventilation, increasing attention is being given to air quality control. Air movement is the main means for pollutant transport within buildings. The previous sections have indicated how air movement is affected by the thermal environment. However the thermal environment also has a significant effect on other aspects of IAQ control. Pollutant transport is concerned not just with air movement, but also with pollutant sources and sinks. The strengths of such source and sink terms can be temperature and humidity dependent.

3. Approaches to Infiltration/Ventilation Modelling

The following sections detail different approaches to modelling air flow in buildings, either as a stand alone program or integrated within a thermal model. These various methods will be reviewed, and their relative strengths and weaknesses assessed.

3.1 Fixed air change rates

These are the methods to be found in the CIBSE Guide and ASHRAE Fundamentals. There are two methods outlined in the CIBSE Guide. In the first, infiltration is calculated by graphical means using local wind speed, building height and building quality data. A basic infiltration rate is calculated based on design wind speed. Adjustments are then made to individual room rates according to the window distribution and room level. The second approach makes use of a table of expected values for buildings of typical construction. This is the most commonly used method of estimating infiltration/ventilation, and is the reference point against which other methods will be assessed.

3.2 Scheduled rate

This approach is an enhancement of 3.1, where the flowrates are varied with time based on assumptions about day time window opening, varying ventilation needs etc.

3.3 Regression Techniques

This method is based on the results of statistical fits to long term time series data of infiltration rate measurements and associated climatic data. In its basic form air infiltration is expressed as a linear function of wind and temperature.

The main value of this approach is in the extrapolation of the results beyond a measured period. Typically, hourly rates of air infiltration are continuously measured over a period of a few days. Appropriate regression coefficients are then evaluated and the performance of the air infiltration equation is verified over a further short measurement period. The regression equation may then be used to estimate air infiltration performance of the building over a wider set of climatic conditions.

The main disadvantage of this method is that the calculated regression coefficients are unique to the building since they reflect not only the air tightness performance of the building but also its orientation with respect to adjacent obstacles. It is therefore not possible to transfer the coefficients to other buildings. Although representative values of regression coefficients have been published for design purposes, they can be unreliable.

3.4 Mass balance - Single zone

This method is used to calculate the infiltration into a single space, and is therefore best suited for whole building infiltration rate assessments. A single zone network approach offers many advantages which include – comparative ease of calculation, the addition of any number of flow paths, the inclusion of any combination of wind, stack and mechanically induced pressures, and the ability to identify the flow direction and the magnitude of the flow rate through each of the defined openings.

The principal disadvantage of a single zone approach is its unsuitability for multi zone buildings. Although it is possible to subdivide the building into an array of single zones assuming that they are isolated internally, this approach does not consider the interaction between zones. Such interactions are very important for natural ventilation strategies, pollutant transport modelling, etc.

3.5 Mass Balance – Multi zone

If internal partitioning presents an impedance to the movement of air, then for most purposes a multi zone or multi cell approach is needed. Multi zone models have also been applied to predicting patterns of air movement in large spaces such as atria. A number of such programs have been developed for the purpose of incorporation into combined thermal and air transport models.

Multi zone models are complex programs in their own right, and very many different programs have been developed in different research centres around the world. Through such activities as the COMIS group, there have been moves to bring the best of such models into a common system. A major focus of current research activity is how such models can be coupled into detailed thermal models. Three methods have been proposed as described below.

3.5.1 Sequential Coupling

This is the simplest coupling method, and involves running the two programs independently. Firstly, the multi zone air flow model is run, with assumed values of zone temperatures. The air flow model is solved for each of the weather conditions (temperature, wind speed and direction) for the period of interest of the thermal simulation. This generates a schedule of zone air change rates as a function of time. These are then used in the thermal simulation as described in 3.2, the only difference being the schedule of flows has been calculated rather than assumed. The model user should confirm that the values of zone temperature assumed for the multi zone air flow model were reasonable by checking against the zone temperatures predicted by the thermal simulation. If there are significant differences, then the loop should be repeated.

3.5.2 'Ping–Pong'

The 'ping–pong method involves running the multi zone model and the thermal simulation concurrently, but independently. The thermal program will just predict the zone temperatures based on the current air exchange rates, climate etc. These temperatures are then passed onto the air flow model which will calculate a revised set of air exchanges. These will be passed back to the thermal simulation to be used for the next simulation time step. Ideally a supervisor should generate a time loop that alternately starts a one–step simulation run with the two codes involved. Each model solves its own problem with its own method. At each time step the supervisor must decide whether the solution converges, by comparing the values from one time step with the values from the previous time step and decide if the solution is valid. The advantage of this method is that if a generic coupling environment exists, various codes can be coupled without having to rewrite specific subroutines.

One problem with this technique is that the calculated air flows are assumed to be constant for the simulation time step. When there are large openings, buoyancy driven exchanges can result in very large heat flows. If the simulation time step is too large, the constant heat flow may significantly effect the temperature distribution, so the air flows may even reverse. In reality the temperatures will reduce as the temperatures equalise, but as the programs are run independently, this effect cannot be handled effectively.

3.5.3 Full Integration

This method involves the explicit solution of the air flow equations as part of the overall simulation. Thermal simulation programs set up a series of equations defining the energy balance of each space. These are solved using a variety of solution methods. Full integration of the air flow requires that the equations that describing the zonal air flows are incorporated within the matrix covering all the other heat transfer processes, and solved simultaneously.

3.6 Computational fluid dynamics (CFD)

More recently, attempts have been made to incorporate computational fluid dynamics codes (CFD) into thermal models.

When attempting to model the air flow patterns in large spaces, the application of multi zone techniques can present a few drawbacks. Multi zone assume that all the air in each zone is fully mixed. This requires that stratified spaces be subdivided into a number of coupled zones. The basis for such subdivision requires considerable expertise and experience.

One of the problems of CFD is to properly describe the boundary conditions of surface temperature, surface heat fluxes etc. This is particularly important where natural convection is dominant. Thermal models can be used to predict these values, and so the thermal and CFD models can be run sequentially, much as described in section 3 5.2. However, because of the very large run times of the CFD method, the models are only likely to be used in tandem to investigate a single 'snap shot' in time rather than any extended period. Another problem is that in the main, thermal models produce surface average temperatures for input to the CFD code. It is local 'hot spots' from solar patches falling onto a floor or wall which generate the convection currents, and the strength of this effect may be minimised by surface averaging.

4. Relevant Parameters

In assessing the applicability of various algorithms, the requirements and availability of the data to drive them is a very important issue. The following paragraphs detail some of the additional data needs over and above those commonly used in thermal models. The review will highlight these data needs and their significance, and give some pointers to available datasets. It is clear that there is little value in having a very sophisticated modelling technique if there is little or no practical advice on appropriate data sets to plug into those models.

4.1 Climate

Infiltration is often wind dominated, particularly in low rise buildings and the temperate U.K. climate. Most thermal models only make use of wind speed to vary the external film coefficient for convective heat transfer. However, wind pressure coefficients are also influenced by wind direction, and so this will require extensive data sets on C_p values for varying angles of attack and different building geometries.

4.2 Terrain

The local terrain will have an influence on the wind profile and hence distribution of wind pressure over the building facades. This requires methods of varying the C_p data for terrain roughness and any local obstructions.

4.3 Leakage Data

The air flow through cracks and purpose provided openings is a function of the size and nature of that opening. This will require extensive datasets on leakage characteristics of typical building components.

4.4 HVAC Components

When mechanical ventilation systems are used, then the pressure flow characteristics of the system need to be modelled to determine the operating point. This in turn requires information on fan characteristics, and the pressure loss characteristics of ductwork components (bends, tees, terminal devices etc).

4.5 Occupancy Effects

The influence of the occupant is especially important in the context of natural ventilation strategies – varying window opening etc.

ARUP ZONAL MODELS

I am only concerned with the application of airflow models to building design. The majority of modelling is not carried out with that specific objective, it is used to analyse proposals. That is to carry out a number of "what if's?" I do not say that to do that is wrong, what I do say is that for such an approach to be cost effective it is necessary to have a very good idea of what the correct design solution is before carrying out the analysis. A simple, easily used and understood model is invaluable in identifying the important parameters and may indeed be all that is necessary to produce a successful design. A good example of this case can be taken from the design of the Arup Associates building, Gateway 2. The objective was to be confident that covering over a lightwell would not prevent natural ventilation of the surrounding office space. A simple zonal model that represented the main features of the ventilation system demonstrated that the building could work and highlighted the need to consider restricting window opening at the various floor levels to ensure balanced flows through each space. (See M J Holmes. Design for Ventilation 6th AIC Conference, 1985, for more information).

A more complex, traditional, multizone, ventilation model was used to analyse potential flow patterns within Gateway 2 and later for a proposed design of a bus station. In the latter case the problem was to prevent exhaust fumes from entering the passenger waiting areas. The problem was complicated in that it was necessary to run air-conditioning systems in the buses, so therefore the engines could not be turned off. An examination of potential boundary conditions, including ventilation strategies for surrounding spaces, and climatic variations showed that a minimum of 96 runs would be necessary to examine the effect of features the designers thought to be important. In practice a more limited number of options were examined. This work is reported in a paper by Holmes and Salusbury to the 9th AIVC Conference (Ventilation Design for a Bus Station). Once again it is possible that a simpler approach would have highlighted significant features related to the performance of the building. This is a problem that current education trends are not solving - students appear to be encouraged to use sophisticated software instead of thinking around the problem.

A further example of the application of simple zonal models is the analysis presented by Holmes and Cousins (Optimisation of the Thermal and Ventilation Performance of Naturally Ventilation Building Facades, 13th AIVC Conference) which was used to identify the design limits for buildings using a thermal flue to drive natural ventilation. The paper does not answer all questions but shows how a simple approach can once be used to give guidance for a more detailed analysis. Once again a conventional zonal model was used to examine a few "what if's".

The Arup multi-cell model (VENT) is implemented via the Oasys Ltd system. BEANS provides an integrated data base from which most Arup building services software can take its data. This ensures an element of Quality Assurance in that there is no question that the same building - or building element - is being used by each program within the system. The most used Arup airflow model is also run via BEANS, the simple zonal model within the ROOM comfort prediction program. This is a one or two cell model linked to a dynamic thermal model and Fanger comfort prediction algorithms with the objective of producing comfort contours within the occupied zone (Connor & Holmes, ROOM, a program to predict comfort at any point in a space. 1991 CIBSE Conference).

M J Holmes

22nd October 1992

USE OF FLOW3D

Norman Rhodes
Mott MacDonald
20-26 Wellesley Road
Croydon CR9 2UL

Overview of presentation

At Mott MacDonald we utilise a variety of CFD techniques to analyse and evaluate flow situations relevant to our design activities. These include zone models for overall building performance evaluation; one and two-dimensional codes based on the Method of Characteristics, for predicting aerodynamic and thermodynamic effects in tunnels; and three-dimensional codes for prediction of a variety of situations.

The FLOW3D code has been used by our CFD group for about three years. The general application areas include 3D aerodynamics of both bluff and streamlined bodies such as offshore platforms and trains, gas combustion in landfill site flare stacks, power-station steam condensers, ventilation and smoke movement.

The code provides a comprehensive choice of model-building tools. Grid generation is very straightforward whether the command-language-based or the interactive procedure is used. Formulation of the modelled equations and boundary conditions is obtained through a further command-language file. The commands themselves have obvious meanings and sensible default values are specified. For the more experienced modeller there are a number of differencing schemes and turbulence models built into FLOW3D.

Typical applications which illustrate our use of the code include the evaluation of smoke control strategies in a railway station and smoke ingress in an offshore platform.

The station smoke control study involved setting up the main features of the geometry, including the platform, platform screen doors, passenger concourse and vents in the roof which were open to atmosphere. A train fire scenario was specified and modelled, and smoke and temperature concentration plots used to evaluate the smoke behaviour. The model was then extended to include smoke curtains above the track at 20 m intervals and mechanical ventilation of the reservoir formed by the curtains. An identical fire scenario was modelled and similar plots of smoke concentration and temperature enabled a comparison of the two schemes.

Modelling the smoke ingress into an offshore platform requires more elaborate steps to determine the boundary conditions. Mott MacDonald's CFD group have undertaken substantial calculations of the external flow around platforms to predict the pressure distribution arising from a particular wind direction and strength and smoke concentrations from a given fire scenario.

These data have then provided the inputs to very detailed flow models of the internal features and HVAC system and its control procedure.

The normal output of FLOW3D has been enhanced by the use of animation techniques. These have been particularly useful in explaining the physical behaviour of smoke transients and illustrating the consequences of particular actions such as switching on emergency ventilation.

BSRIA | AIVC/BEPAC WORKSHOP | 1/8

Building Airflow Simulation
Usability of CFD & Zonal Models in Practice
22 October, 1992
British Gas Midlands Research Station, Solihull

FLOVENT, a special purpose CFD program for environmental and building services engineers

DR FARSHAD ALAMDARI
Microclimate Centre, BSRIA

BSRIA | BACKGROUND | 2/8

FLOVENT is a special-purpose Computational Fluid Dynamics (CFD) program which has been conceived, specified and developed by collaboration between **BSRIA** and **Flomerics** Limited.

FLOVENT is designed specifically for environmental and building services engineers. It is a computer-based predictive tool that enables designers to analyse the airflow and related phenomena such as smoke or airborne contaminant movement in and around buildings.

FLOVENT is a finite-volume "field" model that solves, numerically, the governing flow equations for a finite set of control volumes, formed by discretization of the flow domain via a computational grid.

BSRIA

FLOVENT FEATURES

GEOMETRICAL ASPECTS

- Architectural features of buildings and furniture can be represented by using combinations of cuboid and triangular blocks.
- Building spaces can be discretized by using rectangular cells (or control volumes) and adopting simple, three-dimensional Cartesian coordinates.

BOUNDARY CONDITIONS

- Built-in boundary conditions for common HVAC equipment, such as fan-coil units, induction units, etc.
- External surfaces may be defined by internal surface temperatures, or external convective heat transfer coefficient and the surface conductivities, or fabric 'U' values.
- Time-dependent interaction between the building structure, equipment and internal conditions can be modelled.

BSRIA

FLOVENT FEATURES

THERMOFLUID PARAMETERS

- **Turbulent flows** - in turbulent flows, the closure of the governing equations is achieved by the adoption of of K-ε turbulence model.
- **Buoyancy effects** - buoyancy effects may be introduced either by the use of the so-called Boussinesq approximation (for constant air density) or by calculating the density of the air in each control volume (ideal gas law).
- **Conjugate heat transfer** - the temperature distribution in building fabric, internal features , etc can be solved simultaneously with air temperature.
- **Radiation** - the mean radiant temperature at each computational grid point can be calculated as a function of the surface areas, shapes, surface temperatures and emissivities of the enclosing elements *seen* from that point.

BSRIA

APPLICATIONS

5/8

In the context of the built environment, **FLOVENT** can be used as an efficient and powerful tool for:

- ❑ **THERMAL COMFORT ASSESSMENTS**
  This is an important aspect of air conditioning performance analysis and preventing problems, such as discomfort and 'sick building syndrome', which are mainly associated with poor air movement and temperature distribution within the space.

- ❑ **ENVIRONMENTAL SAFETY ASSESSMENTS**
  The effectiveness and performance of the ventilation system in removing the smoke as a result of fire or other contaminants is important in the safe design of buildings.

BSRIA

APPLICATIONS

6/8

- ❑ ***Atria*** - for example, in environmental performance analysis under various summer and winter climatic conditions for over-heating and cold downdraughts respectively.

- ❑ ***Art galleries and museums*** - such as preventing the risk of damaging expensive displays as the result of air temperature and humidity.

- ❑ ***Shopping centres, tunnels, airport terminals, etc.*** - for instance predicting the smoke movement as a result of fire.

- ❑ ***Air conditioned offices*** - such as providing satisfactory environmental thermal comfort and air distribution.

- ❑ ***Clean rooms and operating theatres*** - particularly in predicting the flow pattern and contaminant movement.

- ❑ ***Industrial buildings*** - for example, in performance assessment of air distribution and ventilation systems in removing airborne dust, etc.

BSRIA

A CASE STUDY

MICROCLIMATE PERFORMANCE OF AN OPEN ATRIUM OFFICE BUILDING
(source: F Alamdari et al, IMechE seminar "CFD - tool or toy?", Nov. 1991)

The design of the headquarters for Hoare Lea and Partners, consulting engineering, coincided with the greater emphasis on energy efficient and environmentally friendly methods for engineering buildings. A naturally-ventilated atrium design was therefore adopted consisting of two two-storey buildings connected by an entrance hall, each with a central open atrium.

FLOVENT has been used to examine the environmental performance of these offices under various summer and winter climatic conditions from computed distributions of air velocities and temperatures.

BSRIA

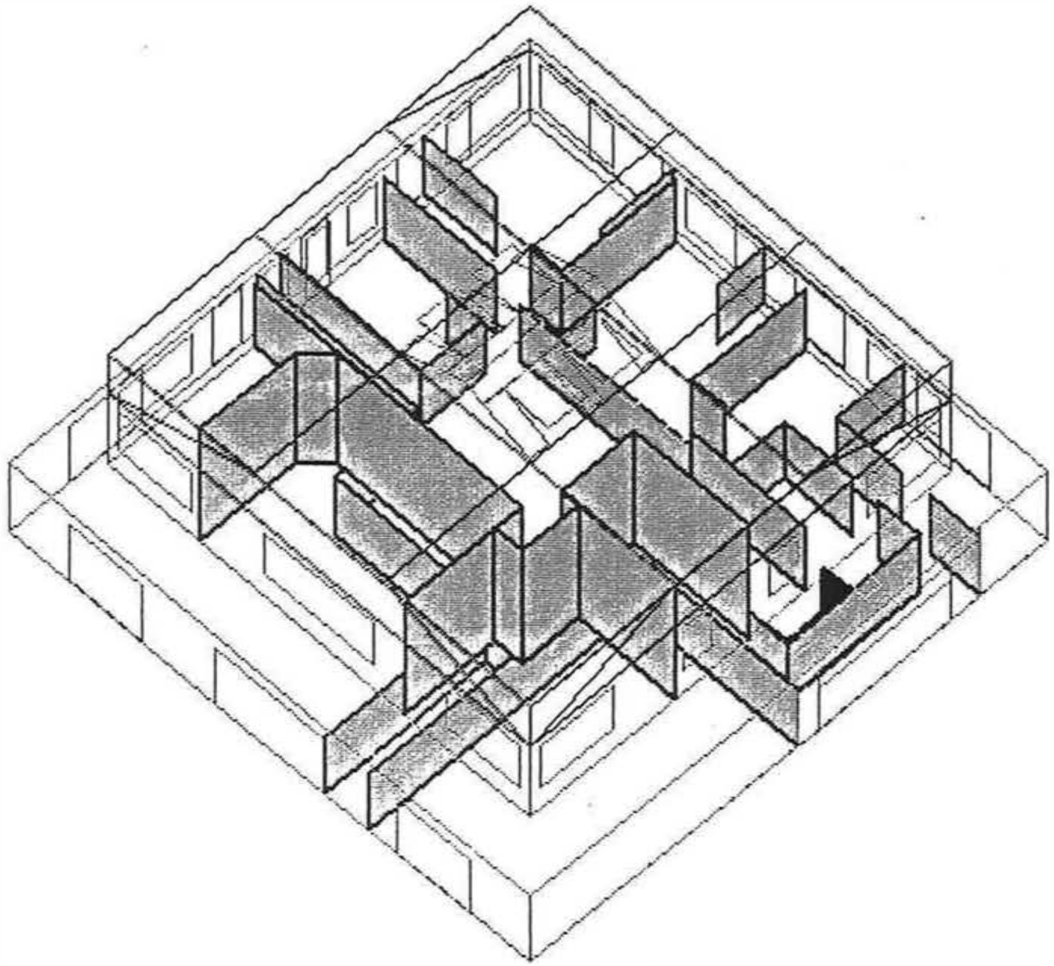
A CASE STUDY

VENTILATION PERFORMANCE USING DISPLACEMENT FLOW IN OFFICES
(source: K M Bennett et al, BSRIA report 77220/1, 1992)

Displacement ventilation is a method that intends to provide conditioned air to indoor environments with the view to improving air quality whilst reducing energy usage.

A study has been carried out at **BSRIA** in which the operational performance of several air conditioned offices incorporating displacement ventilation has been investigated.

FLOVENT has been used for a comparative performance assessment of one of the building sites. The numerical study was then extended to study the criticality of various factors, e.g. grille air flow characteristics, location of obstructions and floor free area, and distribution of internal heat sources.



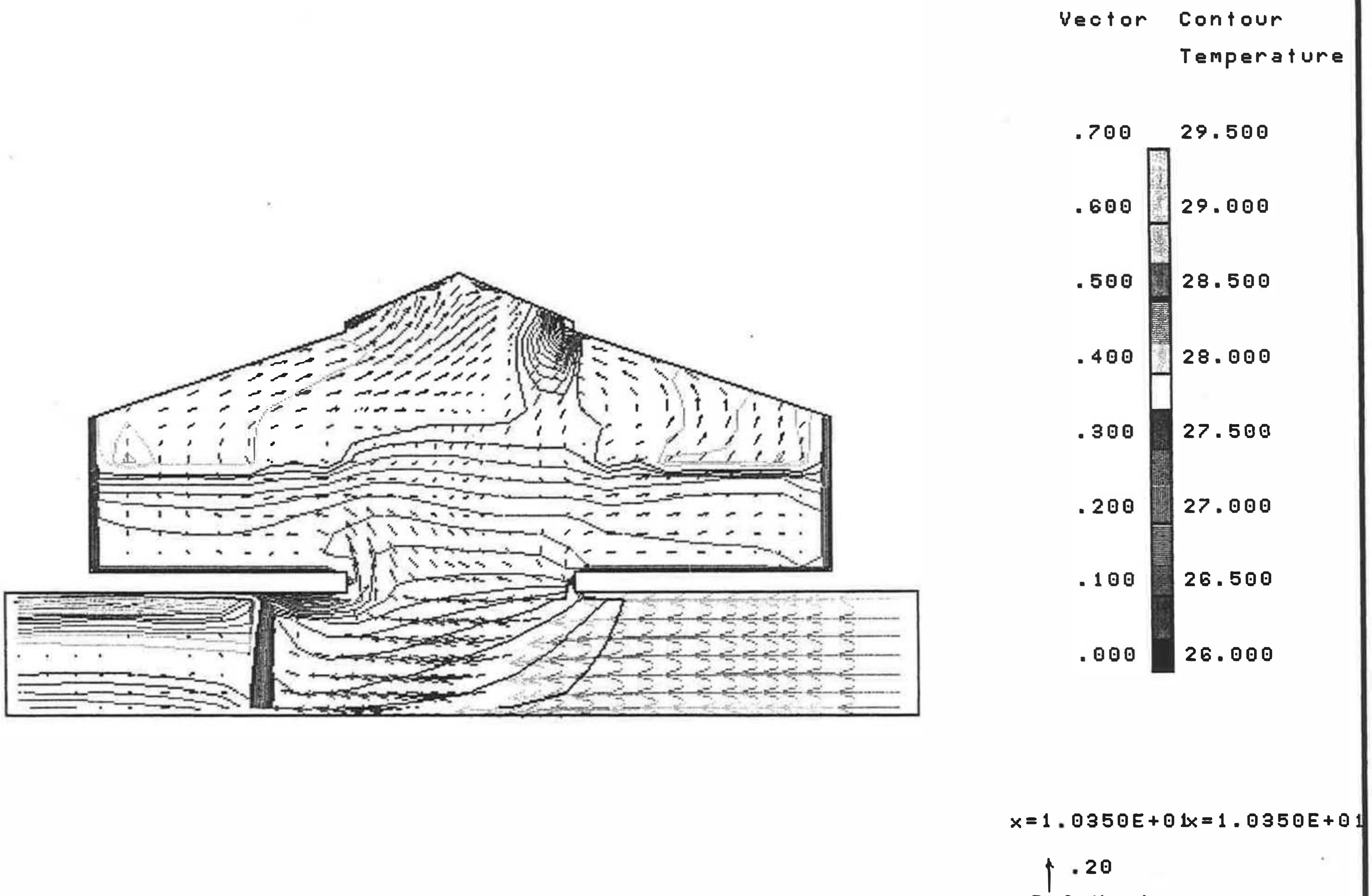
Boundary Types

Cuboid
Triangle
Volume Source
Free
Boundary Wall
Internal Wall
Opening
Planar Source
Angled Wall
Window
Door
Thick Wall

Job: p100sum

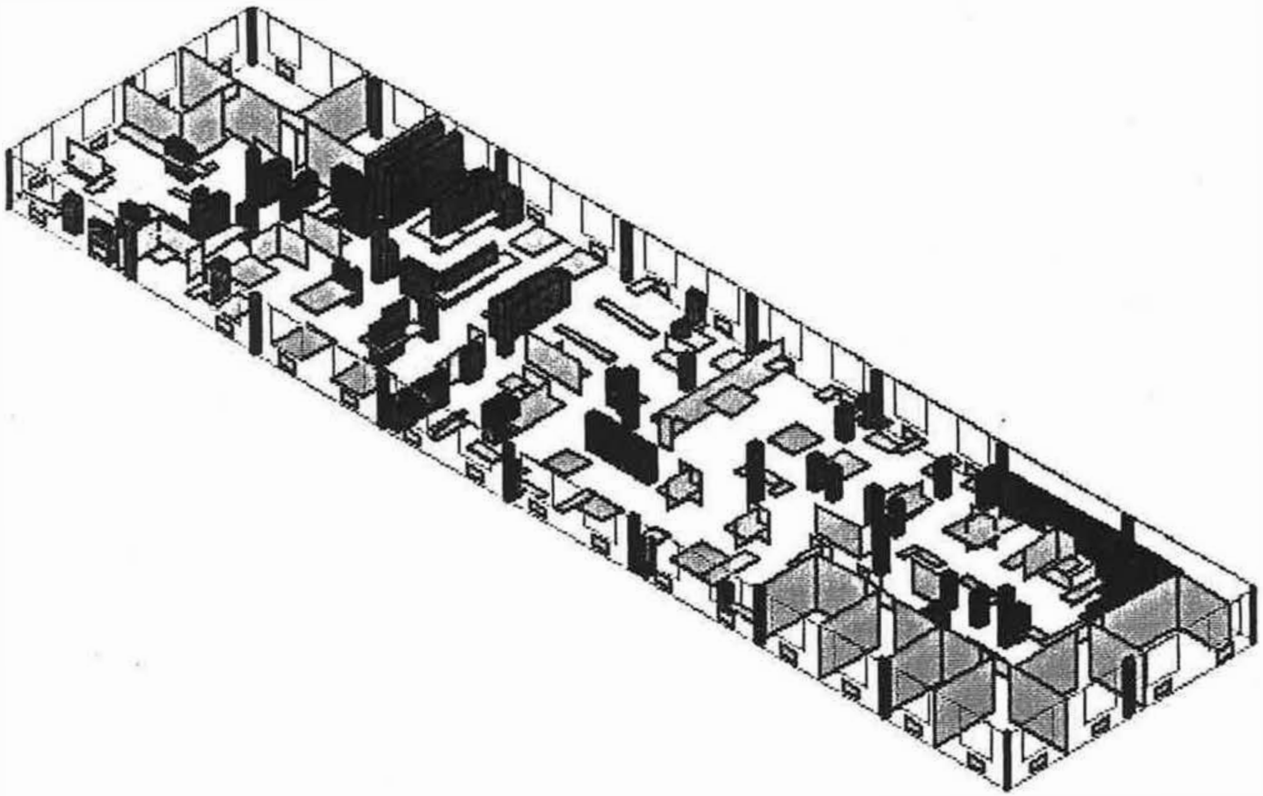
Title: Open Atrium Office Building - Summer Condition

FLOMERICS



Job: p100sum   Title: Open Atrium Office Building - Summer Condition

FLOMERICS



Boundary Types

Cuboid
Volume Source
Boundary Wall
Internal Wall
Supply
Opening
Planar Res.
Extract
Planar Source
Window

Job: dispvent

Title: Displacement vent - Large open plan office

FLOMERICS

Job: dispvent

Title: Displacement vent - Large open plan office

FLOMERICS

BEPAC PUBLICATIONS

TN 89/1	Predicting hourly internal daylight illuminances for dynamic building energy modelling - P J Littlefair
TN 89/2	The documentation and evaluation of building simulation models - T J Wiltshire and A J Wright
TN90/1	Availability of UK climatic data for use in simulation - E J Keeble
TN 90/2	Standard dwellings for modelling: details of dimensions, construction and occupancy schedules - E J Allen and A A Pinney
TN 90/3	Scale models and artificial skies in daylighting studies - P J Littlefair and C R T Lindsay
TN 91/4	Data sets for validating thermal models of buildings - K J Lomas
TN 90/5	A set of standard office descriptions for use in modelling studies - D J Leighton and A A Pinney
TN 91/6	The Harmonisation of Thermal Properties of Building Materials J A Clarke, P P Yaneske & A A Pinney Price: £25 to BEPAC members £100 to non-members
TN 91/7	Proceedings of BEP '91, Canterbury Price: £20 to BEPAC members £30 to IBPSA members £40 to non-members of either organisation
RR 91/1	Controls options in building energy simulation programs - Report compiled by E R Hitchin
	Papers presented at the Summer Meeting 'Low Energy Design', June 1992 Price: £5.00

Except where indicated, the cost of all publications is £6 to BEPAC members and £12 to non-members. The price is inclusive of postage and packaging.