A FLUID FLOW NETWORK SOLVER FOR INTEGRATED BUILDING AND PLANT ENERGY SIMULATION

J.L.M. Hensen[*] *and J.A. Clarke*[**]

[*] FAGO, Eindhoven University of Technology
P.O. Box 513, 5600 MB EINDHOVEN, The Netherlands
[**] Energy Simulation Research Unit, University of Strathclyde
131 Rottenrow, GLASGOW G4 0NG, Scotland

ABSTRACT

This paper outlines the ESP approach to the simulation of coupled heat and mass flows in integrated building and plant systems. It describes the equation-sets used to represent inter-zonal (building) and inter-component (plant) fluid flow, the method used for the simultaneous solution of these non-linear equations, and the solution coupling of the heat and mass conservation equation-sets. By means of a case study, the application in a real building performance evaluation context is demonstrated.

1. INTRODUCTION

In combined building and HVAC plant simulation, fluid flow phenomena are encountered in four principle areas:

- air flow through cracks and various openings in the building structure in the form of infiltration and natural ventilation;
- the flow of air through the distribution networks which exist to satisfy the heating/ cooling demands and ventilation requirements;
- the flow of heating/ cooling fluids through the plant system network;
- and the convective fluid flows within interior building spaces and plant components.

Some knowledge of the magnitude of these flows is necessary for load and energy calculations, system control analysis, thermal comfort assessment and contaminant/ moisture dispersal estimation. Although fluid flow is demonstrably an important aspect of building/ plant performance assessment, the sophistication of its treatment in many modelling systems has tended to lag the treatment applied to the other important energy flow paths. The principal reason for this would appear to be the inherent computational difficulties and the lack of sufficient data.
In recent times more emphasis has been placed on fluid flow simulation with two approaches extant:

Computational Fluid Dynamics (CFD); a method in which the conservation equations for mass, momentum and thermal energy are solved for all nodal points of a two- or three-dimensional grid inside or around the object under investigation. A well known example of a CFD model is PHOENICS (Spalding 1981).
While in theory the CFD approach is applicable to any thermo-fluid phenomenon, in practice, and in the building physics domain in particular, there are a several problematic issues of which the amount of necessary computing power (Chen 1988), the nature of the flow fields and the assessment of the complex, occupant-dependent boundary conditions are the most problematic. This has often led to CFD applications being restricted to the steady-state case which, in many building performance contexts, is atypical. Application examples in the field of building energy simulation are the prediction of temperature and velocity fields inside large or technically complex enclosures such as atria and television studios (Markatos 1984) and the

prediction of the pressure field around a building (Haggkvist et al 1989).

The Zonal Method; a method in which a building and its plant are treated as a collection of nodes representing rooms, parts of rooms and plant components, with inter-nodal connections representing the distributed flow paths associated with cracks, doors, ducts and the like. The assumption is made that there is a simple, non-linear relationship between the flow through a connection and the pressure difference across it. Conservation of mass for the flows into and out of each node leads to a set of simultaneous, non-linear equations which can be integrated over time to characterise the flow domain.

In the context of combined heat and mass flow simulation it is the zonal method which has proved (for the present at least) to be most commensurate with the modelling approach adopted by the ESP system. The reasons for this are threefold. Firstly, the number of nodes involved - say some 10-100 in a moderately sized building - is considerably less than employed in a CFD approach and so the additional CPU burden is acceptable. Secondly, there is a strong relationship between the nodal networks which represent the fluid regime and the thermal counterpart. This means that the information demands of the energy balance formulations can be directly satisfied. And, finally, the technique can be readily applied to combined multi-zone buildings and multi-component, multi-network plant systems.

It is the zonal method then which has been employed for several years as the basis of the air flow module of the ESP system and which underpins recent developments which have led to an improved equation solver and extensions of the technique to plant systems in general. Within ESP these developments are made available to a user via ESPmfs for use in cases where buoyancy effects are time-invariant, and as an integral encapsulation within ESPbps, the main simulation module, for use in cases where buoyancy has a strong temporal dimension.

This paper describes the theoretical basis of the ESP approach to fluid flow simulation in terms of the flow equation types offered and the underlying numerical solution strategy. The integration of the algorithm within the ESPbps numerical processing scheme is then described to demonstrate the technique employed to simulate combined heat and fluid flow in building/ plant systems. Finally, and briefly, the paper illustrates the application of the approach in practice.

2. THE APPROACH IN OUTLINE

Within the ESP approach, during each simulation time step, the problem is constrained to the steady flow (possibly bi-directional) of an incompressible fluid along the connections which represent the building/ plant mass flow paths network when subjected to certain boundary conditions regarding pressure and/ or flow. The problem reduces therefore to the calculation of fluid flow through these connections with the nodes of the network representing certain pressures. This is achieved by an iterative mass balance approach in which nodal pressures are adjusted until the mass residual of each internal node satisfies some user-specified criterion.

Information on potential mass flows is given by a user in terms of node descriptions, fluid types, flow component types, interconnections and boundary conditions. In this way a nodal network of connecting resistances is constructed. This may then be attached, at its boundaries, to known pressures or to pressure coefficient sets which represent the relationship between free-stream wind vectors and the building external surface pressures to result.
The flow network may consist of several decoupled sub-networks and is not restricted to one type of fluid. However, all nodes and components within a sub-network must relate to the same fluid type.
Nodes may represent rooms, parts of rooms, plant components, connection points in a duct or in a pipe, ambient conditions and so on. Fluid flow components correspond to discrete fluid

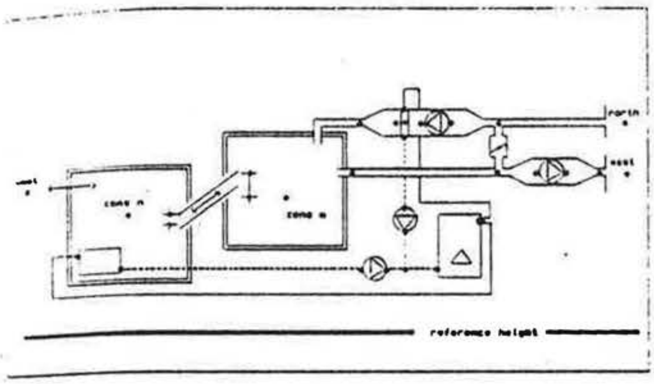

Figure 1 Example building and plant schematic

flow passages such as doorways, construction cracks, ducts, pipes, fans, pumps, etc. As an example Figure 1 shows a sketch of a part of a building consisting of two rooms, some connections between the rooms, a radiator heating system connected to one zone and an air heating system connected to the other zone. In this case the building and plant configuration contains at least two mass flow networks - one for the air and one for the water. One possibility with respect to the translation of this configuration into a nodal scheme is indicated by the dots.

Nodes are characterised by several data items including an identifier, the fluid type, the node type, the height above some arbitrary datum, temperature (for use only in ESPmfs) and several supplementary parameters dependent on the node type. At the present time only two fluid types are supported - air and water - although additional types will be developed as required. The nodal types currently on offer are summarized in Table 1.

Index	Type	Supplementary data
0	Internal; unknown pressure	none
1	Internal; known pressure	total pressure (Pa)
2	Boundary; known pressure	1) total pressure (Pa)
		2) index, indicating that node temperature:
		0 : is identical to some other node or is a constant
		1 : equals outside air temperature
3	Boundary; wind pressure	1) wind pressure coefficients index
	temperature = outside air	2) surface azimuth (° clockwise from North)

Table 1 Mass flow network node types

The nodes of the network represent either internal or boundary pressures. The difference is that only internal nodes are subjected to the mass balance tracking. Note that in the present context 'internal' is not necessary equivalent to 'inside' nor does 'boundary' necessarily equate to 'outside'. Usually the pressure at an internal node is unknown, although it may be treated as a known parameter as could be required, for example, in the case of an expansion vessel in a hydronic radiator system.

Flow components are characterised by an identifier, a type code (indicating duct, pipe, pump, crack, doorway, etc.) and a number of supplementary data items defining the parameters associated with a specific component type. When a certain flow component is repetitively present in the network, it need only be defined once. The currently supported fluid flow component types are summarized in Table 2. Within ESP each flow component has a subroutine counterpart which is used to generate the flow and flow derivative at each iteration. As an example, the type 220 component is elaborated in the next section. Detailed information on all component types can be found elsewhere (Hensen 1990).

Code	Type	Parameters
10	Power law volume flow resistance element	3
15	Power law mass flow resistance element (definition 1.)	3
17	Power law mass flow resistance element (definition 2.)	3
20	Quadratic law volume flow resistance element	3
25	Quadratic law mass flow resistance element	3
30	Constant volume flow rate element	2
35	Constant mass flow rate element	2
40	Common orifice flow element	3
50	Laminar pipe flow element	3
110	Specific air flow opening	1
120	Specific air flow crack	2
130	Specific air flow door	4
210	General flow conduit (duct or pipe)	6
220	Conduit ending in converging 3-leg junction & C = f(q/qc)	13
230	Conduit starting in diverging 3-leg junction & C = f(q/qc)	13
240	Conduit ending in converging 4-leg junction & C = f(q/qc)	17
250	Conduit starting in diverging 4-leg junction & C = f(q/qc)	17
310	General flow inducer (fan or duct)	7
410	General flow corrector (damper or valve)	17
420	Flow corrector with polynomial local loss factor	16
450	Ideal (frictionless) flow controller	8

Table 2 Currently available fluid flow component types

A **flow network** is defined by connections. Each connection is described in terms of the name of the node on its (arbitrarily declared) positive side, the height of the positive linkage point relative to the node on the positive side, the name of the node on the (arbitrarily declared) negative side of the connection, the height of the negative linkage point relative to the node on the negative side, the name of the connecting flow component and supplementary data which depends on the flow component selected. Note that more than one connection may exist between two nodes. The concept of a connection having a positive side and a negative side is used to keep track of the direction of fluid flow. For most mass flow component types, uni-directional fluid flow will result (in either direction). However, some component types may represent bi-directional fluid movement - for example in the case of a doorway where, due to the action of small density variations, over the height, bi-directional flow may exist.

3. THE CALCULATION PROCESS

Consider Figure 2 which shows two zones connected by some fluid flow component. It is assumed that each volume can be characterised by a single temperature and a single static pressure at some height relative to a common data plane. The inlet and outlet of the connecting component are at different heights relative to each other and relative to the nodes representing the volumes. Analysis of the fluid flow through a component i is based on Bernoulli's equation for one-dimensional steady flow of an incompressible Newtonian fluid including a loss term:

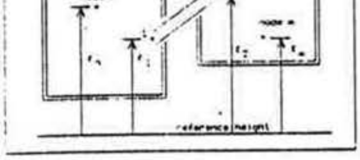

Figure 2 An example two zone connected system

$$\Delta P_i = (p_1 + \rho V_1^2/2) - (p_2 + \rho V_2^2/2) + \rho g (z_1 - z_2) \quad (Pa) \tag{1}$$

where ΔP_i is the sum of all friction and dynamic losses (Pa), p_1, p_2 are entry and exit static pressures (Pa), V_1, V_2 are entry and exit velocities (ms^{-1}), ρ is the density of the fluid flowing through the component (kgm^{-3}), g is the acceleration of gravity (ms^{-2}) and z_1, z_2 are the entry and exit elevations (m).

Bernoulli's equation can be simplified by combining several related terms. Stack effects are represented by the $\rho g(z_1 - z_2)$ term in equation (1). Dynamic pressures are the $\rho v^2/2$ terms, and total pressure is defined to be the sum of static pressure and dynamic pressure; i.e. $P = p + \rho v^2/2$. If nodes n and m represent large volumes (e.g. a room), the dynamic pressures are effectively zero. If the nodes represent some point in a duct or pipe network, there will be a positive dynamic pressure. Equation (1) thus reduces to:

$$\Delta P = P_n - P_m + PS_{nm} \quad (Pa) \tag{2}$$

where P_n, P_m is the total pressure at nodes n and m (Pa), and PS_{nm} is the pressure difference due to density and height differences across connection $n-m$ (Pa).

Equations (1) and (2) define a sign convention for the direction of flow: positive from point 1 to point 2 (or n to m). The flow within each fluid flow component is described by a relation of the form $\dot{m} = f(\Delta P)$. The partial derivatives needed for the establishment of the Jacobian matrix (representing nodal pressure corrections in terms of all branch flow partial derivatives) are thus related by $\partial \dot{m} / \partial \Delta P_{nm} = -\partial \dot{m} / \partial \Delta P_{mn}$.

This section now continues with an elaboration on the calculation of stack effect, followed by the description of an example flow component subroutine, and finally a sub-section on network solution.

3.1. Stack Effect

Within ESP each node is assigned a reference height. The reference height defines the mean height of the associated building zone or plant component. The node reference height may be expressed relative to any arbitrary datum level as long as this datum level is the same for all nodes in the network. The reference height is then used in the calculation of buoyancy driven flows (stack effect) in a manner similar to the approach suggested by Walton (1988).

The pressures at the inlet and outlet of the flow component (p_1, p_2 in Figure 2) can be related to the node pressures by the hydrostatic law:

$$P_1 = P_n + \rho_n g (z_n - z_1) = P_n - \rho_n g h_1 \quad (Pa) \quad \text{where } h_1 = z_1 - z_n \quad (m) \tag{3}$$

$$P_2 = P_m + \rho_m g (z_m - z_2) = P_m - \rho_m g h_2 \quad (Pa) \quad \text{where } h_2 = z_2 - z_m \quad (m) \tag{4}$$

The relative heights, h_1 and h_2, are a convenient way of expressing the flow component inlet and outlet heights. For example, it is quite common for flow components in the building fabric to only differ with respect to inlet and outlet heights relative to the zone heights. On the other hand, if the flow component is part of a duct or pipe network, the relative heights will be zero. Equation 1 can thus be reduced to:

$$\Delta P_i = P_n - P_m + \rho g (z_n + h_1 - z_m - h_2) - \rho_n g h_1 + \rho_m g h_2 \quad (Pa) \tag{5}$$

The terms $[\rho g (z_n + h_1 - z_m - h_2) - \rho_n g h_1 + \rho_m g h_2]$ can be collectively called the stack pressure, PS_i, acting on component i:

$$PS_i = \rho_n g (z_n - z_m) + h_2 g (\rho_m - \rho_n) \quad (Pa) \quad \text{(flow in positive direction)} \tag{6}$$

$$PS_i = \rho_m g (z_n - z_m) + h_1 g (\rho_m - \rho_n) \quad (Pa) \quad \text{(flow in negative direction)} \tag{7}$$

3.2. Flow Calculation

Within ESP each flow component has a subroutine counterpart which is used to generate flow and flow derivative at each iteration. As an example, consider the type 220 component: "conduit ending in converging 3-leg junction". This is a special case of a general flow conduit component (duct or pipe), which is described first.

For fluid flow through a conduit (ie. a duct or a pipe) with (a) uniform cross-sectional area, (b) no pressure gain due to fan or pump, and (c) steady-state conditions, the sum of all friction and dynamic losses is found from the general expression:

$$\Delta P = f L \rho \bar{v}^2 / 2D_h + \Sigma C_i \rho \bar{v}^2 / 2 \quad (Pa) \tag{8}$$

where f is the friction factor (-), L the conduit length (m), D_h the hydraulic diameter (m), A the cross-sectional area (m^2), $\bar{v}$ the average velocity (ms^{-1}), and C_i is the local loss factor due to fitting i (-).

The local loss factors, in the second right hand side term, represent dynamic losses resulting from flow disturbances caused by "fittings". In principle, local loss factors can be used for all those fittings or components which, from a fluid flow point of view, do not have to be modelled separately. Examples are: pipe and duct entries or exits, elbows, bends and obstructions, converging or diverging transitions.

The first right hand side term in the equation above is also known as the Darcy-Weisbach equation for frictional losses of fluid flow through conduits. The friction factor in this term depends on the type of flow, which is characterized by the Reynolds-number:

$$\mathrm{Re} = \bar{v}\, D_h / \nu \quad (-) \tag{9}$$

where ν is the fluid kinematic viscosity ($m^2 s^{-1}$). With respect to calculation of the friction factor, ESPmfs follows the approach as indicated in (ISSO 1986) which recognises only three regions: for Re ≤ 2300 **laminar flow** is assumed, and the friction factor is calculated from:

$$f = 64/\mathrm{Re} \quad (-) \tag{10}$$

In the **transition region**, for 2300 < Re < 3500, the flow may be either laminar or turbulent depending on the degree of disturbance. The friction factor is found by linear interpolation:

$$f = \frac{f_L(3500-\mathrm{Re}) + f_T(\mathrm{Re}-2300)}{3500-2300} \quad (-) \tag{11}$$

where f_L is the friction factor for laminar flow at Re = 2300 (-), and f_T is the friction factor for turbulent flow at Re = 3500 (-).

For Re ≥ 3500 the flow is assumed to be **turbulent**, and the friction factor is calculated from an explicit approximation of the implicit Colebrook-White equation, which is sufficient accurate for most technical purposes:

$$f = 1/[2\log(5.74/\mathrm{Re}^{0.901} + 0.27\,k/D_h)]^2 \quad (-) \tag{12}$$

where k is the absolute wall material roughness (m).

The mass flow rate through a general conduit can now be calculated from a "known" pressure difference by:

$$\dot{m} = A\sqrt{\frac{2\rho\Delta P}{fL/D + \Sigma C_i}} \quad (kg/s) \tag{13}$$

Because, effectively, we have an implicit formulation for $\bar{v}$, the calculation of $\dot{m}$ needs some sort of iteration. ESPmfs uses a fixed point iteration method with the following steps: (1) Re is calculated with $\bar{v}$ based on the most recent flow rate, (2) dependent on Re, a friction factor is established, (3) with this value a new flow rate is calculated which is compared with the one used in step 1, and (4) unless the difference is smaller than some user-specified error limit, the whole process is iteratively repeated.

In the equations above the fluid density and the fluid viscosity both depend on the direction of flow, ie. the temperature of the sending node. The partial derivative for this fluid flow component may be calculated from:

$$\frac{\partial\dot{m}}{\partial\Delta P} = \frac{.5\,\dot{m}}{\Delta P} \quad (kg/s/Pa) \tag{14}$$

unless ΔP is smaller than a certain threshold (say 10^{-20} Pa), in which case ESPmfs switches to numerical approximation:

$$\frac{\partial\dot{m}}{\partial\Delta P} = \frac{\dot{m} - \dot{m}^{\%}}{\Delta P - \Delta P^{\%}} \quad (kg/s/Pa) \tag{15}$$

where $^{\%}$ denotes the value in the previous iteration step.

In the approach above, the dynamic loss factor is treated as a constant. Depending on the problem at hand, this is not always justifiable. Flow junctions (tees, wyes, crosses, etc) for instance, tend to create local dynamic losses which are strongly related to the velocity ratios or flow rate ratios in the different parts of the junction; i.e. they depend on conditions elsewhere in the system network. The most common technique of describing dynamic local losses for junctions is by equations of the form:

$$\Delta P_p = C_{c,p}\,\frac{1}{2}\rho\bar{v}_c^{\,2} \quad (Pa) \tag{16}$$

where ΔP_p represents the total pressure losses through section p of the junction (Pa), $C_{c,p}$ is the dimensionless local loss factor for part p of the junction referenced to the velocity pressure at section c of the junction (usually this is a function of $\dot{q}_p/\dot{q}_c$), $\dot{q}$ is the volume flow rate ($\dot{m}/\rho$ (m^3/s)), subscript c indicates the common flow path, and subscript p indicates either the straight (s) or the branch (b) flow path.

Values for $C_{c,p}$ can be found in several handbooks (e.g. Miller 1971, ASHRAE 1985, Idelchik 1986). Figure 3 is an example of such data. This figure also indicates that different literature sources not always yield identical values for certain fittings.

Through curve fitting on data encountered in literature or found from experiments, it is generally possible to derive, for each flow path's local loss factor, an approximate expression of the form:

$$C_{c,p} = a_0 + a_1\left(\frac{\dot{q}_p}{\dot{q}_c}\right) + a_2\left(\frac{\dot{q}_p}{\dot{q}_c}\right)^2 + a_3\left(\frac{\dot{q}_{p'}}{\dot{q}_c}\right) + a_4\left(\frac{\dot{q}_{p'}}{\dot{q}_c}\right)^2 + a_5\left(\frac{\dot{q}_p}{\dot{q}_c}\cdot\frac{\dot{q}_{p'}}{\dot{q}_c}\right) \quad (-) \tag{17}$$

where a_i are fit coefficients†, subscript p indicates either straight (s) or branch (b) flow path.

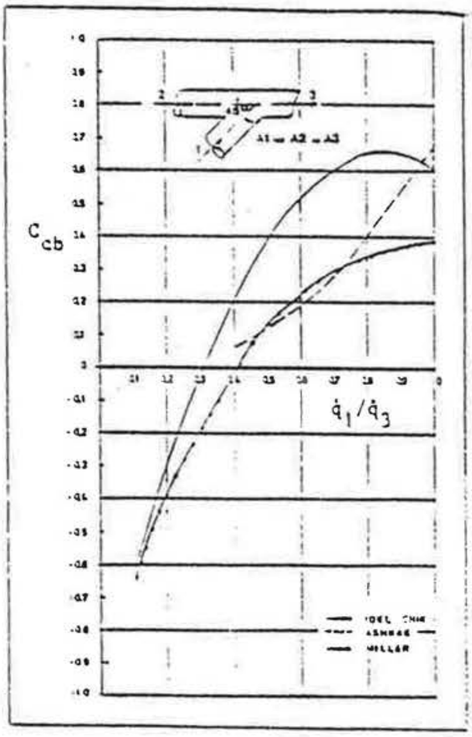

Figure 3 Local loss factor for a converging wye (45 °), round. Graph originates from ISSO (1986) (literature sources indicated are Idelchik 1986, ASHRAE 1985 and Miller 1971).
1 - branch flow path
2 - straight flow path
3 - common flow path

and subscript p' then indicates branch (b) or straight (s) flow path respectively.

Thus, it would be possible to calculate the pressure drop across the different flow paths of the junction. However, there are some difficulties when a junction would be approached as just another type of flow component:

- for many junctions, $C_{c,p}$ becomes very small or even negative for certain values of $\dot{q}_p/\dot{q}_c$ (see e.g. Figure 3). This may easily lead to numerical difficulties when the flow rate through the different flow paths of the junction is to be calculated explicitly from the pressure difference across it.
- a junction flow component type does not blend well with the concept of a flow component type linking two separate nodes - which forms the basis of all other pressure/flow component types - because a junction obviously involves more than two nodes

The former problem is usually dealt with by avoiding explicit evaluation of the flow/pressure relationships at the junction component level. This may be done by adding the dynamic local loss factors to adjacent flow components instead. For this, and to handle the second difficulty, ESPmfs offers some special flow conduit component types which allow assignment of the junction local loss factor $C_{c,s}$ or $C_{c,b}$ to the adjacent conduit.

The type 220 flow component ("conduit ending in converging 3-leg junction") is such a special conduit component which is schematically drawn in Figure 4. The conduit type

[*] note that the coefficients for describing $C_{c,s}$ differ from the fit coefficients which describe $C_{c,b}$. Note also that not necessary all $a_i \neq 0$

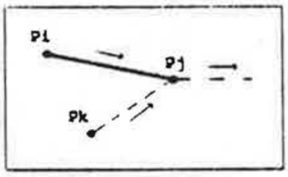

Figure 4 Conduit ending in converging 3-leg junction

component links node i and node j but also comprises the dynamic local losses of either the "straight" or the branch flow path of the junction. The other entrance of the junction has to be represented by the conduit component linking node k with node j; so a type 220 component incorporates only one half of the junction. It is assumed that there is only one connection between node i and node j, and between node k and node j respectively. Furthermore the flow directions are assumed to be as indicated in the figure. If, during simulation, the flow is in the other direction, a warning message will be issued.

In addition to the parameters describing the flow/pressure relationship for the conduit itself (which have the same meaning as in case of a general flow conduit) the following component description data is needed:

- a_0 to a_5 — fit coefficients describing $C_{c,p} = f(\dot{q}_p/\dot{q}_c \, , \, \dot{q}_{p'}/\dot{q}_c)$; subscript p refers to the junction section under consideration and subscript p' refers to the other entrance of the junction
- A_c — cross-sectional area of the common flow path (m^2)

It should be noted that here $C_{c,p}$ is described as a function of $(\dot{q}_p/\dot{q}_c \, , \, \dot{q}_{p'}/\dot{q}_c)$, while in literature one often finds $C_{c,p}$ described as function of $(\dot{q}_b/\dot{q}_c)$ or as function of $(\bar{v}_b/\bar{v}_c)$. Obviously, it is then a quite trivial task to transform the coefficients to the form used in ESPmfs.

The algorithm now functions as follows:

(1) the local loss factor $C_{c,p}$ is evaluated based on the most recent calculated flow rates, with:

$$\dot{q}_p = \dot{m}_{i,j}/\rho_i \quad (m^3/s) \tag{18a}$$

$$\dot{q}_{p'} = \dot{m}_{k,j}/\rho_k \quad (m^3/s) \tag{18b}$$

$$\dot{q}_c = \dot{q}_p + \dot{q}_{p'} \quad (m^3/s) \tag{18c}$$

unless either $\dot{m}_{i,j} \le 0$ or $\dot{m}_{k,j} < 0$, in which case (a) the local loss factor for this junction section is set to zero, (b) a warning message is issued, and (c) the algorithm proceeds with step (4).

(2) the local loss factor $C_{c,p}$ is converted into the local loss factor $C_{i,j}$ referenced to the velocity pressure in section $i - j$ by:

$$C_{i,j} = C_{c,p} \frac{\rho_i}{\rho_c} \left[\frac{\dot{m}_c / A_c}{\dot{m}_{i,j} / A_{i,j}} \right]^2 \quad (-) \tag{19}$$

where $\dot{m}_c = \dot{m}_{i,j} + \dot{m}_{k,j}$ (kgs^{-1}) and $\rho_c = \rho_j$ (kgm^{-3}).

(3) then the local loss factor $C_{i,j}$ is added to the sum of all other local loss factors ΣC_i of conduit $i - j$

(4) and then the algorithm can proceed as if it where a general flow conduit

3.3. Network Solution

Each fluid flow component, i, thus relates the mass flow rate, $\dot{m}_i$, through the component to the pressure drop, ΔP_i, across it. Conservation of mass at each internal node is equivalent to the mathematical statement that the sum of the mass flows must equal zero at such a node. Because these flows are non-linearly related to the connection pressure difference, solution requires the iterative processing of a set of simultaneous non-linear equations subjected to a given set of boundary conditions. The technique employed by ESP is to assign an arbitrary pressure to each internal node to enable the calculation of each connection flow from the appropriate connection equation. The internal node mass flow residuals are then computed from:

$$R_i = \sum_{k=1}^{K_{i,i}} \dot{m}_k \quad (kg/s) \tag{20}$$

where R_i is the node i mass flow residual for the current iteration (kgs^{-1}), $\dot{m}_k$ is the mass flow rate along the kth connection to the node i (kgs^{-1}) and $K_{i,i}$ is the total number of connections linked to node i.
The nodal pressures are then iteratively corrected and the mass balance at each internal node is re-evaluated until some convergence criterion is met. The method used in ESP is based on an approach suggested by Walton (1988).

Problem	Original Solver		New Solver		Iteration	CPU
	CPU Seconds	Iterations 1st hr - 24 hrs	CPU Seconds	Iterations 1st hr - 24 hrs	Ratio 24 hrs	Ratio 24 hrs
1. atria	3087	6363 - 152117	55	137 - 522	291	56
2. house 1	377	374 - 27863	17	29 - 459	60	21
3. house 2	48	146 - 2510	23.2	11 - 105	23	2
4. 2 zone	9	309 - 2376	3.6	16 - 287	8	2
5. 3 zone	3	27 - 358	2.5	4 - 90	3	1
6. Trombe	2168	14009 - 122754	50.2	29 - 474	258	43
		1st hr - 2nd hr		1st hr - 2nd hr	1st - 2nd hr	
7. large		13270 - 25318		24 - 1	552 - 25318	

Table 3 Bench-mark results. All runs were performed on a SUN 3/50 and correspond to a one day (24 hour) simulation

This approach was implemented and tested in an earlier version of ESP and shown to result in considerable speed improvements as evidenced in Table 3 (Clarke & Hensen 1988).

The latest ESP model has a further enhanced solver which has resulted in additional iteration reductions. However, at the time of writing no bench-mark results were available. The solution method is based on a Newton-Raphson technique applied to the set of simultaneous nonlinear equations (for example see Conte and De Boor 1972). With this technique a new estimate of the vector of all node pressures, $\mathbf{P}^*$, is computed from the current pressure field, $\mathbf{P}$, via:

$$\mathbf{P}^* = \mathbf{P} - \mathbf{C} \tag{21}$$

where the node pressure correction vector, $\mathbf{C}$, is determined on the basis of a simultaneous solution of a Jacobian matrix which represents the nodal pressure corrections in terms of all branch flow partial derivatives. The pressure corrections vector $\mathbf{C}$ is given by:

$$\mathbf{C} = \mathbf{R}\,\mathbf{J}^{-1} \tag{22}$$

where $\mathbf{R}$ is the vector of nodal mass flow residuals and $\mathbf{J}^{-1}$ is the inverse of the square Jacobian matrix (N*N for a network of N nodes) whose diagonal elements are given by:

$$J_{n,n} = \sum_{k=1}^{K_{n,n}} \left(\frac{\partial \dot{m}}{\partial \Delta P} \right)_k \quad (kg/s\ Pa) \tag{23}$$

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

$$J_{n,m} = \sum_{k=1}^{K_{n,m}} - \left(\frac{\partial \dot{m}}{\partial \Delta P}\right)_k \quad (kg/s\ Pa) \tag{24}$$

where $K_{n,m}$ is the number of connections between node n and node m. This means that - for internal nodes - the summation of the terms comprising each row of the Jacobian matrix are identically zero.

ESP currently uses LU decomposition with implicit pivoting (also known as Crout's method with partial pivoting) for solution of the matrix equation J C = R for the unknown pressure correction vector C. The implementation in use by ESP originates from an algorithm by Press (et al. 1986). In this case the matrix J is decomposed to a lower triangular matrix L and an upper triangular matrix U, such that L U = J. This decomposition is used to solve the linear set:

$$\mathrm{J\ C = (L\ U)\ C = L\ (U\ C) = R} \tag{25}$$

by first solving, by forward substitution, for the vector Y such that L Y = R and then solving (by back substitution) U C = Y. The advantage is that both substitutions are quite trivial. Pivoting is used to make the method numerically stable.

As a future possibility, sparse matrix methods could be used to reduce further the storage and execution time requirements.

It should be noted that it is quite easy to define a mass flow network which has no unique solution. One requirement for solution is that at least one of the node pressures is known. A second requirement is that all nodes must be linked, through some path, to a known pressure node.

Conservation of mass at each internal node provides the convergence criterion. That is, if $\sum \dot{m}_k = 0$ for all internal nodes for the current system pressure estimate, the exact solution has been found. In practice, iteration stops when all internal node mass flow residuals are below one of two user definable thresholds: ERRMAX the largest percentage residual flow error, or FLOMAX the largest absolute residual flow error.

In some cases, large corrections for the successive pressure correction applied to any node during the iteration process may cause a numerical instable situation. Therefore, ESP offers PMAX a user definable maximum pressure correction applied to any node during the iteration process.

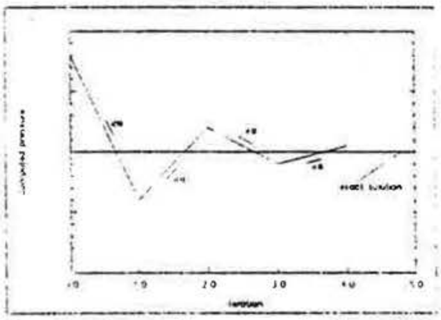

Figure 5 Example of successive computed values of pressure and oscillating pressure correction at a single node

As noted by Walton (1988), there may be occasional instances of low convergence with oscillating pressure corrections on successive iterations at a single node. In the case shown in Figure 5, each successive pressure correction is a constant ratio of the previous correction - that is $C_i = -0.5 * C_i^{\%}$ ($^{\%}$ denotes the previous iteration step). In a number of tests the observed oscillating corrections came close to such a pattern. By assuming a constant ratio, it is simple to extrapolate to the 'final solution':

$$P_i^* = P_i - C_i / (1-r) \quad (Pa) \tag{26}$$

where r is the ratio of C_i for the current iteration to its value in the previous iteration. The factor $1/(1-r)$ is called a relaxation factor. The extrapolated value of node pressure can be used in the next iteration. If it is used, then r is not evaluated for that node in the following iteration but only in the one thereafter. In this way, r is only evaluated with unrelaxed pressure correction values. This process is similar to Steffensen iteration (Conte and De Boor 1972) which may be used with a fixed point iteration method for individual non-linear equations. The iteration correction method presented above gives a variable and node dependent relaxation factor. When the solution is close to convergence, Newton-Raphson iteration converges quadratically. By limiting the application of the relaxation factor to cases where r is less than some value (ESP's user definable parameter STEFFR) such as -0.5, it will not interfere with the rapid convergence.

4. Combined Heat and Mass Flow Simulation

Within the ESP system, a fluid flow simulation may be initiated independently of the main energy simulation or pursued in tandem. In the former case the assumption is made that the flows are predominantly pressure driven and that buoyancy effects, although included, are time invariant (or user specifiable). In the latter case ESP must establish and solve the coupled, matrix equations corresponding to the heat and fluid flows within the multi-zone building and the multi-component plant. The ESP scheme, which is reported fully elsewhere (Clarke 1990), can be summarised, for any computational time-step, as follows.

- the energy balance, state-space equations corresponding to the finite volumes which represent the plant-side discretised components and distribution network are established on the basis of the latest values of the building-side state variables and plant component/ network mass flows
- this plant matrix equation is then solved by a sparse matrix technique taking into account any defined control action
- the energy balance, state-space equations corresponding to the finite volumes which represent the building-side discretised constructions, surfaces and air volumes are then established on the basis of the latest values of the plant flux inputs and building-side air flows
- this building matrix equation is then solved by a customised matrix inversion technique which employs a partitioning and ordering technique which ensures that only non-zero matrix entries are processed and which integrates control system characteristics within the solution process
- the whole-system, fluid flow equations are then solved, iteratively, by the technique described earlier in this paper, utilising the newly established building and plant-side state variables to estimate the buoyancy effects
- if required, time-step control can be activated to prevent the evolution of time in cases where the newly computed state-variables differ markedly from the latest values assumed when the matrix equations were established

- finally, the simulation clock is incremented and the process repeats

Since the time constant associated with the state-space equations representing plant-side finite volumes are often an order of magnitude smaller than their building-side counterpart, a facility is provided to allow the plant system equations to be established and solved at a greater frequency than the building system equations.

5. A Case Study

Recently one of the authors was involved in a case study which involved air infiltration and ventilation modelling with ESPmfs. This study was concerned with the interaction between mechanical ventilation system and a house.

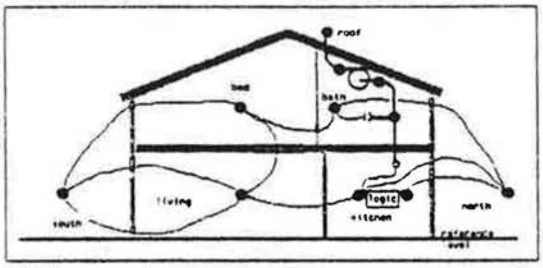

Figure 6 Diagrammatic representation of house, ventilation system and mass flow network

Figure 6 shows a schematic cross-section of a one-family house with a mechanical exhaust-only ventilation system. The air leakage and distribution network is superimposed. The house has a semi-open kitchen and open stairs to the first floor. The ventilation system consists of an exhaust fan to exhaust the indoor air to the outdoors. The make-up air from the outdoors leaks in through cracks and purpose provided vents in the building envelope. It was assumed that during certain times (between 16:00 and 20:00 hrs) the occupants open the kitchen north window unless the inside-outside temperature difference is larger then a certain value (say 20 ° C). A detailed computer simulation study was carried out to predict the results of different system options in terms of occupant comfort, condensation risk, heating capacity required and energy consumption & cost. As part of the study it was necessary to answer two questions for the configuration as indicated in Figure 6: (i) is the amount of air exhausted by the fan influenced by weather and/or occupant behaviour, and (ii) to what extent is the supply of fresh air to the bedrooms influenced by these variables.

Mass flow simulations were carried out for the heating season period of some test climate year. From the results it was clear that in this case the ventilation air exhaust rate is relatively insensitive to wind speed, wind direction, ambient air temperature, and occupant opening of the kitchen window; the exhaust rate remains essentially constant. It should be noted however that in this case, a relatively high standard ventilation system was employed.

Figure 7 shows some typical results with respect to the supply of fresh outdoor air to the bedrooms (flow path from node south to node bed). During the first days one clearly recognizes the effect of the kitchen window opening in the late afternoon (the days thereafter the window was kept shut due to the outdoor temperature). Another effect which is apparent from these results is the influence of wind speed and/or direction. From the results it is clear that the fresh air supply to the bedrooms varies strongly. During certain periods, used air is even transported to outside via the bedrooms.

Here we refrain from inferring any further conclusions from these results. The previous merely

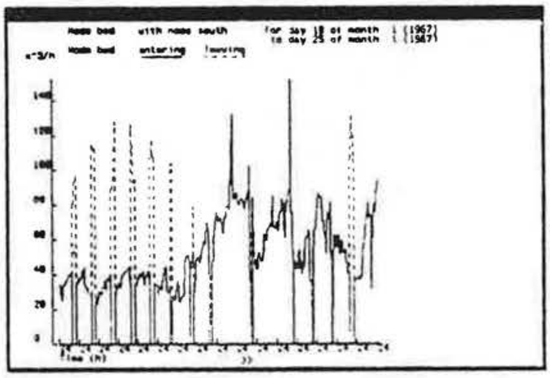

Figure 7 Flow of air through connection south-bed (dashed line indicates opposite flow direction) for January 10^{th} until January 25^{th}

serves to give a taste of the kind of results which may be expected from integrated building and plant simulation.

6. Conclusions

This paper has demonstrated how a mass flow network method can be used to provide a unified model of major building and plant fluid flows. It was shown how this model can be used for the simulation of coupled heat and mass flows in buildings. The performance of the model indicates that it is practical to solve the building/ plant heat and mass flow network in detail. Solution of complex fluid flow networks for problems involving many time steps is now feasible on current small computers.

It is felt that the model reflects the current state of the art in the field of network modelling approach to simulation of coupled heat and mass flows in buildings. Development of the model did reveal however that research is still needed in several areas. These include development of additional fluid flow component models (especially improved large opening models), modelling of intrazone effects by simplified methods and by integration with CFD modelling methods, expansion of the wind pressure database, expansion of the actual building and plant components 'database', and experimental validation of the simplifying assumptions in the flow component models and the network method.

Acknowledgements

The authors are indebted to George Walton who so willingly shared his theoretical approach to the simulation of air flow in buildings.

References

ASHRAE 1985. *Handbook of Fundamentals*, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta.

Chen Qingyan 1988. "Indoor airflow, air quality and energy consumption," PhD Thesis Delft University of Technology.

Clarke, J.A. 1985. *Energy simulation in building design*, Adam Hilger Ltd. Bristol.

Clarke, J.A. and J.L.M. Hensen 1988. "Implementation of a new air flow equation solver in ESP," University of Strathclyde, Energy Simulation Research Unit, ESRU report, Glasgow.

Conte, S.D. and C. de Boor 1972. *Elementary numerical analysis: an algorithmic approach*, McGraw-Hill, New York.

Hensen, J.L.M. 1990. "ESPmfs, a building & plant mass flow network solver," collaborative FAGO/ESRU report 90.14.K, Eindhoven University of Technology.

Häggkvist, K., U. Svensson, and R. Taesler 1989. "Numerical simulations of pressure fields around buildings," *Building and Environment*, vol. 24, no. 1, pp. 65-72.

Idelchick, I.E. 1986. *Handbook of hydraulic resistance*, Springer Verlag, Berlin. 2nd ed., rev. and augmented

ISSO 1986. "Luchtkanaalberekening," ISSO-publikatie 17, Stichting ISSO, Rotterdam.

Markatos, N.C. 1984. "Computer analysis of building ventilation and heating problems," Concentration Heat and Momentum Ltd report, London.

Miller, D.S. 1971. *Internal flow: a guide to losses in pipe and duct systems*, British Hydromechanics Research Association, Fluid Engineering Centre, London.

Press, W.H., B.P. Flannery, and S.A. Teukolsky 1986. *Numerical recipes : the art of scientific computing*, Cambridge University Press, Cambridge, Mass..

Spalding, D.B. 1981. "A general purpose computer program for multi-dimensional one- and two-phase flow," in *Mathematics and Computers in Simulation XXIII*, pp. 267-276, London.

Walton, G.N. 1988. "Airflow network models for element-based building airflow modelling," National Institute of Standards and Technology technical paper, Gaithersburg. intended for ASHRAE 1989 Annual Meeting

Walton, G.N. 1988. "Documentation for AIRNET test program," National Institute of Standards and Technology internal report, Gaithersburg.

DISCUSSION

KOHONEN R. (Finland)

I think you have taken into account only one part of the thermal coupling of air flows and energy balance of a room space. Have you any plans to include the "heat recovery effect" that influences the dynamics of wall structures in your model ?

ANSWER :

The "heat recovery effect" (i.e. heating of infiltration air and cooling of building structure (or vice-versa) can -for the time being- only be modeled in a rudimentary fashion by ESP. This may be achieved by introducing one or more building zones (say located in the wall) and adding additional nodes and air flow components to the fluid flow network. We did not consider a direct approach at this point in time because the current research project focusses on plant side extensions.

HAVES Ph. (UK)

In the case of, for example, VAV air-conditioning systems, what are the limitations on the control action (non-linearity, hysteresis, dead band, etc.) that can be treated using your matrix-based methods ?

ANSWER :

ESPmfs currently offers a general parameterized flow corrector component which may be described in terms of characteristic (linear or logarithmic) and hysteresis. There is another flow corrector available, which may be described in terms of a variable dynamic local loss factor (defined by polynomial coefficients) and hysteresis. There may be more flow corrector types available in the future.

Actual flow control is achieved by "linking" the flow corrector with some input signal originating from another node or component in the network. This could be a previous time-step value, but the user can ensure that the "current value" is used by adjusting the iteration convergence criteria parameters.

CHEUNG J.M. (UK)

Could you please explain why in table 3 the order of the CPU seconds in both methods (i.e. the original and new solvers) is different in item 2 and 3 ?

ANSWER :

Because the system of non-linear equations is solved by an indirect (i.e. iterative) method the improvement ratio depends strongly on the problem on hand which is clearly evidenced by the table. In general : problems involving both relatively large and relatively small flow resistances were very hard to solve for the old solver. This is not the case for the new solver.

SUTER P. (Switzerland)

You mentioned further work on interzonal exchange, i.e., the consideration of different zones in 1 room. On this subject work is under way in the frame of IEA Annex 20 : "Air flow patterns in buildings", which could in the future be included into modelling systems as presented by the paper.

ANSWER :

Yes, we are aware of these research efforts and certainly hope to be able to use results of this work in due time.