Implementing the Results of Ventilation Research
16th AIVC Conference, Palm Springs, USA
19-22 September, 1995

Predicting Indoor Air Flow by Combining Network Approach, CFD and Thermal Simulation

J A Clarke*, J L M Hensen*, C O R Negrao**

* University of Strathclyde, Montrose St., Glasgow G1 1XJ, Scotland, UK
** ESRU PhD Student associated with CNPq-Brasilia, Brazil

SYNOPSIS

This paper describes a method which aims to generate an overall view of multizone building air flow by integrating methods for bulk air flow analysis, air flow field analysis, and building thermal analysis.
This has been achieved by implementing a computational fluid dynamics approach within the ESP-r building energy simulation environment which already incorporated a nodal air flow network approach.

The current state of the method is demonstrated by a case study. The main conclusion from this is that the integrated method is very promising. Other preliminary conclusions concern the difficulty of finding suitable boundary conditions and numerical values for input parameters.

1 INTRODUCTION

In building energy simulation it is still common practice to assume a uniform distribution of air temperature inside a thermal zone. Thus a room - or other type of building space - is considered to be completely mixed.
From this simplification follows that there is neither information on the spatial distribution of air velocities nor on distribution of air temperature and contaminants within a zone.

Although uniform zone air might be a reasonable assumption for many problems where the focus is on (long-term) energy matters, this simplification is not valid for cases involving relatively strong couplings between heat and air flow or involving relatively strong temperature gradients. The former typically occurs when the strategy to avoid overheating in the summer is to increase natural ventilation. A displacement ventilation system is a typical example of the latter.

Given the increased practical importance of such applications, there is a growing interest in practice and academia to establish prediction methods which are able to integrate air flow prediction and building thermal simulation.

Building energy/environmental prediction based on computational modelling is receiving much attention at the present time: mathematical models, discretisation techniques and numerical methods are being refined, and application know-how is maturing. Building energy simulation (BES), in which the building's distributed capacity and air volumes are discretised (the latter relatively crudely), and computational fluid dynamics (CFD), in which some fluid domain is finely discretised, are two significant development fields.

After outlining some other approaches to predicting zonal air flow, this paper describes the method employed in the current research, which is to combine BES and CFD.

2 PREDICTING ZONAL AIR FLOW

There are several approaches for the prediction of zonal air flows:

Empirical

These approaches use jet, plume and other flow theories to predict certain aspects (typically penetration depth, jet dimensions, etc) of intra zonal air flows. As such they have a limited application area, and there are for instance almost no theories which deal with dynamic aspects nor with buoyancy driven flows (except for some specific cases such as drafts from cold windows (Kriegel 1973)).

Nodal network

In this approach (see eg Inard and Buty 1991) a building space is represented by a number of linked nodes, where the interconnections represent certain flowrate vs pressure relationships (usually based on the empirical approaches above). Obviously this approach can not provide detailed results of the temperature and flow fields. It is hoped however that the results will be sufficient - in terms of energy matters - to describe the bulk flows and the main temperature gradients. The main problems associated with this approach are related to nodal distribution, the pressure flowrate relationships, establishing boundary conditions, and the parameter values.

Computational fluid dynamics

CFD approximates an enclosed space by a series of control volumes. Air flow, turbulence, and energy propagation are represented in each of the control volumes by a series of discretised conservation equations. In principle this is an 'ideal' approach, however there are a number of practical problems such as huge computational burden, theoretical limitations of the turbulence models (especially for the type of low Reynolds flow encountered in buildings), etc.

Table 1 Air flow modelling approaches on offer in the ESP-r virtual building energy modelling laboratory

scheduled infiltration and ventilation rates
high frequency measured infiltration and ventilation rates
nodal network air flow method in stand-alone mode
nodal network air flow method integrated with thermal solver
CFD (2D or 3D, steady-state or dynamic) with fixed boundary conditions
CFD with boundary conditions generated by network and thermal solvers

Despite the associated practical problems, it is felt that CFD is the way forward for predicting intra zonal air flows in a building energy context. In order to enable future research in this area, ESP-r was recently extended to include a CFD algorithm. This is in line with the objective for ESP-r to be a virtual laboratory for building energy modelling issues. In terms of air flow modelling features, the system now offers the possibilities as outlined in Table 1.

3 COMBINING BES, NETWORK APPROACH, AND CFD

Elaborate descriptions of the internal workings of the ESP-r building simulation environment in terms of energy simulation in general can be found in (Clarke 1985) and in terms of simultaneous heat and mass flow simulation in (Hensen 1991). The current paper focusses on an approach for combining building energy simulation, with both nodal network and a CFD approaches to air flow prediction.

At the core of the approach is a method which allows ESP-r's existing network flow model (see eg Clarke and Hensen 1991). to operate in tandem with a CFD algorithm which is fully integrated at the source code level. As explained in another paper (Hensen 1995) ESP-r uses partitioning to couple building heat flow and air flow - and building moisture flow and plant heat flow and plant fluid flow(s) and lighting and electric power and ....- because of the advantages which accrue from problem partitioning.

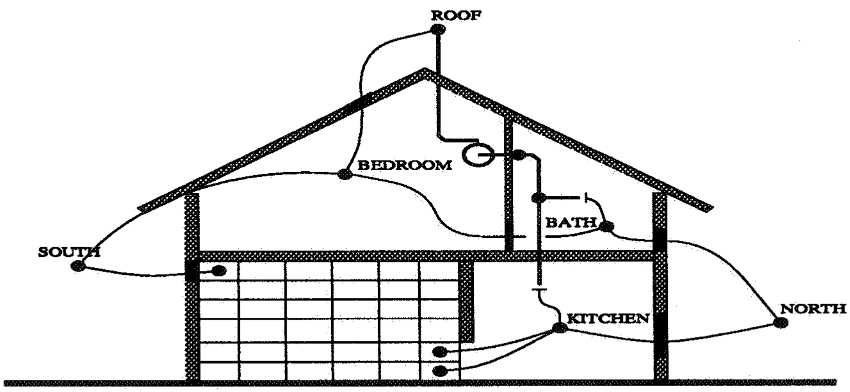

Figure 1: A connected flow network and CFD domain

The integration and implementation details of the CFD algorithm are the subject of another publication (Clarke et al. 1995).
The CFD algorithm is based on the 2D, steady state TEACH system (Gosman and Ideriah 1976), with extensions to enable 3D, transient operation and buoyancy effect. These extensions are reported elsewhere (Negrão 1994).
The implementation of CFD within ESP-r has been enabled at two levels of granularity in order to allow researchers to explore the conflation issue. At the first level, the CFD domain is essentially decoupled with the building/^plant solver (with fully integrated network air flow) passing the intgernal surface temperatures to the CFD solver, which passes back the surface convection coefficients. At the second level, one or more network nodes are replaced by a gridded CFD domain, with the 'snipped' network reconnected to one or more of the CFD cells as illustrated in Figure 1.
The actual coupling details are as illustrated in Figure 2a which shows two possible scenarios: a one-to-one coupling to represent a window crack and a one-to-many coupling to represent a doorway.

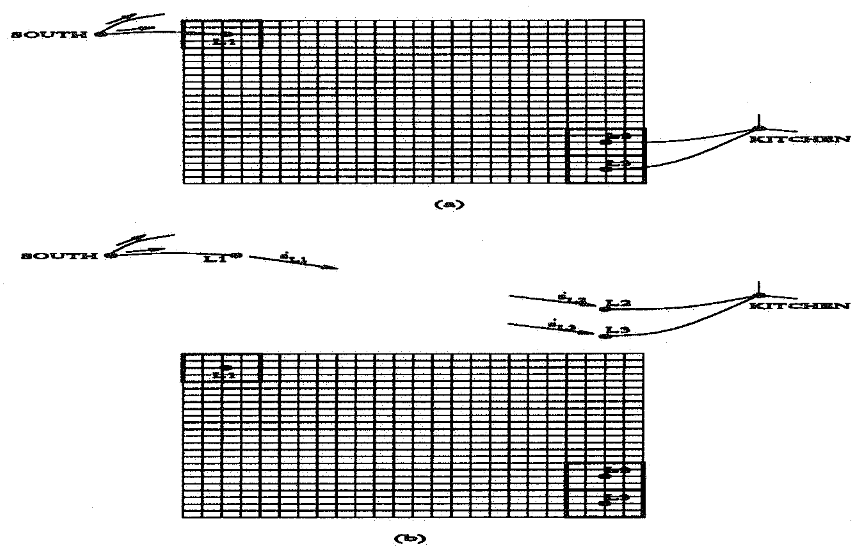

Figure 2: Node-to-grid cell coupling strategies

The nodes L1, L2 and L3 represent the effect of the CFD domain on the network model. To allow each domain solution to be performed separately, the flow network must be decoupled from the CFD domain. Figure 2b shows how the domains are detached from each other. The sources or sink of mass ($\dot{s}_{L1}$, $\dot{s}_{L2}$ and $\dot{s}_{L3}$) on the decoupled network represent the air flow entering or leaving the finely discretised domain. If these sources or sinks of mass are considered known values, the flow network can then be solved. Pressures at the network nodes, including nodes L1, L2 and L3, and flow rates through the network, including the branches SOUTH-L1, L2-KITCHEN, L3-KITCHEN are then determined.

The $\dot{s}_{L1}$, $\dot{s}_{L2}$ and $\dot{s}_{L3}$ quantities which are indicated in Figure 2b are generated by the CFD algorithm. These quantities are the product of the velocity components crossing the interface of the cell, densities and interface cell areas. The source/sink terms are thus computed by the following expression:

$$\dot{s}_k = \sum_{j}^{m} \sum_{i}^{n} (\rho V A)_{i,j,k}; \quad k = L1,\ L2,\ L3 \tag{1}$$

where m is the number of cells which are connected to a mass flow network node, n is the number of interfaces of each cell (the interfaces at the opening boundary are not included), ρ is the air density, V is the velocity component at the cell interface and A is the interface area. If the flow is entering a cell it is considered positive while if it is exiting it is negative. a +ve $\dot{s}_k$ is therefore a source of mass and -ve a sink.

Operating separately, but in tandem, the solution of the CFD domain is carried out using the BES-side generated boundary conditions: imposed velocities (momentum) within the coupling branches (SOUTH-L1, L2-KITCHEN and L3-KITCHEN). This requires knowledge of flow direction in order to determine the correct coupling point. In the current case the velocity is determined from the network-side flows (as computed for SOUTH-L1, L2-KITCHEN and L3-KITCHEN) divided by the product of sending node density and branch area.

Since the air flow between the coupling points is CFD-side dependent, while the pressures or momentum are network-side dependent, the two solvers must iterate until convergence is reached. Since the number of CFD-side equations for a single zone will usually be considerably greater than the number of equations for the building/plant flow network, the CFD-side controls the iteration - i.e. the network solution is initiated and completed for each CFD iteration.

3 CASE STUDY

The following example is intended to indicate the potential of the new method and demonstrate the expected magnitude of the differences in predictions between the combined approach and the network-only approach. The case studied is the house problem of Figure 1 with only relative simple models and coarse grids applied to allow investigation of the CFD network connection strategies. The two cells located at the openings are connected to two flow network nodes, one external (south) and one located within the kitchen. Initially, buoyancy effects are not considered. The wind induced pressures at the external node are evaluated by means of pressure coefficients which differ with surface location. A non-linear relationship between mass flow rate and pressure difference is defined to represent the connection between flow network nodes and CFD cells:

$$\dot{m} = 0.65\ A \sqrt{2\ \Delta P}. \qquad (2)$$

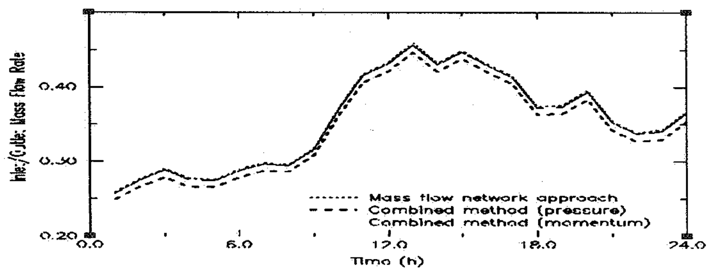

Figure 3: Mass flow rates - buoyancy not included.

Convergence of the CFD domain is only possible if the convergence criteria of the network domain is of the same order of magnitude. Low linear under-relaxation factors for the CFD momentum equations ($\alpha = 0.1$) were necessary to avoid boundary condition oscillations (when pressures and momentum as evaluated by the flow network). Approximately 600 iterations were necessary for a simulation which required approximately 200 iterations for a CFD only model.

In order to compare the combined model with the network flow approach, the flow network as shown in Figure 1 was simulated. A simulation was performed for a day in which the wind vector would induce a pressure at node SOUTH which was higher than that at node KITCHEN, giving a west to east air flow. The two boundary nodes (SOUTH AND KITCHEN) considered above are now connected to node LIVING.

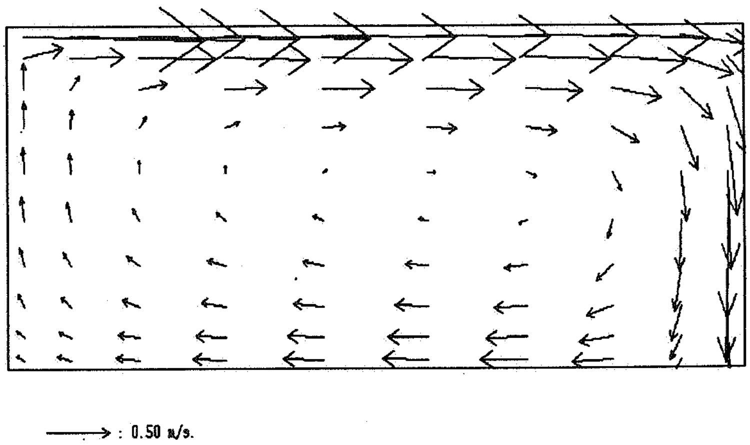

Figure 4: Velocity field - buoyancy not included

The LIVING node then represents the entire pressure field of the zone and no stack effect is considered. Figure 3 shows the differences between the air flows evaluated by the network method and by the combined method. As can be seen the results are similar.

Figure 3 shows two sets of results for the combined approach, representing two boundary condition types: imposed pressure and imposed momentum. Only the latter is considered in the current paper.

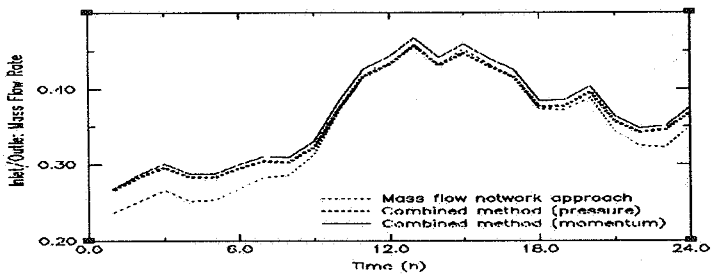

Figure 5: Mass flow rate - buoyancy included

Buoyancy effects are now introduced in order to investigate the effect of natural convection on the flow. The difference in height between nodes SOUTH and KITCHEN induce stack pressures at nodes L1, L2 and L3. As expected, the flow is affected by natural convection. The inlet fresh air produces a recirculating flow inside the zone for either kind of boundary condition. This promotes a higher inlet air flow to the room as evident in a comparison of the flow rates of Figures 3 and 5: in the latter case the differences are more pronounced at the beginning and end of the day. At these times, the outside temperature is lower than the wall surface temperatures and natural convection is more significant. During other periods, the ambient temperature approximates to the wall surface temperatures and the buoyancy effect disappears.

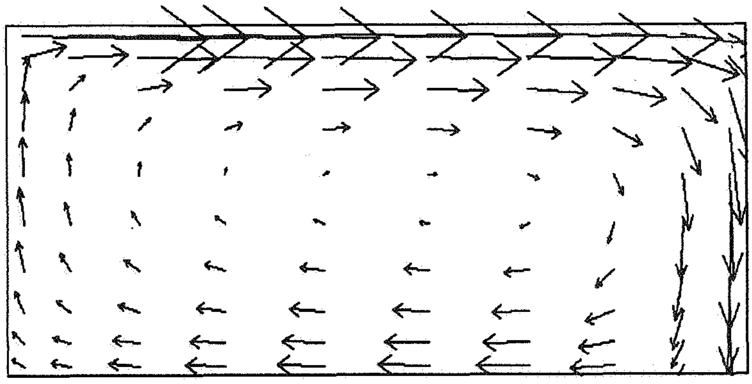

Figure 6: Velocity field - buoyancy included

Figure 6 shows the predicted velocity field. The buoyancy induced flow recirculation is not so evident, although the velocity is higher when buoyancy taken into account. For 3-D transient cases, convergence was not obtained for the case when buoyancy effects are included. The reason for this is the subject of further investigation.

4 CONCLUSIONS

A method has been implemented within ESP-r by which BES and CFD techniques are coupled. Preliminary studies indicate the advantages of this combined approach when compared with the network approach, even with relatively simple CFD models. It should be stressed that currently the main focus is on enabling future research (as opposed to achieving high accuracy now).

The results obtained thus far indicate that conflation of the two modelling approaches can be satisfactorily achieved by maintaining each method's separate solution algorithm. The two modelling approaches are connected via regions which each approach considers as its boundary condition. The overall system balance is achieved through an iterative procedure. Careful consideration has to be given to how the boundary conditions are implemented, especially for the CFD solution which is sensitive to the specifications of the inlet conditions.

It is felt that inclusion of the CFD approach in ESP-r is again a step further towards a fully integral building appraisal system.

References

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

Clarke, J.A. and J.L.M. Hensen 1991. "An approach to the simulation of coupled heat and mass flow in buildings," in *Proc. 11th AIVC Conf. Ventilation System*

Performance held at Belgirate (I) 1990, vol. 2, pp. 339-354, IEA Air Infiltration and Ventilation Centre, Coventry (UK).

Clarke, J.A., W.M. Dempster, and C.O.R. Negrao 1995. "The implementation of a computational fluid dynamics algorithm within the ESP-r system," in *Proc. 4th IBPSA World Congress "Building Simulation '95", Madison, Aug 1995*, pp. ??-??, Int. Building Performance Simulation Association, Madison, WI.

Gosman, A.D. and F.J.K. Ideriah 1976. *TEACH-2E A general computer program for two-dimensional, turbulent, recirculating flows*, Imperial College, London.

Hensen, J.L.M. 1991. "On the thermal interaction of building structure and heating and ventilating system," Doctoral dissertation Eindhoven University of Technology (FAGO).

Inard, C. and D. Buty 1991. "Simulation of thermal coupling between a radiator and a room with zonal models," in *Proc. 12th AIVC Conf. "Air Movement and Ventilation Control within Buildings", Ottawa*, IEA Air Infiltration and Ventilation Centre, Coventry (UK).

Kriegel, B. 1973. "Fallstromungen vor Abkuhlungsflachen in Gebauden und mogliche Schutzmassnahmen," Doctoral dissertation Technical University Berlin.

Negrao, C.O.R. 1994. "Combining mass flow network and CFD approaches," PhD Progress Report, University of Strathclyde, Glasgow.