BUILDING SIMULATIONS USING THERMAL AND CFD MODELS

Dr Ara Setrakian and Dr Don McLean

ABACUS Simulations Limited
Unit 106, Kelvin Campus
West of Scotland Science Park
Maryhill Road
Glasgow, UK, G20 0SP

ABSTRACT

This paper describes two simulation software packages which permit building designers to understand how buildings will perform: the ESP building energy simulation system and the ARIA Computational Fluid Dynamics (CFD) air distribution simulation system. One of the major problems with CFD code is the specification of boundary conditions for the problem. ESP can provide the boundary conditions information for the CFD airflow, simulation.

Two brief case studies are presented which illustrate the ability to provide the boundary conditions for the CFD problem from ESP.

INTRODUCTION

Today there is greater concern about the quality of the internal environment of buildings. One of the critical components in creating a high quality environment is the air quality for building occupants. In modern buildings careful and precise design of the air distribution system and knowledge of natural infiltration is required to ensure thermal comfort of occupants in all parts of the building whilst minimising capital and running costs.

There are a number of design guides and rules of thumb which may be applied for predicting room conditions produced by conventional air distribution systems. These calculation techniques can often prove to be inaccurate. In addition, where non-conventional designs are employed, the designer has in the past had to rely on data obtained from a scaled physical model of the proposed air distribution system. Modifications to this model are made until the desired operating conditions are achieved. This procedure is both very costly and time consuming, and in addition it is seldom possible to construct models with all relevant variables accurately scaled.

Recently, there has been much activity in the field of Computational Fluid Dynamics (CFD) for developing computer codes which numerically solve the Navier-Stokes equations that govern air momentum and heat transfer in enclosed spaces. In the most general terms a CFD program may be viewed as first principle solution to the Navier-Stokes equations of fluid flow for given boundary conditions. The applications are wide: in building design the techniques are already well proven and used increasingly to conduct studies on the performance of mechanical supply and extract systems; air quality; occupant comfort; smoke extraction; contamination spread; etc. Such CFD programs are proving extremely useful at all stages of the design.

One limiting factor to the accuracy of CFD solutions is the accuracy, or degree of realism, of the boundary conditions applied to the problem. When applied to internal building performance (ie room air distribution) CFD codes have typical boundary conditions corresponding to:

1. surface wall, floor and ceiling temperatures
2. air infiltration rates to and from the zone
3. mechanical ventilation rates at supplies and extracts
4. radiant gains via windows and other openings (insolation heat fluxes)

However, wall temperatures and heat fluxes are themselves a result of complex energy and mass flows which are interdependent and very often time-dependent in nature and subject to external influences e.g.

1. Climatic effects: Solar Radiation; Wind speed and direction; Temperature and Humidity
2. Operational effects such as: Occupancy gains from people, lights and equipment; and HVAC system performance
3. Building structure: heavy- or light-weight; porous walls; and transient elements (eg blinds)

Normally boundary conditions are prescribed by estimating through design experience, measurements if possible or employing simple guides like CIBSE or ASHRAE techniques.

Such methods are appropriate to provide suitable accurate boundary conditions for only the most simple designs problems. Therefore the power offered by CFD in building applications is somewhat limited by this fundamental inadequacy. However work has been ongoing for some time to provide these boundary conditions for a particular CFD package called ARIA (Setrakian 1991) from the internationally renowned building energy simulation code, ESP.

This paper describes the information transferred to ARIA from the ESP simulation and how the technique has been applied, through reference to particular case studies, to predict:

1. infiltration rate and solar gain, and their effect on the air flow pattern and temperatures inside an Atrium
2. the effect of concentrated heat inside a clean room

DESCRIPTION OF BUILDING ENERGY SIMULATION CODE (ESP)

ESP is a fully interactive, rigorous energy simulation system which is capable of modelling the energy and fluid flows within combined building and plant systems when constrained to conform to control actions (Clarke and McLean 1991). ESP is equally applicable to existing buildings and new designs, with or without advanced technological features. ESP is used throughout the European Community in the PASSYS project (Strachan and Guy 1991). Furthermore, because of its technical capabilities it is used by many of the world's leading thermal modelling Research and Development Centres.

ESP employs a Finite Volume Heat Balance technique which enables a high degree of insight into the energy flow processes within the building (Clarke 1985). This allows a better understanding of the interrelation between design and performance parameters, to then identify potential problem areas, and so implement and test appropriate building, plant and/or control modifications. The design produced is more energy conscious with better comfort levels attained throughout.

The building geometry is fully three dimensional and data can be input via data transfer from computer aided draughting packages, or, digitised by the user from architectural drawings. Once the geometry has been input it is possible to view the multi-zone model from any perspective viewpoint.

The fabric of the building is defined in terms of the thermophysical properties of the elemental layers comprising each building construction. Thus, the thermal capacity of the structure is automatically accounted for and the guesswork normally associated with estimating the response of a buildings' fabric is eliminated. During simulation the temperatures throughout the construction are calculated at each time step allowing detailed analysis of fabric performance.

In most practical instances the casual gains from artificial lighting, occupants and equipment do not remain at a constant level throughout the day. The effect of fluctuations in these loads can be accounted for by defining time dependent profiles which best reflect the changes in the load levels. In addition the radiant and convective components, plus the latent component of these gains are used to predict their influence on; internal surface temperatures, air temperature and moisture content.

In many instances the rate of air flow into and through a building is assumed to be at a constant. This default condition can be applied or a profile defining the air change rate at discrete periods for each zone can be established with thermostatic control to account for door & window opening, etc. In reality however, the air flow into/out of a building and between building zones will be dependent upon wind pressure, plant induced pressures and thermal buoyancy forces. These phenomena can be simulated dynamically by constructing an air distribution network in which all cracks, doorways, larger openings and input/extract fan operation are defined and run simultaneously for the building.

The type of heating/cooling plant employed and the mode of control can have a significant influence on the plant capacity required, running costs and comfort levels. Modern control systems employ different control regimes dependent on the time of day. ESP simulates 'real systems' by invoking either a whole building or zone specific strategies, with the time dependent variations in control philosophy included.

The type of control sensor can be defined in terms of the proportion of radiant/convective temperature sensed allowing any sensor type to be specified. Also, the location of the sensor within a zone can be specified, and all zones can be controlled by individual sensors, or by a sensor in another zone. This allows existing systems to be modelled accurately and the effects of new strategies, plant systems, sensor positioning, etc.

DESCRIPTION OF CFD CODE: ARIA

ARIA is an advanced Computational Fluid Dynamics (CFD) code that will accurately predict air flow patterns within any enclosure or external to any building (Setrakian 1991)

The physical co-ordinates of the problem, including details of windows, doors, and (internal or external) obstructions (such as furniture, other buildings, etc.) form the basis of the 3D computer model. Multiple supply/extract can be located anywhere on the surfaces of the enclosure and obstructions.

ARIA is a three-dimensional thermo-fluid analysis program which solves the Navier-Stokes equations, the enthalpy equation and the concentration equation using the finite volume method (FVM). These are the fundamental equations describing the flow and heat transfer processes of fluids. The shear stresses (Reynolds stresses), the turbulent heat fluxes and the turbulent concentration fluxes are represented by the k-E turbulence model. These partial differential equations are discretised (transformed to algebraic equations) for solution by successive iterations and relaxation techniques until a converged solution is achieved, i.e. the equations are satisfied within acceptable error bands.

A staggered Cartesian grid is used and control volumes are described for solving the three components of velocity (u,v,w), the temperature (T) and the contamination concentration (c) fields. No equations for the pressure field are necessary because the SIMPLE algorithm (Semi-Implicit Method for Pressure Linked Equations) is used to link the pressure at a grid point to the velocity of the surrounding cells (control volumes).

ESP PROVIDING BOUNDARY CONDITIONS FOR ARIA

As stated earlier, the accuracy of the boundary conditions can critically effect the results of a CFD solution. Typical CFD boundary conditions required to assess internal building performance are:

1. surface wall, floor and ceiling temperatures
2. air infiltration rates and inter-zone air flow to and from the zone
3. mechanical ventilation rates at supplies and extracts
4. radiant gains via windows and other openings (insolation heat fluxes)
5. heat flux from internal heat sources

In most circumstances the user will employ engineering assumptions to estimate the values of these parameters.

ESP is capable of dynamically simulating the coupled interaction of building structure, occupancy, HVAC systems and control against transient real weather boundary conditions. Therefore, in ESP, the surface temperatures are determined by the complex interactions of the various heat fluxes into or out of the wall. The heat flux is affected by the thermal properties of the wall; the adjacent space conditions; the radiant energy absorbed from the sun and from the equipment in the space; and the convective exchange between the wall and air. In addition, macroscopic air flow simulation is conducted simultaneously with the thermal simulation in ESP. The air flow simulation technique is a network model capable of simulating the interactions between infiltration, natural air flow and mechanical air flow. Typically the thermal model would consist of every building element affecting the CFD zone: this may mean the whole building is simulated.

Thus the boundary information can be obtained for any instant in time because ESP conducts simultaneous simulation of these processes.

Ideally the CFD program would be intrinsically coupled with the thermal program to feed back the thermal effects of the microscopic airflow regimes every time step. However, at current levels of computer power coupled simulations operation is impractical.

Practical application of the two model approach is to transfer the boundary conditions from the thermal model (using the macroscopic techniques for solving airflow) for a discrete moment in time. This may be viewed as a "freeze frame" of all values computed at a chosen time during the simulation period. The relevant wall temperatures and heat fluxes etc. can then be applied to the CFD model as boundary conditions to the solution of microscopic airflow effects for that same instant in time.

The procedure is as follows:

1. ESP is used to simulate the problem with due account taken of all air leakage points including dynamic opening and closing of these elements eg door and windows. With ESP's capability of simulating the simultaneous air and energy flow within the building it is possible to identify when the critical occurrences of either occupant discomfort, air temperature or air movement occur during a particular design scenario.

2. Having identified the time at which microscopic air flow study is required the following information is obtained from ESP and used as boundary conditions for ARIA:
   - temperature of each surface in the ESP model
   - air volume flow rates into and out of each external openings of the Atrium model eg doors and windows
   - bulk air temperatures ie temperature of each ESP zone
   - mechanical supply and extract rates at the appropriate instant in time

3. ARIA is then used to provide a microscopic study of the conditions within the Atrium predicting the detailed information on localised occupant discomfort, air movement and temperatures.

CASE STUDIES

Shopping Mall (Atrium)

The atrium is 122m x 26m x 30m high. A barrel vaulted space frame glazed roof with internal perforated aluminium shading (see figure 1). The main air handling equipment has been represented as the net supply of each occupied floor level from a linear grill. The net supply has been taken as the excess of the supply over the extract through the shops at that level. The extract fans at the base of the roof vault have been represented as having linear grilles as well. The air movement is also affected by the opening areas which exist connecting the atrium to outside. These are as follows:

- south door ($10m^2$)
- west and east doors located at the north end of the atrium ($4m^2$)
- fire louvres at low level at the south end of the atrium ($50m^2$)
- louvres at the top of the roof ($100m^2$)

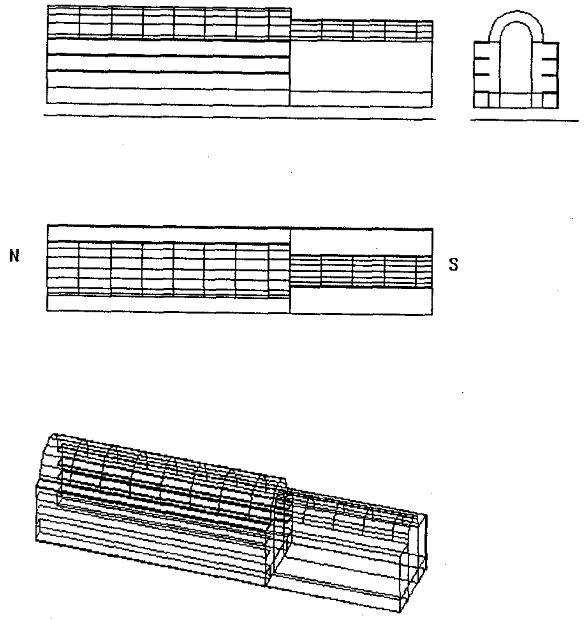

Figure 1. Perspective view of the Atrium.

Fans at the roof have been proposed to prevent the aluminium shading overheating and to ensure that air can be drawn through the mall when needed.

Three alternative design scenarios were chosen to be simulated to identify the best performance to ensure that the building occupants would be comfortable for both the winter and summer design conditions.

The three scenarios which have been studied are as follows:

1. mechanical supply and extract
2. natural ventilation with supplementary mechanical supply and extract to keep the aluminium shading plates cool
3. mechanical supply and louvres closed (atrium closed for rain)

For summer conditions, each case was simulated as a 'worst-case' scenario, ie on a hot sunny day with maximum occupancy. Consequently, from the ESP simulation the aluminium shading was predicted to be around 50°C. At the time selected there was also a breeze of 7.1m/s (force 4).

The zonal airflow networks which the boundary conditions were based on are shown in figure 2 (Sluce 1990).

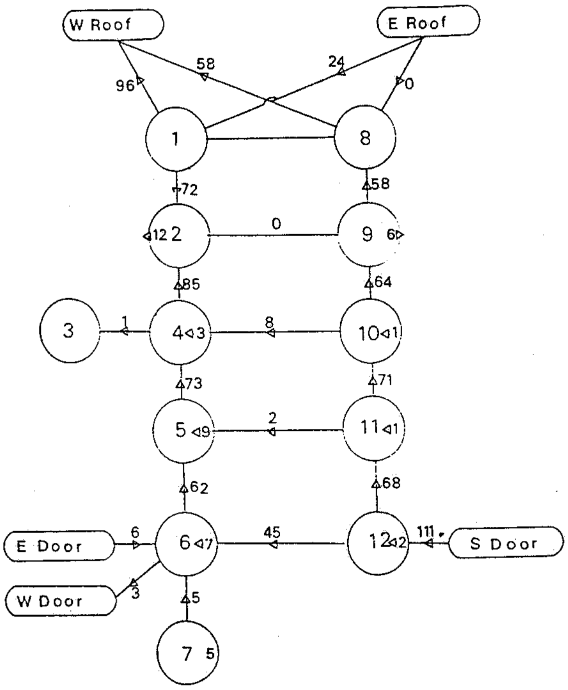

Figure 2. Zone air flow network.

In this paper we will describe only the natural ventilation during summer season, scenario 2.

Natural Ventilation

Under the prevailing conditions at the time simulated this scenario proved to be the most successful at both removing the heat from the shading devices and providing acceptable conditions at the occupied levels of the atrium.

In general an atrium does not consist of a constantly rising mass of air drawn in at low level being warmed and exhausting at high level. An effect which persists in all the modelled situations is that the air forms large vortices which tend to be driven by the velocity pressures at low level and the buoyancy effects higher up. This causes large vortices (recirculation) along the length of the atrium and also across the width in certain situations. The effect of these vortices in this model, is to pull warm air down from the high level shading devices against the natural buoyancy forces.

Wind speed, wind pressure and stack pressure act together achieve a high throughput, with wind speed in the entrance rising to 2m/s (see figure 3). As the wind speed drops the throughput will drop, and the results of the ESP zonal model at different wind speeds have been observed.

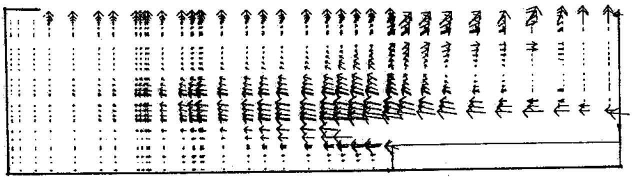

Figure 3 Velocity vector plot across the south door, velocity cut off 0-0.5m/s

With a wind speed of 3.5m/s, which is close to the average wind speed, the volume flow rate moved is equivalent to volume flow rates obtained with the full mechanical supply and extract system simulated in scenario 1.

With below average wind speed (2.1m/s) the stack effect in the afternoon will be sufficient to maintain a throughput at the level achieved in mechanical ventilation system (scenario 1). In the morning before the solar gain has heated up the aluminium shading plates the supplementary mechanical system dominates the natural forces ie wind pressure and stack pressure. The result is outside air is drawn in at the high level louvres by the fan system at the base of the roof vault. The air inside is being intercepted and the balance of pressure is such that the excess supply air at the occupied levels exhaust through the doors and louvres at low level.

With a complete calm, conditions similar to scenario 3 will be obtained. Calms mainly occurred in the night or early morning at Kew during 1967. The risk of calm conditions during the day is therefore considered to be low.

At higher wind speeds the louvres would have to be closed to reduce the wind speed in the entrance.

The ESP simulation of the Atrium predict critical boundary conditions for the ARIA problem such as the temperature of the aluminium shading plate and the air flow through doors and louvres due to natural ventilation. Without ESP the CFD user would have had to guess these boundary conditions.

Clean Room

The room is 3.9m x 3.9m x 2.75m high (see figure 4). The box in the middle of the room represents machine with 5Kw concentrated heat source. The location of the supply and extracts are also shown in figure 4.

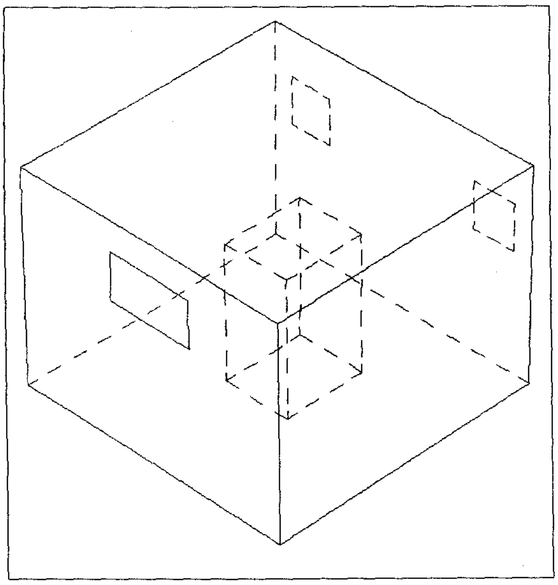

Figure 4 Perspective view of the clean room.

The object was to model the room inclusive of the machine and to predict the air velocity and temperatures within the room and to compare the results with the measurements data. This exercise will prove the accuracy and acceptability of CFD techniques as an engineering design and analysis tool.

The machine and the room were simulated using ESP and the Black Body view factor calculation facilities of the system to accurately establish the boundary conditions for the ARIA model.

The following boundary conditions were applied:

a. The proposed discharge volume flow rate from the supply diffuser (1.2m x 0.6m) with 35% free area = 0.333 m3/s giving a discharge velocity of 0.46m/s.
b. Perforated Stainless Steel sheets were simulated by porosity factor of 35% for supply and 50% for extracts.
c. The supply air temperature was 13.4°C.
d. Total heat flux was 5Kw generated by the machine.
e. The internal temperatures of the boundary surfaces were calculated by ESP to be:
   - roof = 26.5°C
   - floor = 26.0°C
   - supply wall = 27.5°C
   - extract wall = 26.3°C
   - side walls = 25.7°C

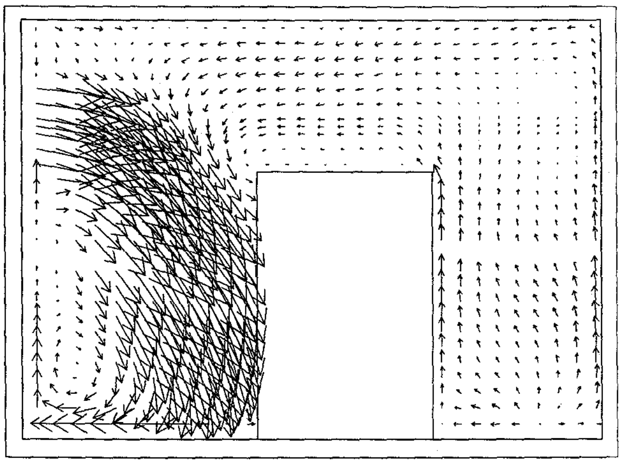

Figure 5. Velocity vector plot across the supply diffuser

Figure 5 shows the cool air dumps into the area where the machine operator would most probably be stationed. This would not normally be considered satisfactory for normal comfort conditions but the aim was to cool down the machine and disperse the heat. This flow was predicted by ARIA and the results confirmed through physical modelling.

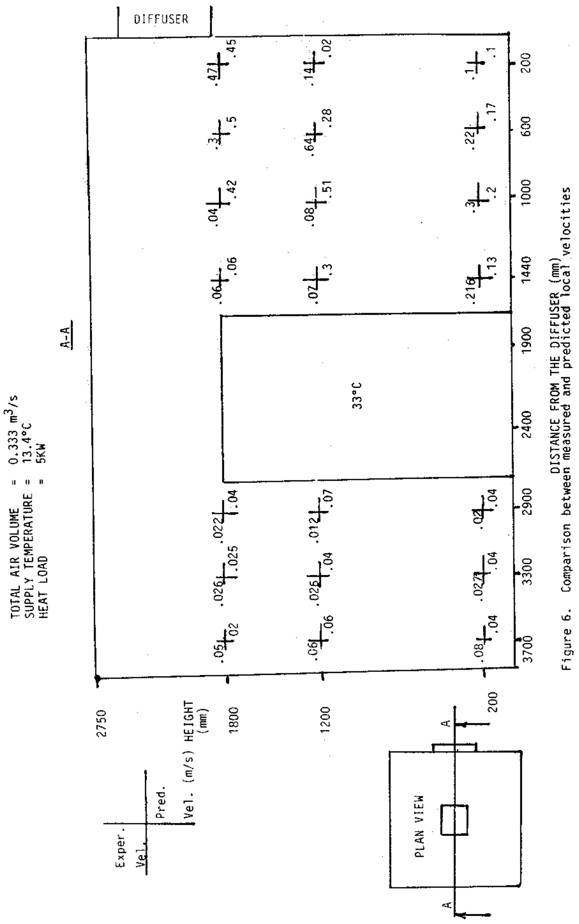

Figure 6. Comparison between measured and predicted local velocities

Figure 6 shows comparison between measured and predicted local velocities and temperatures. This cross section is through the supply and the machine (middle of the room). Predicted velocities are very close to the measured data. The accuracy of these results was considerably enhanced by use of ESP to provide the boundary conditions to the problem.

SUMMARY

The major limitation to the accuracy of CFD code in building applications is that it is difficult to define the boundary conditions to the problem. This paper has described how it is possible to obtain high quality boundary condition information by simulating the problem first by the ESP building energy simulation code.

Practical application of the two model approach is to transfer the boundary conditions at a discrete moment in time from the ESP model of the problem (which conducts simultaneous energy and air flow simulations permitting a macroscopic study of air movement within the building). This may be viewed as a "freeze frame" of all values computed at a chosen time during the simulation period. The relevant wall temperatures and heat fluxes etc. can then be applied to the ARIA model as boundary conditions to the solution of microscopic airflow effects for that same instant in time. Another advantage of transferring the boundary conditions from ESP, than guessing them, is to reduce iteration numbers (CPU time). This has been noticed during these test cases. The iteration numbers have been reduced by around 10-20% of normal simulation time.

Ideally the CFD program would be intrinsically coupled with the thermal program to feed back the thermal effects of the microscopic airflow regimes every time step. However, at current levels of computer power this coupled simulations operation is impractical except for very powerful computers. Nevertheless, the research behind the formal connections between ESP and ARIA are underway and it is hoped that a prototype will be available in the not to distant future.

REFERENCES

1. Clarke, J A and McLean, D J, 'The ESP User Manual', Available from ABACUS Simulations Limited, 1991
2. Strachan, P and Guy, A, 'Modelling as an Aid in the Thermal Performance Assessment of Passive Solar Components', BEPAC, Building Environmental Performance '91, Canterbury, 10/11 April 1991
3. Clarke, J A, 'Energy Simulation in Building Design', Adam Hilger Ltd, 1985
4. Setrakian, A A, 'The Effect of Rectangular Obstacles on the Diffusion of a Wall Jet', PhD Thesis, November 1988
5. Sluce, A J, 'The Bentall Centre - CFD Analysis of Airflow in the Mall', Building Design Partnership, Manchester, 1990