Chapter 2

CFD in Indoor Environment Design

Dr Hazim B. Awbi*

INTRODUCTION

Modern indoor environmental design is becoming a very complex process involving many disciplines: architectural design, thermal and air flow design, air quality and thermal comfort assessment, lighting design, acoustic design, etc. One of the most important and complex phenomenon in the design process is the prediction of air flow within the space and sometimes outside the building, whether one is dealing with mechanically ventilated or naturally ventilated buildings. The flow of air into a building and subsequently within the building has many implications on the quality of indoor environment, such as: thermal comfort, air quality, ventilation effectiveness and energy efficiency

Until about a decade or so ago the only air flow modelling techniques available to the designer were the wind tunnel and full scale or small scale mock-ups which were used for studying external flows and internal flows respectively. Although these techniques are still used today, they are now complemented, and in some cases replaced, by

* **Dr Hazim B. Awbi** is Senior Lecturer in the Department of Construction Management and Engineering, The University of Reading

computational fluid dynamics (CFD). CFD initiated in the aerospace and process industry but nowadays it is used in numerous applications ranging from aerospace vehicles to the blood flow in human body.

The application of CFD to studying the indoor environment started over 30 years ago (Nielsen 1974) but its use in design followed much later. The proceedings of major international conferences such as *Roomvent*, *Indoor Air* and *Ventilation* contain nowadays a large number of papers on CFD applications in research and design.

BASIS OF CFD SIMULATION

Governing Equations and Solution Techniques

A basic CFD simulation involves the numerical solution (in discretized form) of the partial differential equations for continuity of flow, momentum (Navier-Stokes equation), thermal energy, concentration of species and, when the flow is turbulent, the appropriate turbulence equations depending on the turbulence model being applied. In some applications, additional equations (differential or algebraic) may need to be solved also, such as when computing the radiation exchange between enclosure surfaces, the equation for the age of air and the thermal comfort equations. These equations are normally solved in three dimensions and sometimes in time domain too such as when fire is simulated. Further details can be found in Awbi (1998).

The discretized form of the above equations are normally solved using either the finite volume method (FVM) or the finite element method (FEM). However, the former method is more economical in computational time and more robust, hence widely used particularly in CFD programs used for indoor environment design.

Boundary Conditions

To obtain a solution for the discretized equations, boundary conditions must be specified. These are known physical quantities and empirical or semi-empirical expressions. The accuracy of the CFD solution will depend *inter alia* on the accuracy of specifying the physical quantities at the boundaries and the expressions used in linking these to the bulk fluid, i.e. the "wall functions". Typical quantities for boundary conditions are: inlet geometry (i.e. geometry of air terminal device (ATD)), inlet velocity, pressure at inlet, inlet temperature, turbulence intensity at inlet, temperature of room surfaces, heat fluxes, wall function expressions, etc. In most situations, specially when real buildings are being simulated, simplifications are made in order to be able to represent an "electronic model" of the building. In some cases, however, exact presentation of the room physics may not be feasible such as for complicated ATD's.

MODELLING TECHNIQUES

Building Model

The building geometry together with inlet and outlet openings, obstructions, sources of heat and pollution, etc. need to be specified in as much detail as possible. In general, the more detail is given the more accurate the solution would be but, at the same time, the longer the setting up and computational times would be. In many practical situations some building detail may be reduced or ignored completely without jeopardising the accuracy of the solution. Therefore, each situation must be considered individually and good judgement in translating the real building into an "electronic building" which is suitable for a CFD solution can only be exercised after considerable modelling experience by the user.

Depending on whether external (outside the building), internal (inside the building) or both types of flow are to be modelled, the geometrical details are selected accordingly. In large buildings there may be a difficulty in representing minute but nevertheless important geometrical detail because of the large computational grid which would be required. The capacity of the computer and computational time is directly related to the grid size being used to represent geometric details. As an example, for a 3D problem doubling the number of grid in each co-ordinate direction would increase the computer memory and the time required for solving the problem by approximately 8 times.

On the internal and external surfaces of the building, boundary conditions for all the scalar variables to be solved (e.g. temperature, heat flux, pollution source emission, etc.) must be specified. The vector quantities (i.e. velocity components) are not normally required because non-slip conditions usually exist. CFD programs have "wall function" expressions built in them for "bridging" the surface conditions with the flow field. For external flows, conditions at the free boundaries will need to be specified such as pressure, velocity and temperature.

Modelling Obstructions

In both internal and external flow modelling, the obstructions in the flow field can have a major influence on the flow. In external flow, common obstructions are adjacent buildings, trees, land topography, etc. In internal flow, people, furnishing and room fixtures (such as partitions, structural elements, lights, etc.) could have an important influence on the air flow in the space.

In addition to the space they occupy within the flow field, some obstructions also represent a source of mass flow, heat, pollution, etc. which must be represented when specifying the boundary conditions. Boundary conditions for obstructions are specified in the same way as building surfaces.

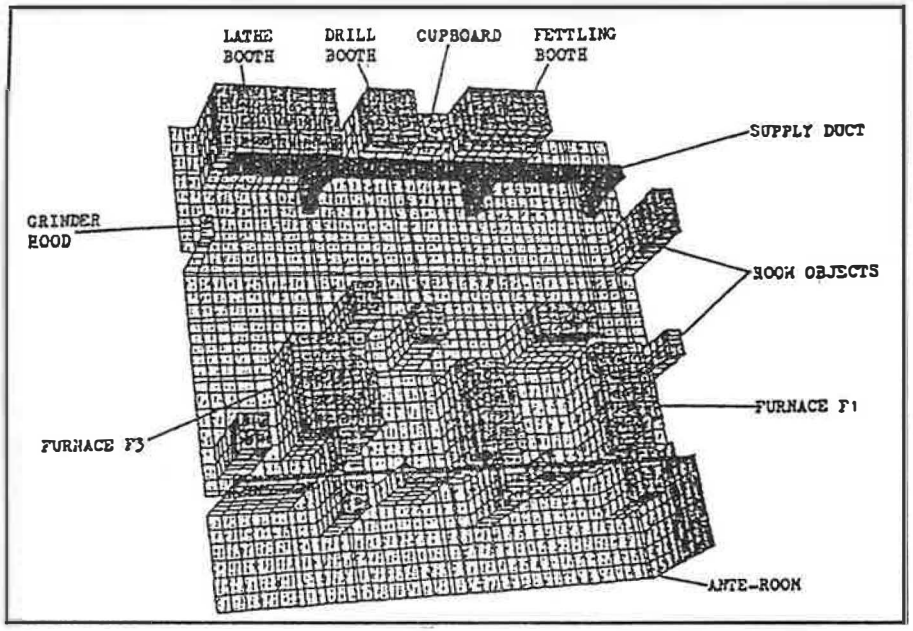

Figure 2.1 **A cartesian co-ordinate grid for a large industrial building**

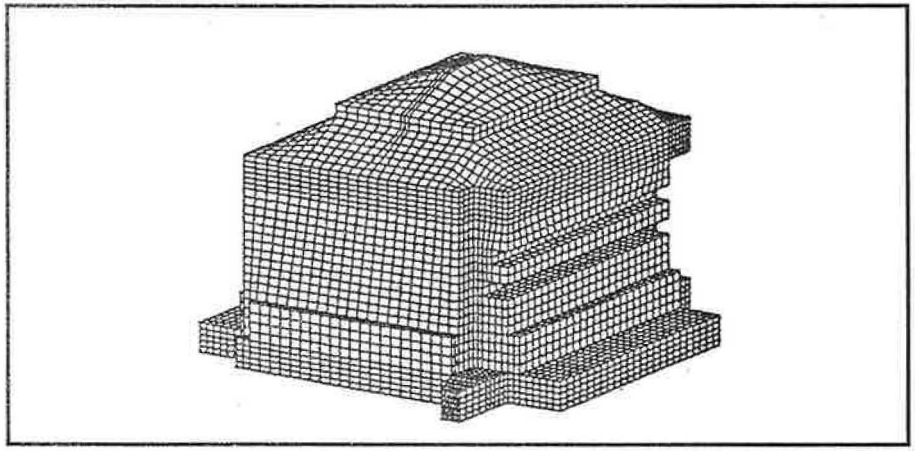

Figure 2.2 **A body-fitted co-ordinate grid for an atrium building** (Renz and Vogl 1996)

Modelling Air Terminal Devices

One of the most challenging tasks in CFD modelling of room air movement is the representation of the geometric and flow conditions of the air terminal devices, such as grilles, diffusers and hoods. Real ATD's not only have complex geometries but the flow

field is usually also complex and in most cases non-uniform. This invariably lead to simplifications being made of the geometry and the flow from these devices to enable a CFD solution of the room flow. Some methods which are used are described in Awbi (1990) and Skovgaard and Nielsen (1991). However, modelling of ATD's used in displacement ventilation (such as the perforated wall diffuser or the swirl flow floor diffuser) is still a major obstacle in CFD modelling.

Assuming that the geometry of the ATD is properly represented, the modeller has to also specify the velocity (i.e. the jet momentum), temperature, concentration of species and turbulence intensity of the air supply.

Computational Grid

After specifying the building geometry, obstructions, inlet and outlet openings, etc., the whole flow domain has to be fitted with a computational grid or cells. The discretized equations are then iteratively solved over the whole domain recalculating at every sweep the values at the first grid points from the boundary using the specified "wall functions". It is therefore essential that these points are located very close to the surface, usually only few mm away.

There are generally three types of gird systems that are in use: Cartesian co-ordinates (CCs), body-fitted co-ordinates (BFCs) and unstructured grid, Figures 2.1, 2.2 and 2.3. CCs specify either a uniform or non-uniform mesh in the x-, y- and z-directions to suit the geometry and physical parameters in the model. Usually a non-uniform grid is preferred because of its flexibility in resolving critical flow domains (such as air inlets and outlets) more accurately. The CC system is the simplest to apply and still the most widely used in indoor environment simulations.

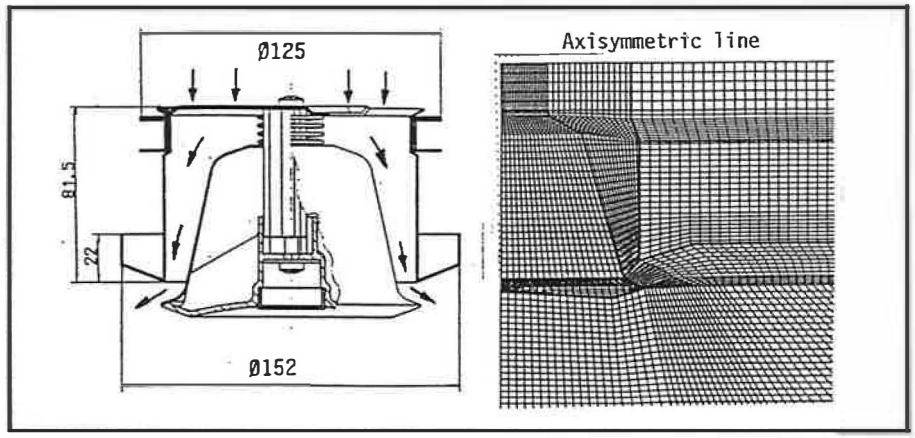

Figure 2.3 **An unstructured grid for the flow in a bell-shaped diffuser** (Chen and Jiang 1996)

The BFC grid is useful for resolving complex geometrical shapes by fitting the grid around the shape usually by comformal mapping. However, the solution is less efficient in computing terms than the CC grid and could sometimes lead to difficulty in reaching a converged solution of the discretized equations. In the unstructured grid system, the geometry of the flow domain is built up with blocks which are then connected together to cover the whole flow domain. Within each block however a different grid is specified. This method gives much more flexibility for representing complex geometries. However, the computational time for solutions using unstructured grid is considerably greater than the CC grid and even the BFC grid.

Interpretation and Presentation of Simulation Results

It was mentioned that the CFD modeller has to exercise skills in modelling the building but he/she also has to show similar skills in interpreting the CFD results. Because a CFD solution normally generates huge amount of data, it is extremely tedious to inspect all the data. Instead, most CFD packages incorporate a graphical post-processor to visualise the results and generate plots in the form of vector diagrams for velocity, contour and fills for the scalar variables such as pressure, temperature, pollution concentration, etc. In addition, summary results are sometimes given for the key input and output parameters for reference.

From the graphical and numerical data the designer should then be able to assess the environment in or around the building and make the necessary decisions for designing the heating, cooling or ventilation system.

Advantages and Disadvantages of CFD Simulation

Air flow modelling can be carried out using one of the following techniques:

- Physical models with air (full scale or reduced scale) or water (reduced scale) - if water is used then buoyancy may be represented by creating differences in the water density using salt solutions of different concentrations.
- Flow element models in which case the room is divided into different flow regions, such as supply air jets, exhaust flow, thermal plumes, boundary layer flows, gravitational currents, etc. and a relevant theoretical model which describes the flow in each region is assigned.
- CFD simulation.

There are many advantages for using CFD over the other two techniques such as:

- lower investments than in physical modelling
- CFD generate results for a full-scale building and there is no uncertainties associated with scaling as there are with most physical models

- CFD is much faster than physical modelling
- unless there are known empirical expressions for each region, flow element models will not produce accurate results which is not the case with a CFD simulation since the fundamental flow and energy equation are solved in full
- CFD can provide considerably more detail than the other two methods
- CFD offers more scope for considering different design scenarios very quickly.

However, there are also some disadvantages associated with CFD, namely:

- the limitations of most turbulence models used in CFD codes
- difficulties in the representation of complex geometries and ATD's
- certain difficulties in dealing with buoyant flows such as plumes
- the need for experienced user of the CFD code.

TYPICAL APPLICATIONS

Wind Flow over Buildings

The wind flow around a building is studied to determine the pressure distribution around the building or the wind flow pattern. The pressure distribution is used to estimate the

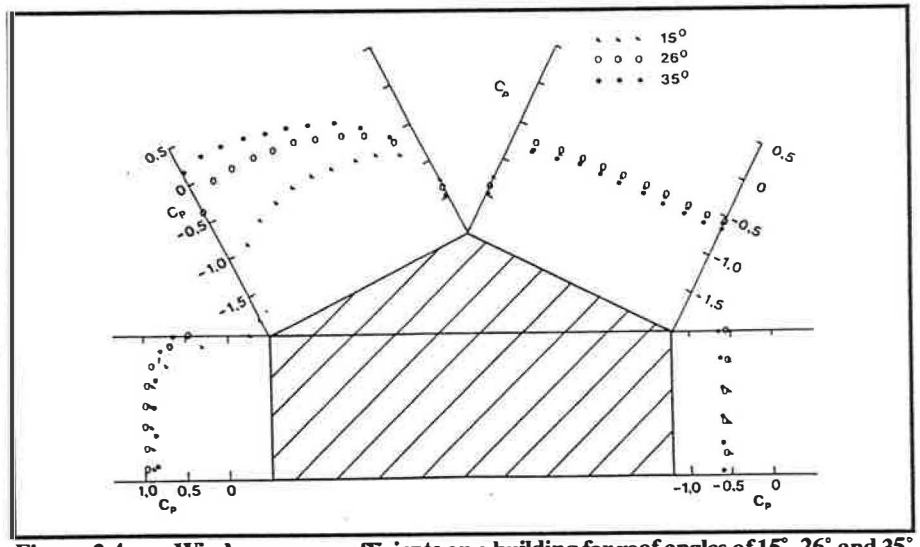

Figure 2.4 Wind pressure coefficients on a building for roof angles of 15°, 26° and 35°

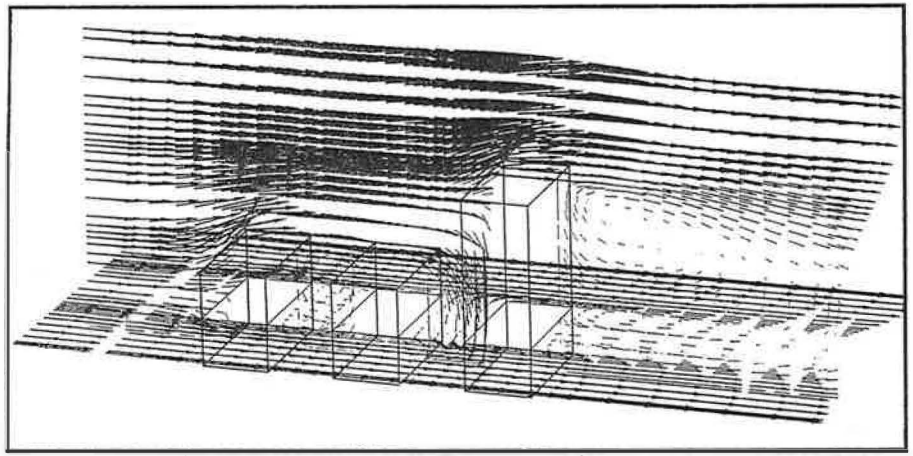

Figure 2.5 Wind flow patterns around a group of three buildings in tandem.

wind loading over a structure or for sizing and positioning openings on the building envelope in naturally ventilated buildings. The wind flow pattern is required by the building designer to study the wind flow at street level and the effect of adjacent buildings on the wind flow. CFD has been used for both purposes and some examples are given below.

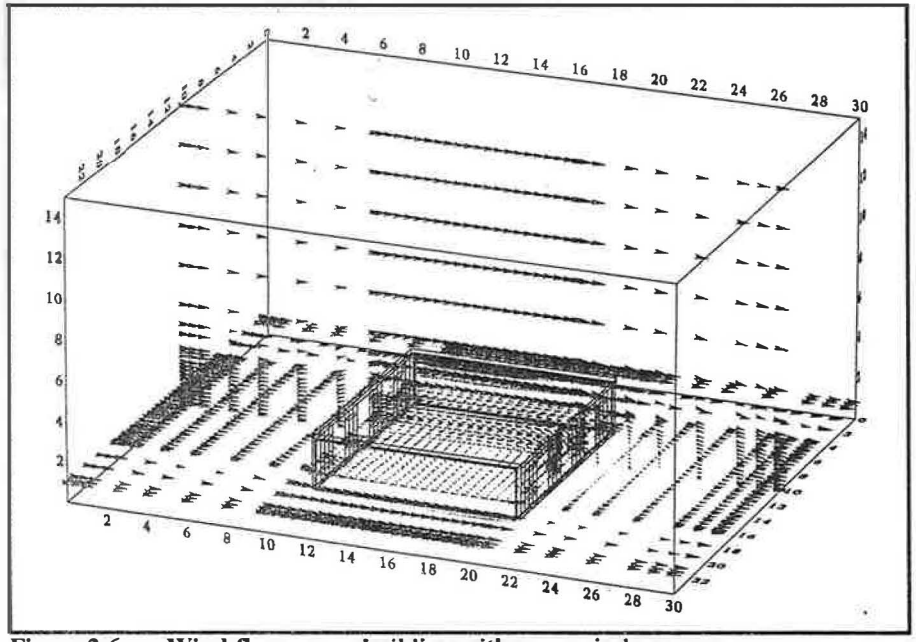

Figure 2.6 Wind flow over a building with open windows.

Figure 2.4 shows the distribution of wind pressure coefficients (Cp) around a house for different roof angles (Mathews, *et. al.* 1988). Figure 2.5 shows three buildings in tandem to the wind and velocity vectors in a vertical and a horizontal planes showing the stationary vortex between the two small buildings, the downwash in front of the tall building and the flow separation from the top of tall building.

Figure 2.6 shows a velocity vector diagram in a vertical and a horizontal plane 1.0m above the ground of the wind flow over a building with open windows in the windward and leeward side. The flow field inside and outside the building is shown.

ROOM AIR FLOW

Ventilation Strategy

The design of mechanical and natural ventilation requires a knowledge of the air flow in the ventilated space. There are two ventilation strategies which are usually used in commercial or residential buildings, *viz.* Mixing (Dilution) Ventilation and Displacement Ventilation. The principle behind the former is the mixing of a large momentum (and velocity) supply air jet with room air and that of the latter is the displacement of room air by a low momentum supply jet and plumes from heat sources in the room. Numerous

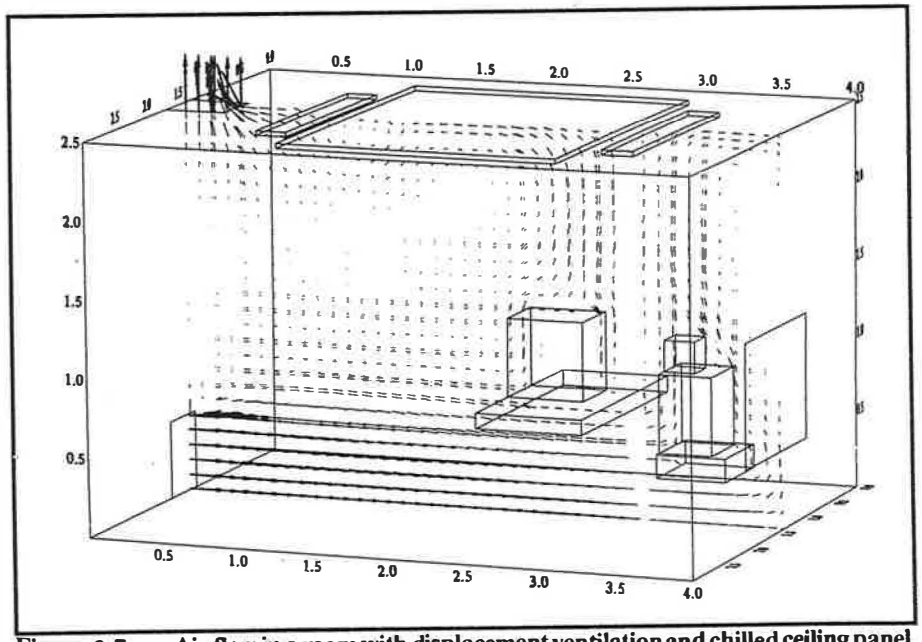

Figure 2.7 Air flow in a room with displacement ventilation and chilled ceiling panel.

CFD codes have been used in the last two decades for predicting the air velocity and temperature distributions in buildings ventilated by mixing and displacement systems. Figure 2.7 shows a velocity vector plot for a small office room ventilated using a displacement system with a chilled ceiling panel. The plumes from the block representing a computer and the two blocks repressing the torso and head of a person are clearly noticeable.

Thermal Comfort Prediction

Full thermal comfort analysis requires not only the air velocity and temperature distribution in the occupied zone but also the mean radiant temperature distribution. Most general purpose CFD codes are not capable of full comfort analysis because they do not have a

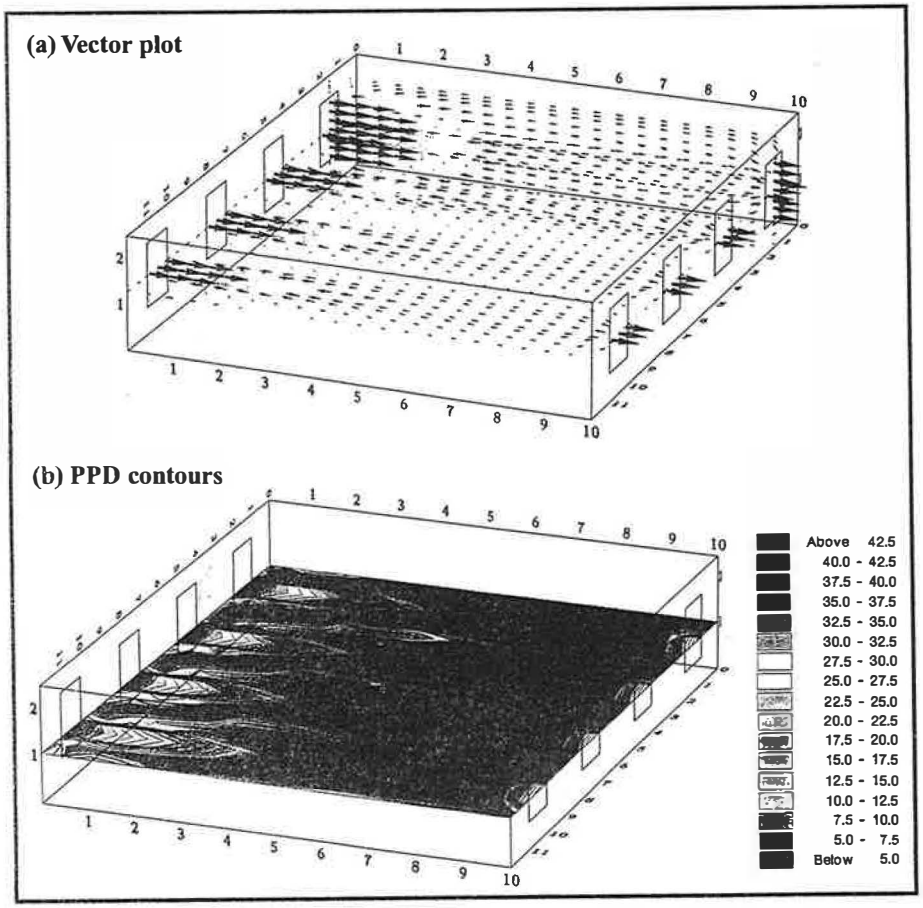

Figure 2.8 **Thermal comfort prediction in a naturally ventilated room**

facility for surface-to-surface thermal radiation exchange. In this case, the user has to use the temperatures of the room surfaces to calculate an average radiant temperature for the whole room and use this in the comfort analysis. However, full comfort evaluation, such as PMV and PPD predictions in the space, can be obtained using codes with radiation heat exchange and thermal comfort analysis, e.g. the VORTEX code (Gan and Awbi 1994).

Figure 2.8(a, b) shows the velocity vectors and the corresponding PPD contours in a horizontal plane 1.0 m above the floor of a room which is naturally ventilated by cross-ventilation through window openings. This simulation was obtained using VORTEX with radiation calculations and Fanger's thermal comfort equation was calculated for each computational cell in the room. It can be seen in Fig. 2.6(b) that the PPD is high near the windows where the air speed is large.

Indoor Air Quality

Most CFD codes are capable of analysing the diffusion and convection of pollution in a building by solving the concentration of species equation. Few research codes are also capable of predicting the "local age of air" distribution in a ventilated room. This is the residence time of the supply air at any point in the room which is a useful parameter for studying the air quality and air exchange rate at different locations in the building. Figure 2.9 shows the age of air contours calculated under the same conditions as for the room shown in Fig. 2.7. The plumes rising from the simulated computer and the person has low age values because fresh air from the floor is entrained by the two plumes.

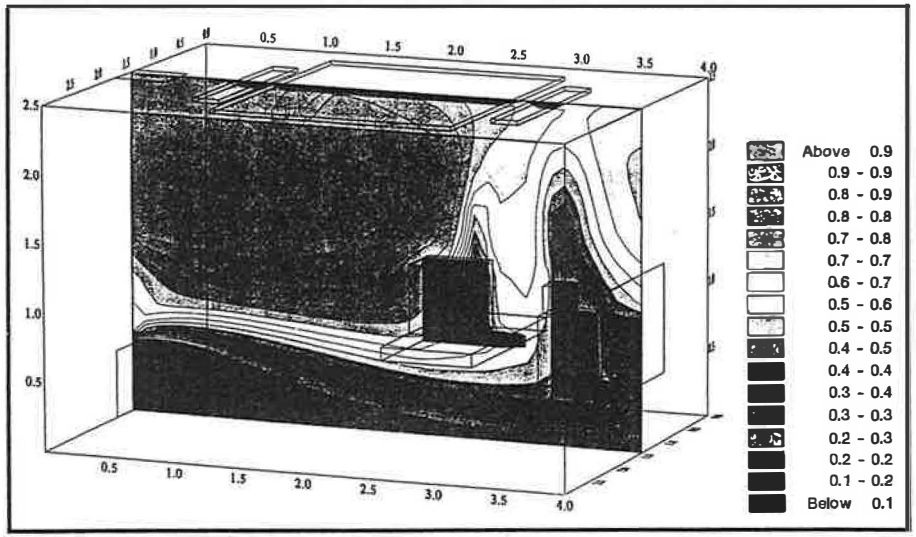

Figure 2.9 **Age of air distribution in a room with displacement ventilation and chilled ceiling**

Fire Spread

CFD has been used for simulating the fire and smoke spread in buildings for the purpose of positioning evacuation routes. However, this requires solving all the governing equations in the time domain, i.e. a transient solution. Figure 2.10 shows the spread of fire in an atrium building with two doors open at each end of the building and a fire extract in the roof. The figure shows the velocity vectors and the temperature distribution after 180 seconds from the start of a 4MW fire on the floor.

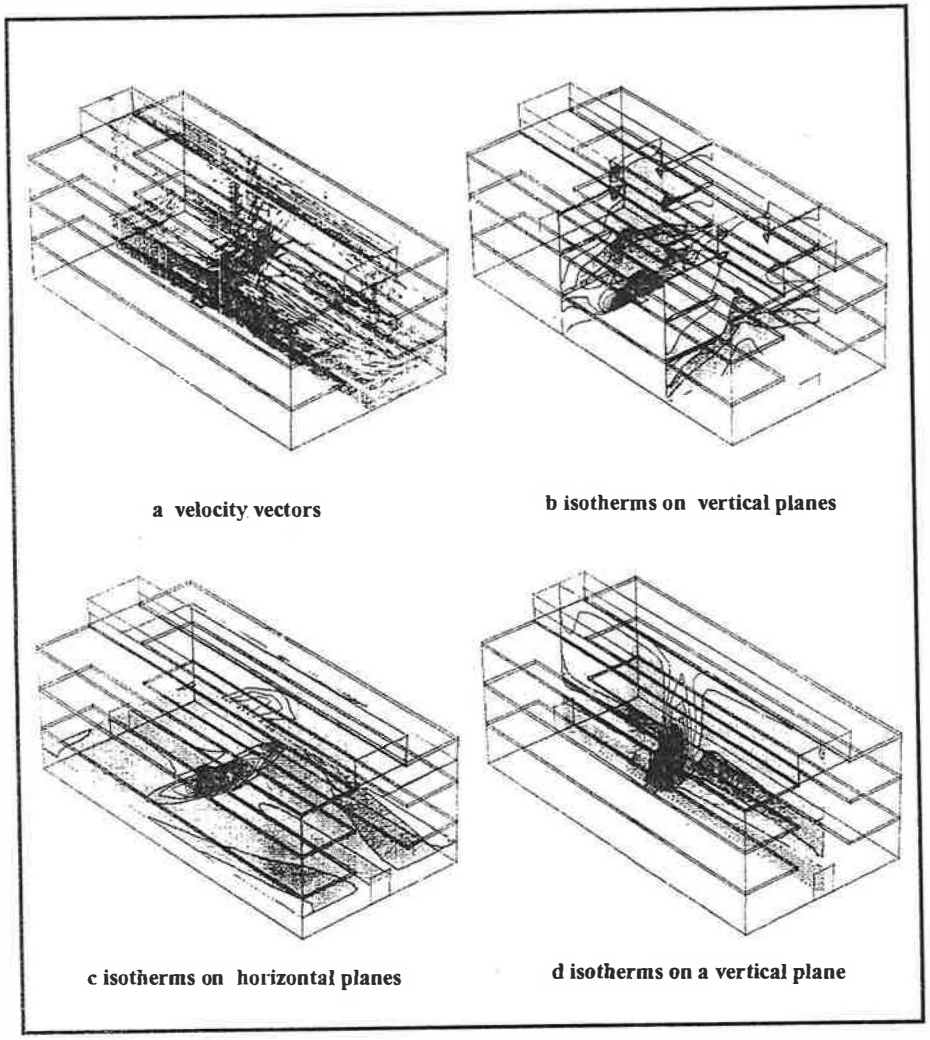

Figure 2.10 Fire spread in an atrium building after 180 s from the start of fire.

CONCLUSIONS

Room air movement prediction is an essential requirement for the design of natural or mechanical ventilation systems and CFD has a very important role to play in this process. CFD techniques for indoor environment prediction has been developing rapidly during the last ten years and this is now a recognised technique in this field. With the vast expansion in computing power and data storage, we can nowadays conduct an air flow simulation for complex buildings and boundary conditions. There are, however, certain problems which still exist, such as the simulation of the air flow in air diffusion terminal devices and the accurate calculations of the heat transfer from building surfaces and in plumes produced by heat sources. Such problems will need to be resolved if the building and system designer is to rely solely on CFD predictions.

REFERENCES

Awbi H.B. (1990), 'The role of numerical solutions in room air distribution design' Proc. *ROOMVENT '90*, Session A1, Paper 2, Oslo, Norway.

Awbi H.B. (1998), '*Ventilation of Buildings*', Spon, London.

Chen, Q. And Jiang, Z. (1996), 'Simulation of a complex air diffuser with CFD technique', *Proc. ROOMVENT '96*, **1**, 227-234.

Gan, G. and Awbi, H.B. (1994), 'Numerical simulation of the indoor environment, *Building and Environment,* **29**, 449-459.

Mathews, E.H. *et. al* (1988), 'Numerical prediction of wind loads on buildings', *J. Wind Eng. Ind. Aero.*, **31**, 241-250.

Nielsen, P.V. (1974), 'Flow in Air Conditioned Rooms' *PhD Thesis*, Technical University of Denmark.

Renz, U. And Vogl, N. (1996), 'Numerical prediction of air flow patterns in large enclosures, *Proc. ROOMVENT '96*, **3**, 163-170.

Skovgaard, M. and Nielsen, P.V. (1991), 'Modelling complex inlet geometries in CFD - Applied to air flow in ventilated rooms', Porc. AIVC Conference, Ottawa, Canada.

Mention
in Air
if space
for Mar 01

INDOOR ENVIRONMENT *AND* AIR QUALITY

Editors **Richard H Rooley** *FEng* & **Dr Alan Sherratt**