5707

Applications of computational fluid dynamics in building services engineering

A A SETRAKIAN, MSc, PhD, AMIMechE, D J McLEAN, BS, PhD,
D A MORGAN, BEng, AMIMechE, DipM, AMCIM
Abacus Simulations Ltd, West of Scotland Science Park, Glasgow, UK.

SYNOPSIS

This paper describes the advantages of using a customised Computational Fluid Dynamics (CFD) software program designed for use in building services engineering. The minimum requirements of a building services CFD program are detailed. Case studies are also presented as example results of having such a code.

1 INTRODUCTION

Computer Fluid Dynamics (CFD) programs are powerful design tools that can be used to predict the behaviour of fluid flow regimes. Their applications are extensive and in the field of building services engineering the techniques are already proved and their use increasing.

CFD programs are used increasingly to conduct studies in performance assessment of mechanical supply and extract systems; air quality; occupant comfort; smoke extraction; contamination spread etc. Such CFD programs are proving useful at all stages of the design, offering increased accuracy and cost effectiveness over empirical and scale modelling techniques.

In the most general terms a CFD program may be viewed as the first principle solution to the Navier-Stokes equations of fluid flow for given a boundary condition. There are many CFD programs in existence and these vary extensively in content. The majority of CFD programmes have evolved from different origins each being design to solve a particular physical phenomena, e.g. Combustion and Aerodynamics. Therefore these models exhibit different strengths and weaknesses in their applicability to the selection of particular fluid flow phenomena. The minimum requirements of a specialized CFD program for use in Building Services applications are detailed in first section of this paper.

2 MINIMUM REQUIREMENTS OF A CFD SYSTEM SPECIFICALLY DESIGNED FOR BUILDING SERVICES APPLICATIONS.

There are many CFD programs in existence, these vary greatly in complexity and field of applications. This paper outlines the minimum requirements of a CFD system specifically designed for building services applications, supported by case studies performed by such a specialised system "ARIA".

2.1 Capability

In the majority of building service applications, the flow regimes within buildings are non-laminar, and can have complex flow pattern even in the simplest symmetrically shaped buildings. Therefore the software must be capable of solving Navier-Stokes equations of fluid flow for turbulent, three dimensional regimes.

The K-E Turbulence model (1) is currently the best model available for solving elliptic flow problems (i.e. flows with large recirculation zones).

2.2 Boundary Conditions

The major limiting factor to the accuracy of a CFD solution is the quality of the information relating to the boundary condition applied to the problem. When applied to internal building performance (i.e. 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. inter-zone heat and mass transfer between zones
4. mechanical ventilation rates at supplies and extracts
5. 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)

Boundary conditions are attained through physical measurement or predictive calculation. Physical measurement is suitable for existing buildings and offers high quality information for the measured period. However this is time consuming and often not possible due to the operational constraints of the building. In addition this inforamtion can not be used to predict the consequences of a proposed design change. Alternatively predictive calculation methods offer boundary condition values for existing buildings with or without design changes and new buildings. The level of accuracy of predictive calculations vary greatly and only Dynamic Building Energy Simulation offers the integrity and accuracy required for CFD solutions.

2.3 Interface

The interface should minimize effort required in using the CFD system. For a specialized building services CFD system, the user interface dialogue should be expressed in a language familiar to the engineer (e.g. physical boundaries expressed as walls, ceilings, windows and doors etc). Inlets and outlets expressed as diffusers and grills etc.

The interface should be capable of importing external geometrical data from CAD systems and boundary conditions from building energy models. This greatly increases productivity and avoids repetition of data.

The mesh generation, necessary in describing a solution domain requires knowledge of CFD numerical techniques appropriate to the particular problem to be solved (i.e. fine mesh in areas of step gradients, recirculation zones and boundaries etc). However the interface should facilitate the correct mesh refinement where appropriate, substantially reducing the user's effort in generating the specific meshing.

Owing to the quantity and complexity of the results produced by the CFD system, the results should be presented graphically. Scaler variables can be present as contours (annotated or colours) and vector variables, as scaled vector arrows.

2.4 Hardware

The level of accuracy depends on the discretisation (2) of the partial differential equations (Navier-Stokes) for solution by successive iterations and relaxation techniques until a converged solution is achieved; i.e. the equations are satisfied within acceptable error bands.

Therefore the minimum requirement for CFD Systems are:

- Bit-mapped graphics terminal.
- A 32 bit (or greater) micro/mini-computer or mainframe capable of supporting up to 4 MBytes of Memory.
- Hard disk (not floppy) with capacity in excess of 100 MBytes.

3. Case Studies

In the majority of Environmental Analysis Design Studies where Computational Fluid Dynamics (CFD) programs are applied, measured data is often unavailable. Therefore a significant contribution is made to the design of HVAC and environmental systems by the use of CFD programs. CFD programs contribute to the prediction and analysis of many fluid flow phenomenal including optimisation of inlet temperatures, flow rate, position of extracts etc.

Two airflow simulations are presented here to predict the buoyancy driven airflow regime inside a room, using 3 dimensional solutions.

3.1 Test room

The test room dimensions are 3.9m x 3.9m with a ceiling of 2.75m. The box in the middle of the room represents a machine with a 5Kw concentrated heat source. The high-level supply grille and two extract diffusers are located opposite each other (see figure 1).

CFD - ARIA

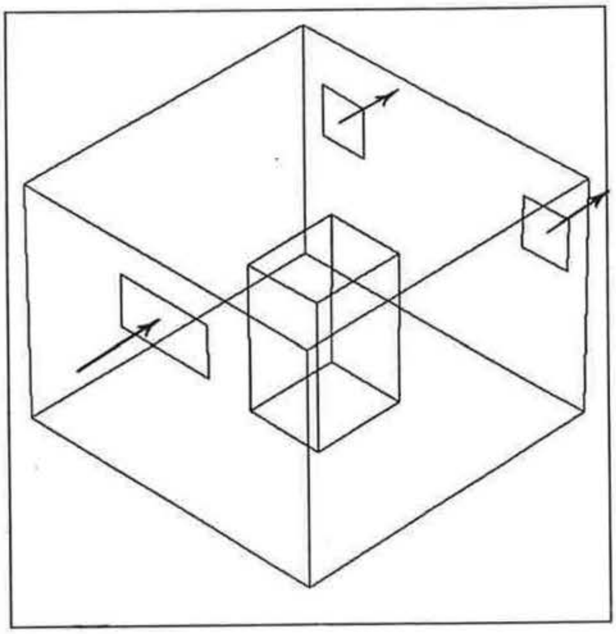

Fig 1 Perspective view of test room

The objective of this investigation was to predict the air velocity and temperatures within the room and to compare the predicted results with the measured data (3). This exercise proves the accuracy and acceptability of Computational Fluid Dynamics (CFD) techniques as an Engineering design and analysis tool.

Before predicting the air movement inside the room, the room was simulated thermally using ESP (4) to accurately establish the boundary conditions for the ARIA (5) 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 = 333 l/s giving a discharge velocity of 0.46 m/s.

b. Perforated stainless steel sheets were simulated by porosity factor of 35% for supply and 50% for each extract.

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

Velocity vector plots of the flow pattern predicted by ARIA show that cool air dumps into the area where the machine operator would most probably be stationed (see figure 2). This would not normally be considered satisfactory because it does not achieve the design objective of cooling down the machine and in addition it will cause operator discomfort due to the air dumping.

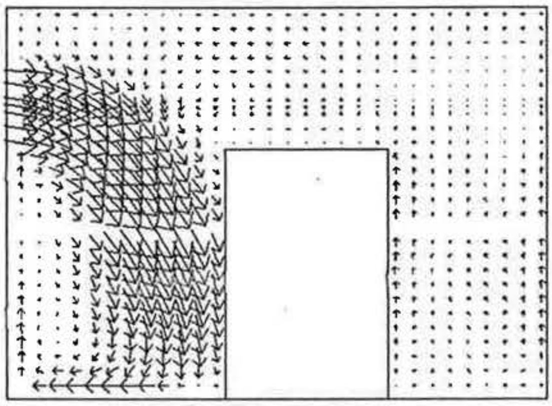

Fig 2 Velocity vector plot across the supply diffuser

Non-toxic, non-irritant and non-contaminant smoke was used in the physical visualisation of airflow patterns for the experimental test (6). The observations and measured data show close agreement with the results of the CFD solution from ARIA (7). The comparison between measured data and predicted results across the supply diffuser and the machine is shown in figure 3.

This exercise proves that CFD solution can provide information which can be used to complement data obtained from physical model tests and it be used as design aid to predict the behaviour of airflow regimes.

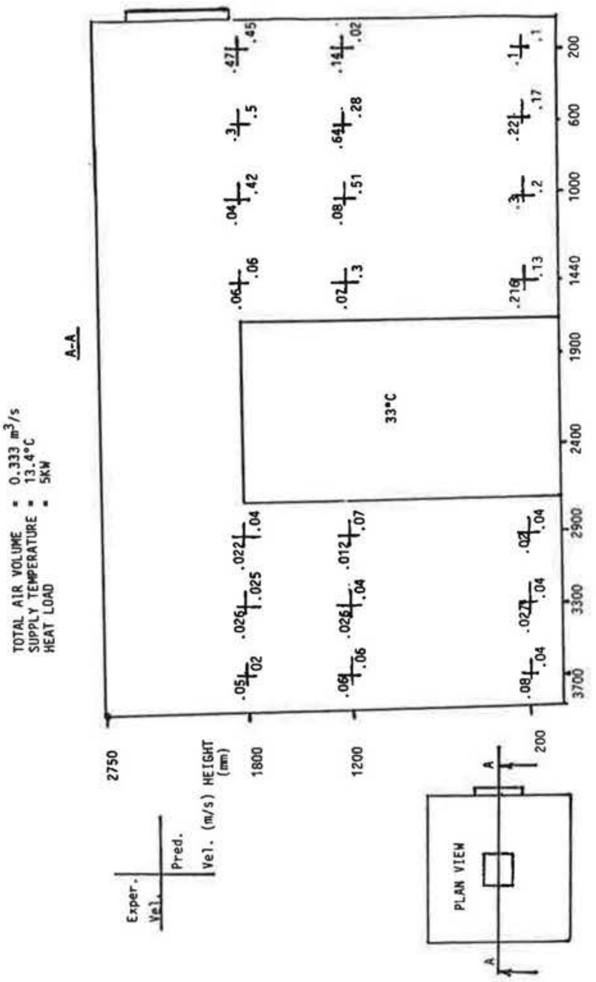

Fig 3 Comparison between measured and predicted local velocities

3.2 Displacement Ventilation

Displacement Ventilation systems have proved to be an efficient means of removing contaminants and excess heat. In this system air is supplied to the room with low velocity through openings close to the floor at a room air temperature. The buoyancy effect from items such as occupants and equipment will influence the flow pattern inside the zone.The concept of Displacement Ventilation also means that the air near the floor is driven by the buoyancy forces acting on the supply air due to the low velocity.

This case study shows how CFD techniques were used to predict 3D airflow patterns inside a modern office using Displacement Ventilation.

The office dimensions are 4.25m x 3.0m tapering to 2.6m with a ceiling height of 2.65m. The obstructions on the right hand side of the office represents a table, personal computer and a person sitting beside the table. The low level supply grille and two high level extract diffusers are located on the same wall (see figure 4). The opposite wall was assumed to be a double glazed window. A radiant cooling element was used in the summer to absorb the heat from the office.

CFD - ARIA

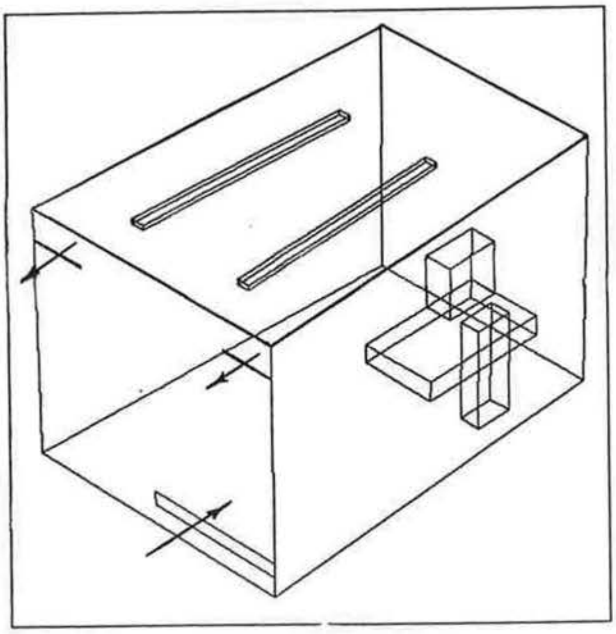

Fig 4 Perspective view of displacement ventilation

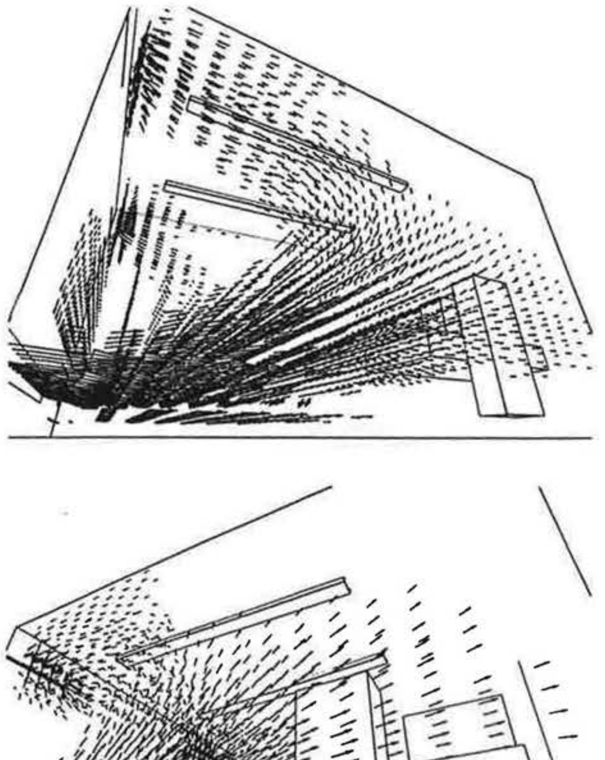

Fig 5 3D velocity vector plot from three different angles

The radiant cooling element is attached to the ceiling near the window (not shown in figure 4). The object of the Summer simulation was to predict the air velocities and temperatures within the office space, and to predict the effect of the window and radiant cooling element on the air pattern and comfort in the occupied zone.

To establish the boundary conditions accurately for the ARIA (CFD Model), the office was simulated by ESP taking full account of the casual gains from the occupant, Lighting and personal computer.

The following boundary conditions were applied:

a. The proposed discharge volume flow rate from the supply diffuser was 80 m3/h giving a discharge velocity of 0.1 m/s.

b. The air supply temperature was 20°C (Summer), and 24°C (Winter).

c. Total heat flux from Lighting was 35 W/m².

d. The internal temperatures of the boundary surfaces were calculated by ESP to be.

   - window = 30°C (Summer), and = 14.5°C (Winter)

   - radiant cooling temperature = 17°C (Summer only)

   - all other surfaces approximately 26°C (Summer), and approximately 22°C (Winter)

Figure 5 shows 3D velocity vector plot from three different angles. The fresh air entered the room at low level 0.2m above the floor. The jet attached to the floor due to the Coanda effect at first instance, and then started rising as the air heats up (buoyancy effect) causing separation from the floor. Separation line is somewhere in the middle of the room.

The predicted airflow pattern by ARIA (see figure 5) shows the fresh air entering at low velocity (less than 0.1 m/s) and at room air temperature. This created a very comfortable and pleasant environment inside the room. The effect of large glazed windows was compensated by radiant cooling from the ceiling in Summer and by increasing the supply temperature in Winter to 24°C.

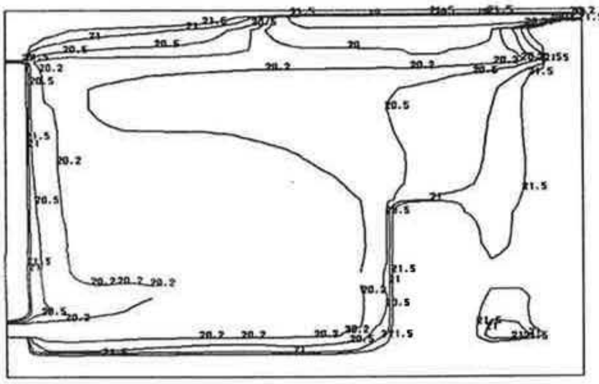

Fig 6 Temperature contours across the supply diffuser

Figure 6 represents temperature contours across the supply diffuser (front elevation). A reasonably uniform temperature; with only a small temperature gradient was achieved by the displacement ventilation system.

As a result of using CFD technique to predict airflow patterns and temperature distributions it was possible to identify the cause of thermal discomfort in the room with displacement ventilation and to improve the comfort level.

4. Conclusion

Conventional design methods progressing from Scale models to prototypes are expensive in cost and resources. Measurement data or empirical calculations may prove sparse, such as air movement, heat transfer or turbulence and may not be represented in these calculations.

Computational Fluid Dynamics Simulation, by solving fundamental fluid flow equations, offers a cost effective alternative. The case studies presented in this paper demonstrate the potential of a CFD technique in the design of air distribution systems. The cases considered in this paper, shows, the prediction of air flow patterns in a room of both negative and positive buoyancy forces. The predicted results in first case study show good agreement with measured data at local Velocities and the flow patterns.

The Case Studies shown in this paper were taken from a design office and performed under usual conditions and pressures of a working office.

This paper has shown that a CFD program can be a powerful tool in building services design.

5 REFERENCES

(1) LAUNDER, B. E. and SPALDING D. B. The Numerical Computational of Turbulent Flows Comput. Methods. Appl. Mech and Eng, 3, 1974, pp. 269-289.

(2) PATANKAR, S. V. and SPALDING D. B. A Calculation Procedure of Heat, Mass, and Momentum Transfer in Three Dimensional Parabolic Flows, Int. J. Heat Mass Transfer, 15, 1972, pp. 1787-1806.

(3) SETRAKIAN, A. A. and MCLEAN D. J. Building Simulations using Thermal and CFD Models. Building Simulation '91, NICE, FRANCE 1991.

(4) CLARKE, J. A. Energy Simulation in Building Design. Adam Hilger Ltd, 1985.

(5) SETRAKIAN, A. A. The Effect of Rectangular Obstacles on the Diffusion of a Wall Jet. PhD Thesis, November 1988, Mechanical Engineering Department, Napier Polytechnic of Edinburgh.

(6) TROX TECHNIK, Test Report Number AD235, July 1989.

(7) ABACUS Simulations Limited, Airflow Study, Report Number AC5169/AS, May 1990.