CFD AS A BUILDING SERVICES ENGINEERING TOOL

C.A.J. Fletcher*, I.F. Mayer and A. Eghlimi
CANCES, University of New South Wales, Sydney, Australia

K.H.A. Wee
CFD Research (S) Pte Ltd, Science Park II, Singapore

(Received 17 May 2001; Accepted 1 August 2001)

ABSTRACT

The use of computational fluid dynamics (CFD) as part of the design of new buildings and assessment of existing buildings is compared with its use for research into such areas as ventilation, thermal comfort, contaminant transport and fire safety. The very severe demands on project time and cost make the setting up (pre-processing) of the engineering problem and the (visual) assessment of the flow solutions as important as the solution capability of the CFD software. Through five examples ranging from photocopier emissions to heat stacking effects in air wells, the way that CFD is used as an engineering tool is demonstrated. The emerging role of CFD in relation to regulatory procedures is discussed and the resources needed to exploit CFD are indicated.

1. INTRODUCTION

Over time the standards required for building construction and fitting out are rising. Formally these standards are set by government regulations, expressed through building codes. Informally, standards are set by the expectations of the broader community in relation to living and working conditions. As standards of living rise, in financial terms, expectations of health standards and comfort levels also rise. This process is expected to continue in the future, with progressively lower levels of air and water pollution being tolerated, and finer control over thermal comfort being demanded.

Building codes are also influenced by the analysis that engineers and architects can be expected to carry out. Traditionally hand, or simple electronic calculations, have been used with rather conservative requirements, i.e. large safety factors or deliberate over-design. Now practical engineering tools are available, based on finite element analysis (FEA) for structural design and computational fluid dynamics (CFD) for ventilation, thermal comfort, air quality, fire and safety and wind loading etc. These tools allow buildings to be designed and constructed with considerable precision. Thermal comfort and air quality can be determined with less than 5% error in local temperature or species concentration level, on a routine basis.

The maturity of FEA and CFD [1] as engineering tools in the building and construction industry follows the completely integrated use of such tools in the aerospace, automotive and chemical engineering industry sectors. In the Building Services area, CFD has been used to analyse both internal [2] and external flows [3], but more often as part of research studies than for direct engineering design. Previous reviews are provided by Chow [4,5] and Chen [6,7]. However the use of CFD to solve practical problems of building design, in relation to ventilation [8] and fire safety [9], is being emphasised and shown to be an effective tool for increasing engineering productivity and precision.

Engineers and architects are being trained to use FEA and CFD as part of Bachelor degree programs. In addition, reliable commercial FEA and CFD software running on "entry-level" PCs is readily available. Consequently, building codes are no longer restricted by the accuracy of the analysis tools available to practicing engineers and architects. So government regulations in the building area can reflect directly the standards of health and comfort demanded by the broader community. This implies standards will rise rapidly and FEA and CFD analysis will become a requirement as part of the tender submission/ process.

The present paper provides an assessment of what state-of-the-art CFD is capable of as an engineering tool. Although CFD is used across the spectrum from fundamental research to day-to-day engineering, there are some subtle distinctions to be aware of when comparing CFD in a research project and CFD used in an engineering project.

* Reprint with colour figures available from c.fletcher@cances.atp.com.au

In a research environment, much CFD analysis is performed using codes developed by the research group itself, e.g. Murakami et al. [10]. In the hands of skilled researchers CFD is an effective laboratory facility providing large quantities of data that is complementary to experimental measurements, e.g. using PIV or LDV in wind tunnels. Used this way the time taken to set up geometry or to visually interpret the CFD output is not usually a critical factor, since they are overshadowed by the time necessary to analyse and interpret the data.

Often research-driven use of CFD will look at phenomena that require more demanding physical models. For example for the severe wind flows around high-rise buildings, it is necessary to use large-eddy simulation (LES) turbulence models to capture accurately the unsteady separation from forward-facing corners and the corresponding instantaneous very low pressures experienced just downstream of the corner [11]. In addition it is necessary to use very accurate, often higher-order, discretisation schemes for the convective terms and fine spatial and temporal grids. LES turbulence modelling is also used with CFD simulations of internal flow [7].

However the use of CFD by a consulting engineering company to provide a client with an assessment of a proposed car park design, for example, will have different "terms of reference" from the research use of CFD. A key issue is that the available project time, e.g. 2-3 weeks, is likely to be much shorter than the characteristic time of a research project, 2-3 years. This factor of fifty has some immediate operational consequences. First all "industrial" CFD usage is based on commercial CFD software; second the engineer using CFD does not have much time to explore alternative scenarios.

The short project duration makes specific demands on the productivity of the commercial CFD software. It must be possible to set up the geometry and grid very quickly and be confident that grid refinement is reasonable. Typically the geometry is available in a CAD file, e.g. using AutoCAD. Effective commercial CFD software is able to read in the geometry from any common CAD file format, and set up and assess the grid quality with relatively little user intervention. The running of the software should rely, as much as possible, on default solver settings and be automatic if possible. Once the solution is available, the user needs immediate visual interaction with the CFD predictions, and will need the assistance of the visual output in presenting results to the client, typically. So for engineering use of CFD, the pre- and post-processing capabilities assume a greater importance, than for research use of CFD.

In Section 2 we examine the role of CFD as an enabling technology for building services, its use in explaining the influence of ventilation systems on local flow behaviour to clients, and its potential use in government regulations and building codes. A number of typical building services problems are examined in Section 3 and used to illustrate the capabilities of CFD. The process of using CFD as an engineering tool is described in Section 4, in relation to operation of a typical commercial code.

2. ROLE OF CFD

Computational fluid dynamics solves the full Navier-Stokes equations on a discrete grid. A typical three-dimensional grid would contain 50,000 to 1,000,000 cells. The solution in each cell could include, three velocity components, pressure, temperature, turbulent kinetic energy and dissipation and various species concentrations as primary data. Derived or secondary data could include mean age of air, relative humidity or PMV etc.

Theoretically, sufficient grid refinement will produce arbitrary accuracy. In practice 5% to 10% accuracy is sufficient to make most engineering design decisions. This level of accuracy can be obtained on coarser grids for many problems. Significantly CFD obtains local as well as global information. So locations of high velocity or temperature can be obtained quickly. However integral quantities, like mass flow rate through a device or forces or loads on a structure can also be determined.

At the start of a CFD investigation it is appropriate to test a case where the flow solution is known accurately, either from direct measurements or wind-tunnel simulations. The test case should be similar to the practical engineering problem of interest. The test case is useful in determining how much grid refinement will be needed to achieve the required accuracy and the adequacy of the numerical and physical models used in the CFD software.

Once the CFD model is validated, the engineering problem can be investigated to determine detailed velocity, temperature and species concentration distributions. This may involve systematically altering a key parameter, e.g. a component dimension or inflow rate, and observing the influence on a related outcome. This could be the local heat transfer to a section of wall caused by varying the (ambient) temperature of the inflow.

CFD provides the user with both visual and quantitative output. Visual output could be in the form of contours in planes or on surfaces, velocity vectors or massless particle trajectories. It is also possible to quickly create animations which aid the understanding of complex flow processes. As an example, Fig. 1 displays the airflow via the trajectories of massless particles, typically coloured by the age of the ventilation air supply, for a laboratory workspace in which the operator is manipulating some toxic sample and the global and local ventilation systems are designed to move toxic fumes away from the operator. It can be seen that a local air supply from the top of the workbench is drawn through an extractor at the back of the workbench to maintain safe operating conditions.

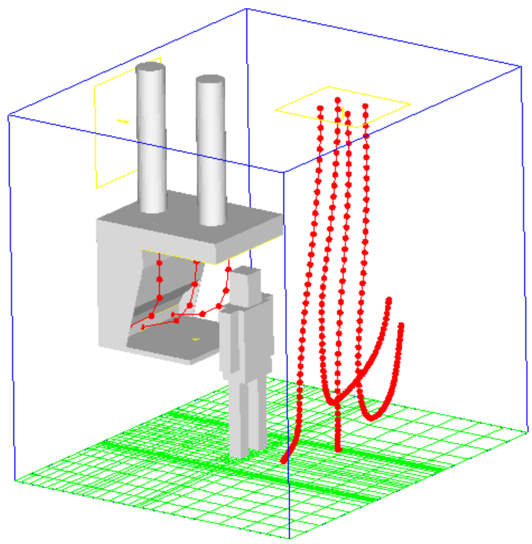

Fig. 1: Typical visual output produced by CFD software

Visual output is also very useful for explaining a complex flow phenomenon to a client or stakeholder. With the CFD solution on a notebook computer it is quite feasible to display various features of the flow in response to client questions, in addition to integration into more formal Powerpoint presentations. The slightly derogatory definition of CFD as "Colourful Fluid Dynamics" actually turns out to be a compliment, as it highlights the ability of high-quality commercial CFD codes to make visible, in a very effective way, the otherwise invisible air and pollutant behaviour. In a presentation the user can move seamlessly from the graphical output to the solver process and back again, as appropriate to explain specific features to a client, and to answer "what-if" questions.

By exporting the output files to specialised software, such as AVI, it is also practical to create videos or integrate the CFD solutions into other media, for marketing and promotion of consulting services, etc. As computing power per unit cost continues to increase, it is now feasible to simulate unsteady or transient phenomena, e.g. the early transient development of a compartment fire. So the presentation of the animated sequences is necessary to absorb the flow-related information. Such visualisation capability is evolving as a standard component of commercial CFD software.

The resources needed to exploit CFD include:

- an engineer trained in thermofluids
- a commercial software package
- a Pentium III PC with 512 Mbytes of memory and at least a 500 MHz CPU

The most important component is the engineer with fluids and thermal engineering (HVAC) skills. Compared with salary costs of the two engineers using CFD, the software cost is about 30% to 35% of the salary cost and the computer hardware cost is about 5% (in Australia). The training needed to be proficient is about 2 to 4 days for product familiarity, followed by one "real" engineering problem (1 to 2 months) to acquire expertise. However modern CFD software facilitates learning by doing. That is the skills can be acquired directly from the use of CFD itself.

This speed of learning is leading to much more in-house use of CFD, rather than employing external consultants. As well as being more economical, the immediate access to in-house resources leads to the application of CFD to problems so small as to not justify involving an external consultant. For example, a quick pre-tender analysis of a specific feature can be achieved with a CFD run on a coarse grid and only partially converged to confirm that a design idea of an experienced thermofluids engineer will be effective.

The ease of obtaining CFD solutions and the precision and reliability of the results, is expected to have a significant influence on building codes and government regulations. Traditionally building codes have allowed relatively simple calculation procedures, but have imposed substantial margins of safety. Now it is practical to reduce safety margins where the reliability and integrity of the CFD analysis is beyond doubt. There are indications that some regulating authorities are accepting and even demanding CFD analysis be undertaken, e.g. Land Transport Authority, Singapore.

A related trend is developing where regulating authorities use CFD themselves to assess the quality of engineering tender submissions. This is appropriate, as it makes the approval process faster, cheaper and produces more precise engineering solutions. It is expected that CFD analysis, as part of tender submissions, will be mandatory in the next three to five years, in many countries. This activity emphasises a secondary function of exploiting visualisation of the CFD solution to discuss building code requirements with regulating authorities.

The widespread use of CFD in building services engineering will make the provision of personalised air-conditioning a practical reality. Current research [16,17] indicates that providing pure or filtered personalised air that is free of contaminants, and cool and dry, contributes to the perception of thermal comfort as well as being more healthy. Providing a small amount of locally controlled heating completes the personalised air-conditioning requirement, and is a straightforward CFD design task.

3. TYPICAL APPLICATIONS

In this section some typical applications of CFD to common building services design problems are provided to illustrate what information CFD can provide and how it can contribute to engineering decisions.

3.1 Contamination and Thermal Comfort in a Photocopier Room

The pattern of air circulation produced in a room by natural or forced ventilation impacts on the thermal comfort and well-being of the occupants, particularly if there is a source of contamination or air pollution in the room. CFD reveals the (often not obvious) flow patterns, and provides quantitative details of contamination levels as well as human comfort indicators such as Predicted Mean Vote, Predicted Percentage Dissatisfied [14], radiation temperature, relative humidity, and mean age of air. This is illustrated by the familiar example of a room containing a photocopier which, in addition to being a source of heat, also emits contaminants.

The room (Fig. 2), contains the photocopier, the person operating it and various items of furniture. To maintain indoor air quality, air is drawn into the room through three inflow vents at one end of the ceiling, and extracted through a positive flow ventilation duct at the other end. To mitigate the heat and contaminant odours emitted by the machine, the operator will often keep the door open, thereby providing an additional air entry. However, the CFD analysis shows that this practice is, in fact, counter-productive. A small extension of the CFD analysis then verifies that a simple engineering modification will provide improved air quality whether the door is open or closed.

The geometry of the room and contents was prepared using Airpak's object-orientated preprocessor. The objects used (Fig. 17) were Blocks, Openings, Vents and the Person. When the CAD model was complete, the automatic meshing procedure was used to generate a computational grid of 33,800 (mainly) hex cells. Tets or prisms are used locally as required. The present grid is relatively coarse but is sufficient to resolve the overall features of the copier room ventilation. The governing equations, including a k-epsilon turbulence model and a transport equation for the contaminant, were solved iteratively until the residuals had been reduced by a factor of 10000, and the post-processing features of Airpak, v 2.0.6, were used to obtain the results shown in Figs. 2 to 5.

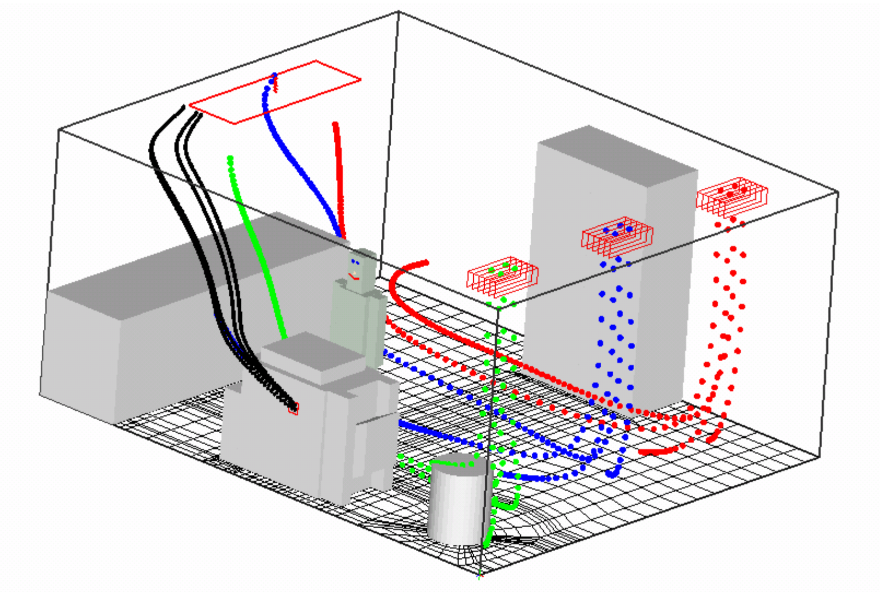

Fig. 2: Photocopier room, showing path of contaminant vapour escaping from rear of machine

In addition to displaying the geometry of the room and its contents, Fig. 2 shows the paths taken by several mass-less 'particles' of contaminant released from the opening at the rear of the copying machine. These indicate that the major contaminant flows towards the ceiling extractor. However particle paths do not give any indication of the dispersion of the contaminant. The contaminant concentration adjacent to the operator's face is indicated in Table 1. During post-processing, the values of solution variables, like the concentration, can be obtained at a specified point through a menu (Report, Fig. 17) initiated point data report.

If the door is opened the ventilation extractor draws most of its air directly from the open doorway. This results in higher contaminant concentration in the vicinity of the copier and operator (darker zones in Fig. 3). This is corroborated by the higher point value (Table 1). The door-open case results were obtained with a tetrahedral grid of 48,600 cells, which is relatively coarse for the amount of detail in the computational domain.

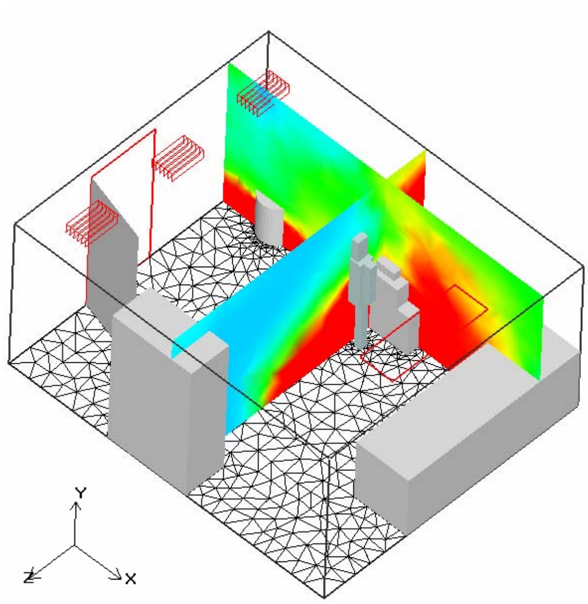

Fig. 3: Contaminant contours with door open, and high contamination near the operator's location

Since some users of the copier are likely to keep the door open, a more effective engineering solution can be sought by converting the passive ceiling vent, closest to the photocopier, into an active inlet by installing a fan in it. Two further CFD cases show that such a modification does indeed keep the contaminant away from the operator, whether the door is open or closed. Fig. 4 shows the path of the air flow immediately under the inlet vents for the door closed case, and the contaminant concentration in a horizontal plane close to the operator's breathing zone. The dark region suggests a locally high value of the contaminant concentration, which is also confirmed by Table 1.

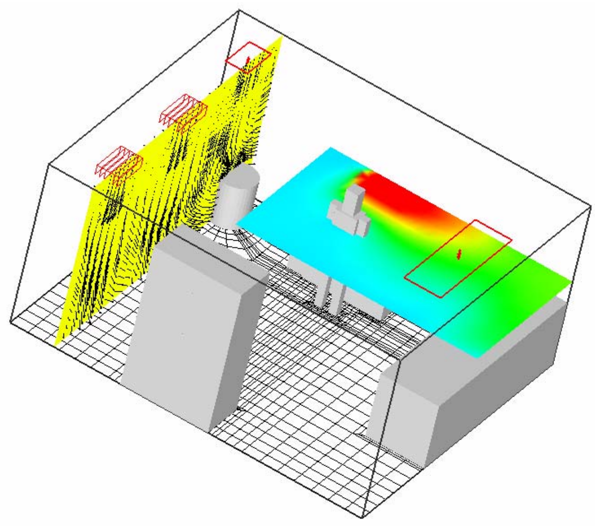

Fig. 4: Active inflow (furthest left-side duct) and door closed, showing local down flow and high contaminant near the operator's breathing zone

For the door-open case (Fig. 5) the active inflow vent provides a balancing downward flow which combines with the flow drawn through the door to keep the contaminant closer to the floor and effectively "sweep" it away from the operator to the end of the room under ventilation extractor. The low concentration level close to the operator confirms the effectiveness of this ventilation configuration.

Table 1: Contamination levels and PMV values near the operator's breathing zone

Case	Figure	Grid (type; no. of cells)	Contaminant concentration (mass fraction)	Temperature (°C)	Predicted Mean Vote (PMV)
Door shut	2	Hex; 33,800	0.0045	20.35	0.38
Door open	3	Tet; 48,600	0.0062	20.33	0.43
Door shut, with active inflow	4	Hex; 33,800	0.0040	20.25	0.05
Door open, with active inflow	5	Hex; 33,500	0.0011	20.05	0.07

Table 1 indicates the local temperature and predicted mean vote (thermal comfort) in the operator's breathing zone. Although the temperature doesn't vary between cases, the airflow is effective in countering the local heating effect of the copier. A PMV value of 0.4 implies feeling slightly hot, a value of zero is comfortable.

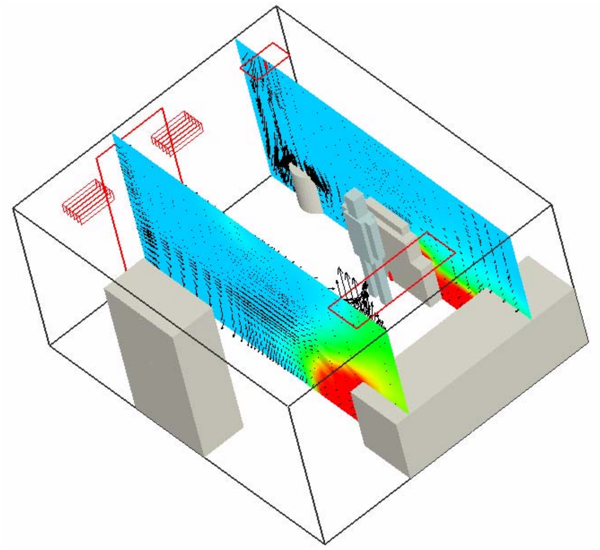

Fig. 5: Active inflow (furthest left-side duct) and door closed, showing local flow vectors and concentration in longitudinal planes

Clearly the proposed solution of activating the inflow fan is effective in reducing the contaminant level, whether the door is open or closed. The use of CFD reveals the unexpected result that the two open-door cases are responsible for the most and the least contaminant concentration adjacent to the operator. In addition to guiding the problem analysis and solution, the four CFD cases provide a wealth of quantitative data, such as the thermal comfort level of the operator which is shown in Table 1. By defining the clothes being worn by the person and the degree of activity (Fig. 18), the comfort level can be predicted according to Fanger's [14] model. This model is widely accepted, although CFD analysis [10] suggests that this model could be further refined, but possibly requiring the use of a finer grid and hence longer CPU time.

3.2 Pollution Control in an Enclosed Car Park

Internal car parks frequently cover a large area, but are usually of limited height and so provide a demonstration of the use of CFD in such 'extreme' geometries. Fig. 6 shows a car park covering 60 × 75 m with a ceiling height of 3 m. To conserve computational resources, the decision was taken to model rows of parked vehicles, rather than individual cars. The blocks represent eight double rows of parked vehicles, and two single rows. The vehicles are assumed to be 1.5 m high, and to have a ground clearance of 20 cm. Because a row of parked cars is not as impervious to the passage of air as a solid block, the blocks were modelled as 'resistances' (porous objects). The free area ratios of the blocks were set at 0.25 in the vertical direction, and 0.3 in the other two directions. The pressure loss coefficients were all set at 0.3. Ventilation air with a vertical velocity of 2 ms^{-1} is forced in through a ceiling duct (DE) running between the two banks of car rows, and escapes through the vehicle entry opening (A), as well as one large (B) and one smaller (C) vent in two of the other corners of the room. With the above assumptions it is possible to accurately represent the domain with a grid of 97,330 cells and to achieve sufficient convergence (rms residuals reduced by 10000) in 1700 steps.

A preliminary CFD analysis shows that a local air re-circulation or 'vortex' is established near the corner of the park not having an exit vent. Consequently there is concern that the exhaust gases from an idling car (pollution source) near the unvented corner might become trapped there, resulting in unacceptably high contaminant levels. To investigate this possibility, a source of carbon monoxide rising from ground level with a vertical velocity of 0.1 ms^{-1} was introduced in this region (F). Mass-less 'particle' traces from this source are shown in Fig. 6. It can be seen that while some of the CO is carried out the vehicle entrance, a greater proportion becomes entrained by the incoming ventilation air from the duct (DE).

The side view of the pollution source and adjacent row of cars is shown close-up in Fig. 7. An examination of Figs. 6 and 7 indicates that as the CO rises above car top level, it is carried by the overall circulation towards the air inlet region. That portion of the CO which reaches the ceiling at some distance from the air inlet is carried to the vehicle entry opening. However the CO portion which reaches the ceiling close to the air inlet is drawn towards it, and eventually mixes with the inlet air.

Fig. 7 also shows the concentration of CO in two intersecting planes. The dark regions correspond to a CO mass fraction of 6%. It is clear from Fig. 7 that the CO disperses mainly in the space above the parked cars due to the "transport" by the ventilation air. The present flow analysis suggests that adding a horizontal component to the inlet air velocity (e.g. with louvres) would substantially mitigate the local high pollution levels. This may be economically more attractive than installing a new exhaust vent in the previously un-vented corner of the car park.

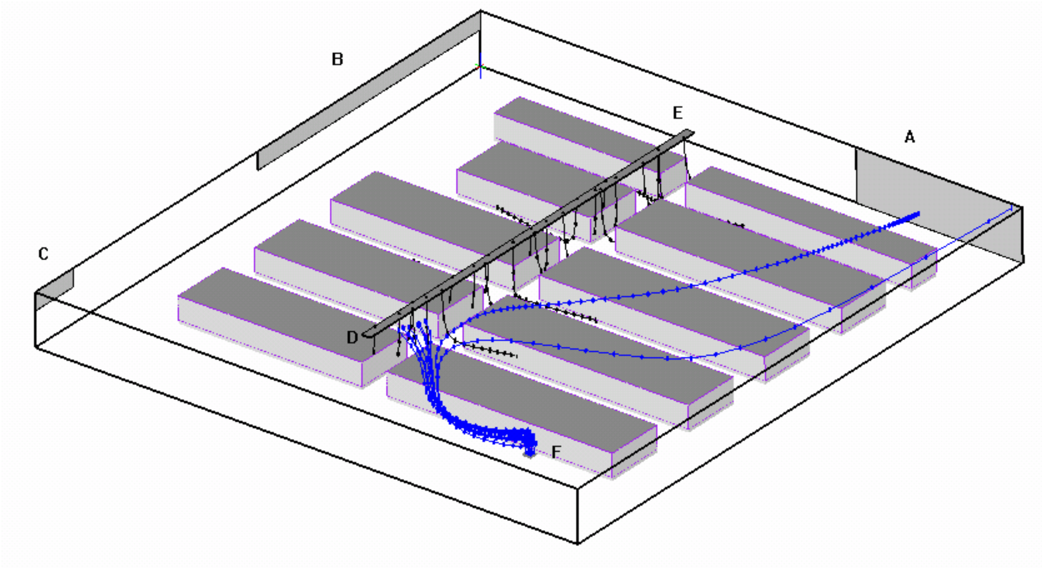

Fig. 6: Car park geometry and trajectories of a pollution from source (F) and inlet duct (DE) air (vertical dimension is stretched by a factor of 2)

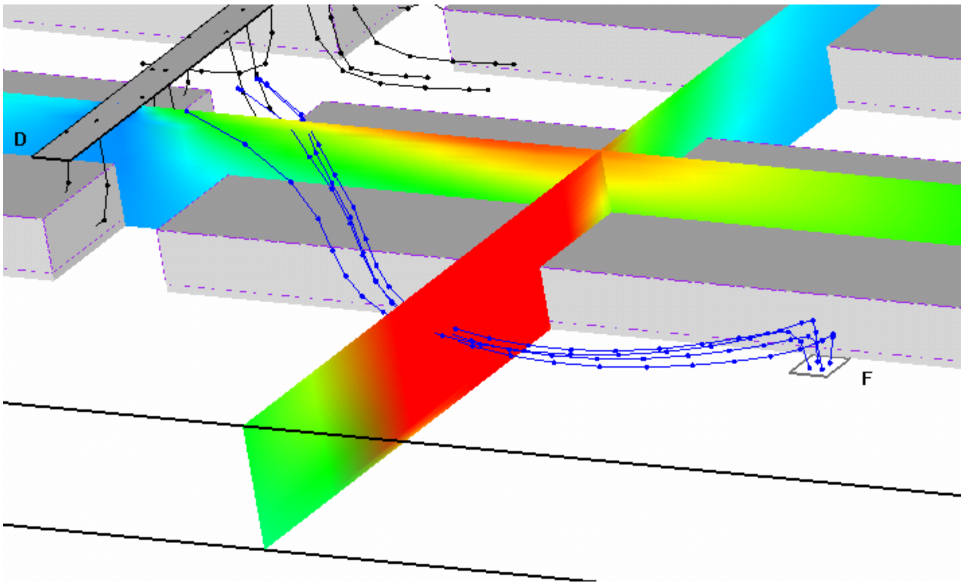

Fig. 7: Side view of carbon monoxide trajectories and concentration near source (F)

3.3 Natural Ventilation and Thermal Comfort with Solar Heating

In this example the use of wind flows to provide ventilation of a building without mechanical ventilation systems is considered. The most basic building configuration is assumed, namely a single room with a window on the windward side and a door on the downwind side (Fig. 8).

In addition it is assumed that solar heating is creating a heat load on the roof and one wall. So it is of interest to see what the potential is to maintain a reasonable level of comfort. A steady wind velocity at the inflow boundary of 10 ms^{-1} is imposed. At the outflow boundary atmospheric pressure is set, and symmetry is assumed on the two side boundaries. An outflow boundary is also imposed on the top boundary to allow for the deflection of the incident wind flow due to the obstructing effect of the building.

The computational domain is shown in Fig. 8. Local grid refinement is provided where velocity gradients are high, such as in the immediate vicinity of the window and door. The overall computational domain requires a grid of 95,500 cells and the solution for velocity, pressure and temperature requires 760 iterations to converge. Air velocity vectors at window height are shown in Fig. 8. Also selected flow paths from the inflow boundary show that some air does flow through the window into the room. But flow from adjacent locations may flow around or over the building.

The building has a blocking effect on the wind flow causing an acceleration of the flow around the sides and over the top of the building. The build-up in pressure on the windward face of the building and the low pressure immediately downstream of the building creates a local "ventilating" air flow in through the window and out through the door. Inside the room it is expected that the orientation of the window and door creates a local air flow pattern, which maintains a reasonable distribution of temperature and thermal comfort (PMV), as indicated in Fig. 9.

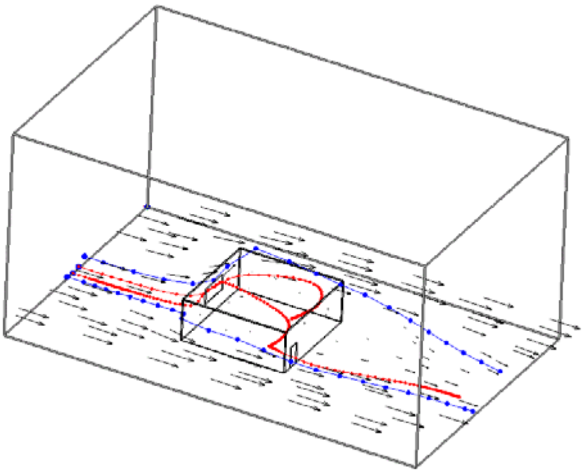

Fig. 8: Computational domain and selected air velocity vectors and path lines

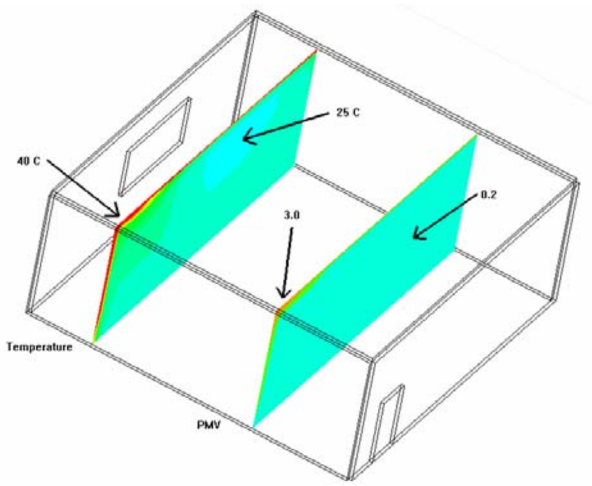

Fig. 9: Distribution of temperature and PMV (thermal comfort index) in a naturally ventilated room

The distribution of temperature contours in a vertical plane near the window, in Fig. 9, shows the effect of solar heating which is assumed to be asymmetric providing a heat load on the ceiling and one wall. The other three walls are assumed to be adiabatic. The layer of hot air rising up the solar heated wall and lying under the (solar heated) ceiling creates a local temperature of 40^{o}C and a PMV = 3.00.

The Predicted Mean Vote (PMV) is calculated for persons wearing normal weight clothing, and engaged in moderate working activities. Fig. 9 shows the dominant effect of the natural ventilation. It is apparent that, despite the substantial heat load, natural ventilation from the strong wind ensures that most regions of the room would be comfortable for the occupants.

The distribution of the mean age of air is shown in Fig. 10. The cross ventilation maintains reasonable freshness throughout the room. The stalest air is seen to be concentrated in the top corner of the room, which is also the hottest location. The air flow paths indicate the natural ventilation creates an overall flow pattern directed into the corner away from the door. This is corroborated by the lower values for the mean age of air (10 s). Some of the air eventually flows out the door (Fig. 8).

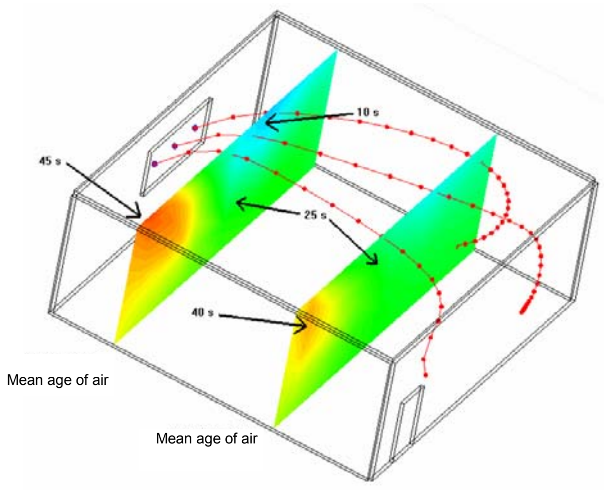

Fig. 10: Relationship between mean age of air and air flow paths

This example demonstrates the power of CFD to establish the overall flow pattern, and then to zoom in on specific features, such as the hot region adjacent to the ceiling which receives a high solar load but reduced cooling from the internal flow pattern in the room. The present CFD model could be used to examine the influence on natural ventilation effectiveness of wind speed and direction, shadowing effects of adjacent buildings and of ambient temperature and solar heating loads. In a more realistic building this could guide the location and size of windows and doors, and whether auxiliary air conditioning is required.

3.4 Heat Stacking Effect in a High-Rise Air Well

CFD is used to model the steady-state heat stacking effect in an air well of a typical high-rise apartment block installed with air-conditioner condensing units. The heat dissipated by the condensing units causes the surrounding air to rise (buoyancy effect) and as a result, temperature rises in the air well due to a heat stacking effect. Each condensing unit on each higher level is operating in a higher ambient temperature environment, which will lower the operational efficiency of the condensing units. It is important to understand the detailed airflow and temperature distribution for an air well design, in the light of energy conservation, resident comfort and equipment operation.

The high-rise apartment block consists of 10 levels (30 m) with an air well and passage as shown in Fig. 11. There are 4 condensing units (marked in blue) installed on the air-conditioning ledge at each level in the air well. Each condensing unit has dimensions $1 \times 0.6 \times 0.4$ m. It has an airflow rate of 0.6 kgs^{-1} and dissipates 7 KW approximately. It is assumed that the ambient temperature is $33^{o}C$ with no wind (a worst case scenario) and that all condensing units are operating simultaneously.

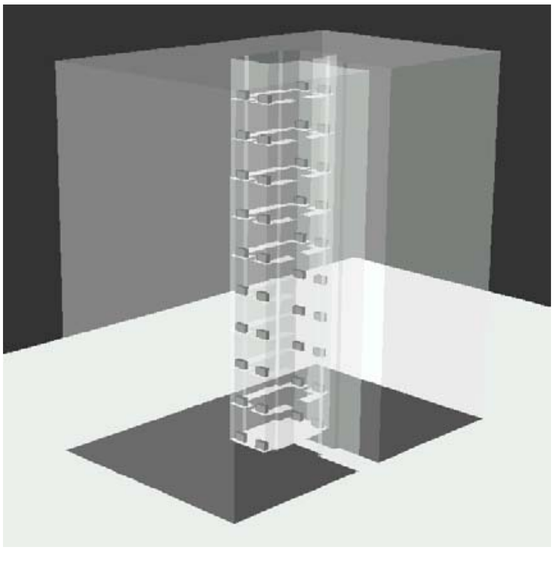

Fig. 11: Isometric view of a typical high rise apartment (10 levels) showing an air well and air-conditioner condensing units

The geometrical model is built in less than 20 minutes using the pre-defined objects in Airpak. A total number of 29,972 computational cells (hexahedra elements) are generated for the model using the automatic grid generation in Airpak. Finer grids are also generated around individual objects automatically to resolve the flow physics of the problem optimally.

The heat stacking effect belongs to the class of mixed (forced & natural) convection problems and the effect of the buoyancy force is modelled by adding the gravitational force, as a source term, to the Y-momentum equation in the solution procedure. The surface-to-surface radiation model is used to model the radiative heat transfer and the view factors are computed automatically in Airpak. The turbulence closure is achieved by using the standard k-epsilon model coupled with the standard wall function.

The additional source term (gravitational force) contributes to the numerical challenge of the computation and requires smaller under-relaxation factors or other pre-conditioning techniques to ensure a converged solution. Hence, it is computational more demanding to solve this class of problem compared with a forced convection problem. Approximately 1,250 iterations and less than 2 hours of CPU time are required for a converged solution on a typical personal computer.

In general, the temperature in the air well rises due to the heat stacking effect and causes the warm air to rise. As a result, cool air is drawn through the air passage to replace it. The temperature and velocity contours in the air well and passage at various levels are indicated in Figs. 12 and 13 respectively.

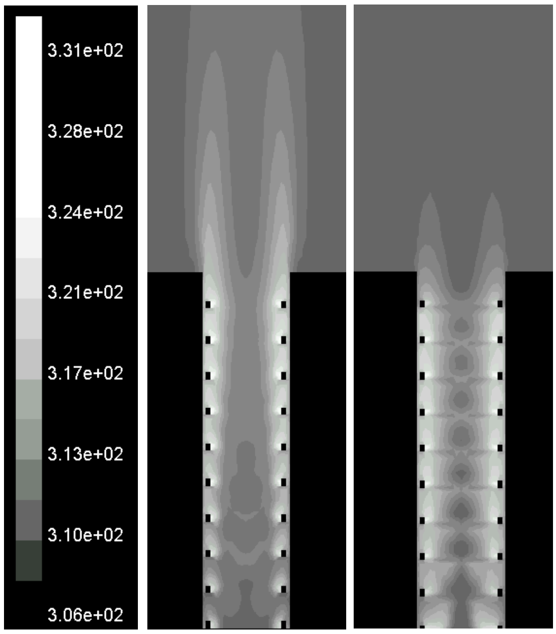

Fig. 12: Temperature (K) contours at inner (left) and outer (right) condensing units indicating temperature rises in the air well

It can be observed from Table 2, that the inlet temperature of the inner condensing units is higher than that of the outer condensing units. This is attributable to the flow recirculation/reversal in the air well, opposite of the air passage flow, which causes poor airflow circulation at the inlet of the inner condensing units, as shown in Fig. 13.

It is interesting to note that the inlet temperature of the outer condensing unit decreases at the second level and then increases again for higher levels. This is due to the direction of the airflow, which tends toward the outer condensing unit as shown in Fig. 13. It is also observed that the inlet and outlet temperature of the condensing units at the tenth level are lower compared with the condensing units at the ninth level. This is due to the additional supply of cool air from the top of the air passage as shown in Fig. 13.

This case study demonstrates that CFD can be used as a highly effective design and analysis tool to model the detailed air flow and temperature associated with the heat stacking effect in an air well of a typical high-rise apartment block. CFD provides both architects and engineers with the ability to analyse airflow and heat transfer in an air well design and to study the impact of heat stacking effect in relation to the performance of the condensing units for a high rise apartment block, and to adjust the design accordingly, prior to any construction.

Table 2: Mean temperature of the outer and inner condensing units

Level	Outer condensing unit		Inner condensing unit	
	Inlet temperature (^oC)	Outlet temperature (^oC)	Inlet temperature (^oC)	Outlet temperature (^oC)
1	38.78	47.02	39.00	47.48
2	37.76	45.89	39.51	48.21
3	40.76	49.02	41.38	50.06
4	41.74	49.91	42.55	51.39
5	42.86	50.97	43.74	52.44
6	43.36	51.48	44.40	53.20
7	44.00	52.06	45.18	53.83
8	44.34	52.45	45.74	54.52
9	44.41	52.45	46.36	54.99
10	39.15	47.19	44.81	53.40

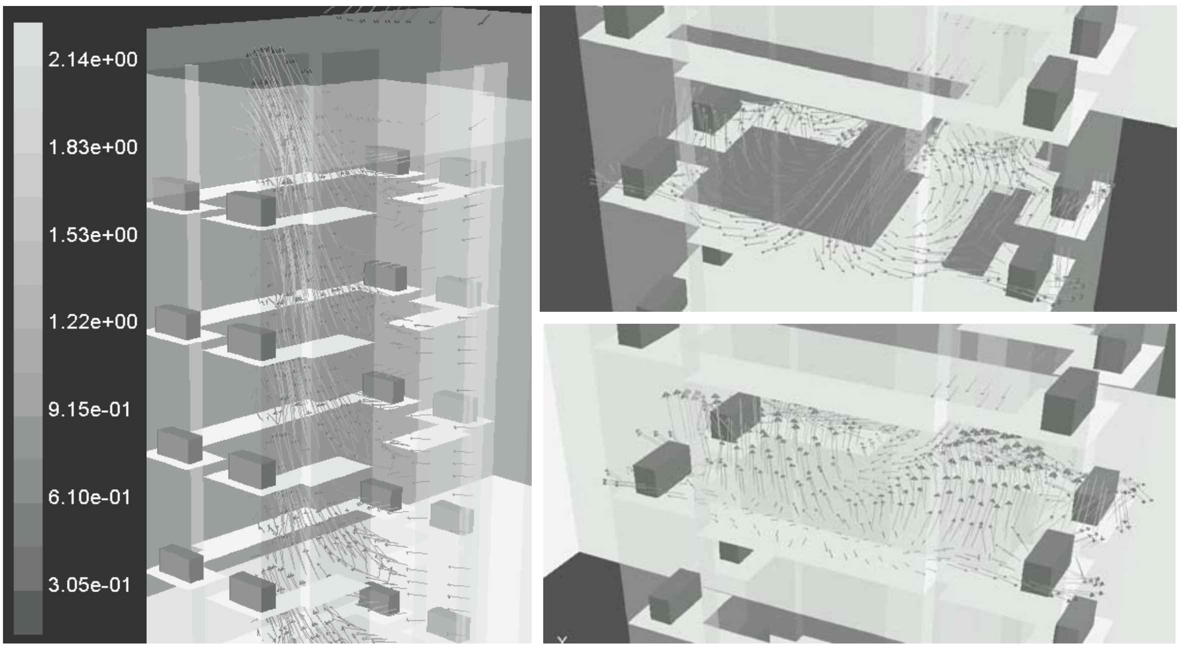

Fig. 13: Velocity (ms^{-1}) vector plot indicating air drawn through top & side of the air passage (left), and at 2nd level (right, above) indicating airflow recirculation region and at 5th level (right, below) indicating airflow reversal region

3.5 Thermal Plumes in a High-Rise Apartment Building

The CFD analysis of thermal plumes in high-rise buildings has been carried out before [12]. However, the analysis of the dispersion of combustion products in a high-rise building surrounded by other buildings requires a relatively fine computational grid and more advanced CFD software to model the geometric complexity. In many industrial building services CFD codes the required three-dimensional modelling, in conjunction with having a sufficiently accurate turbulence model in the presence of high temperature gradients and multi species causes divergence of the iterative solver. With the most recently developed CFD codes these complexities can be addressed and analysed accurately in many practical applications, including the fire and heat dispersion in high-rise buildings surrounded by other buildings.

For practical engineering outcomes, the CFD software must be able to create complicated three-dimensional geometries and associated grids with minimal user input and in a small amount of time. It should be possible to assess the quality of the grid and improve the grid, locally or globally, semi-automatically. In addition, the CFD software needs to model turbulence, buoyancy effects, thermal and species diffusion accurately without compromising the robustness of the pressure-velocity coupling and multigrid solution algorithm.

In this study, five buildings with different sizes are placed in a computational domain of 2500 × 1600 × 1000 m (Fig. 14). Symmetry boundary conditions are applied to simulate the existence of identical multi-block configurations on each side of the indicated building complex. The distance from the inlet to the exit is taken to be 2500 m to reduce the influence of upstream and downstream boundary conditions. A relatively coarse tetrahedral mesh has been used to capture the dominant flow features economically. Next to the buildings a finer mesh should be used to capture flow gradients more accurately. The inlet velocity of air is taken to be 2 ms^{-1}.

The buildings configuration and the flow distribution around the buildings in a vertical plane are indicated in Fig. 14. As it shows, the flow goes around each building generating a small downstream vortex. The velocity vectors shown in Fig. 14 provide the distribution at x = 750 m from the origin of the computational domain. The vortex generated behind "Front Left" building reaches the "Tower" (the highest building in this simulation) that in turn contributes to a small vortex behind the "Tower". The length of the arrows corresponds to the magnitude of velocities at different locations.

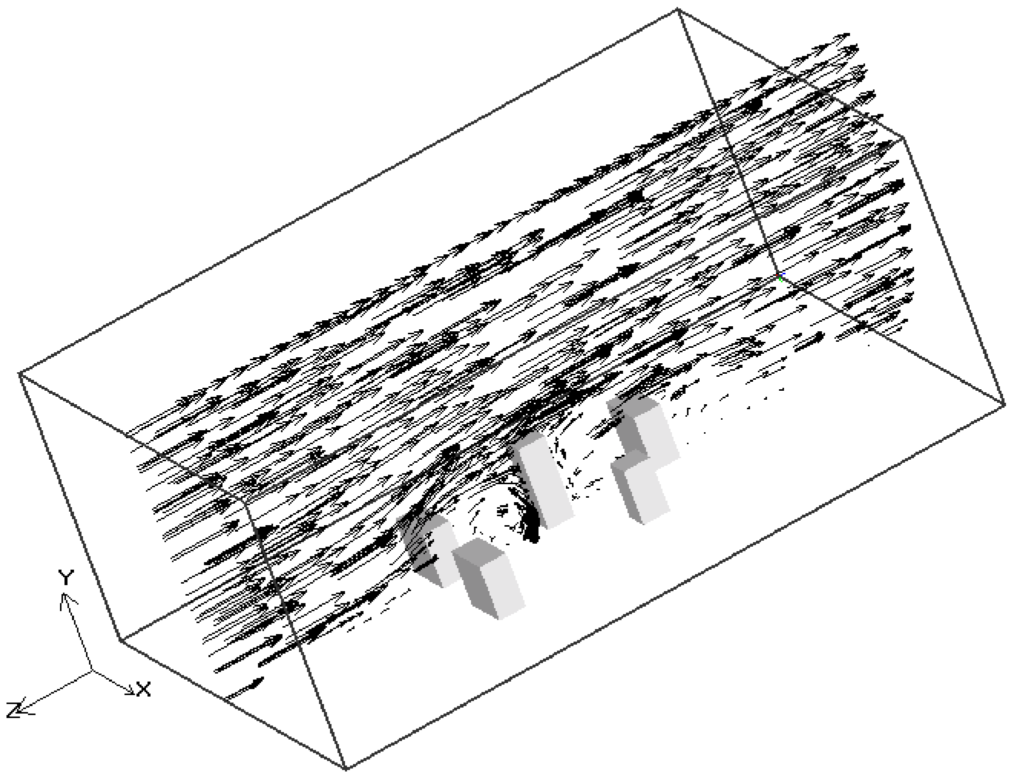

Fig. 14: Velocity distribution in a vertical plane and the buildings configuration

The fire source produces a secondary gas (CO) with 100% concentration at the 50 m height of the "Tower" building and a maximum temperature of 1000 ^{o}C at the source centre. The fire is assumed to discharge carbon monoxide with 1 and 0.5 ms^{-1} "y" and "z" velocities at the fire location and to be generating a heat transfer output consistent with real flame behaviour [13]. The temperature distribution is presented in Fig. 15. The grid is adequate for preliminary analysis, but would need to be refined to extract reliable quantitative information.

Since the size of the fire source is much smaller than the whole computational domain, the dispersion of carbon monoxide in Fig. 16 covers only a small section. This study shows one of the strong features of Airpak in simulating fire dispersion using a tetrahedral grid and an RNG-based turbulence model. The concentration (mass fraction) of CO next to the fire source is shown in Fig. 16. Interestingly, the flow recirculation generated in front of the tower building transports most of the CO downwards.

If a fire in an apartment, i.e. a representative compartment fire, has a natural ventilation passage, e.g. a broken window, the fire spread will often occur up the outside of the building [12] and ignite the next higher apartment if a window is open. The present results suggest this will still be the case if the ambient wind is sufficiently light. However the present CFD model could be used to examine the influence of wind strength, direction and wind-shadowing effects of neighbouring buildings on the fire progress within the tower building.

This present study shows the value of CFD when field measurements are not easy to obtain and the cost of building and testing a realistic prototype would be prohibitive. In a parametric approach (Fig. 19) it would be feasible to systematically vary the size and temperature of the fire source, the shape and location of the tower and nearby buildings to confirm satisfaction of relevant building codes.

4. CFD AS AN ENGINEERING TOOL

Using CFD as an effective engineering tool has changed hugely in the last 15 to 20 years. A useful analogy can be made by comparing a state-of-the-art car in 1905 with a modern automobile in 2001, and the role of the driver. In 1905 the driver needed a deep knowledge of all the contributing functions to keeping the car moving smoothly, balancing the speed of the engine with the speed of the car, manually adjusting ignition timing depending on the engine load. In 1905 a state-of-the-art car was not a very robust tool for guaranteeing you could get from A to B. In 2001 the driver needs primary skills, e.g. steering, braking and accelerating, but much of the engine and handling robustness and reliability is built into the detailed design.

In 1980 most CFD codes were not very robust and needed the close attention of their originators to obtain accurate predictions, reliably and consistently. However in 2001, high-quality commercial CFD codes are as reliable and robust as a modern automobile. The engineer uses a commercial CFD package as an engineering tool, without needing to worry about subtleties of finite volume discretisation or algebraic multigrid solvers. In the modern context we are in the "CFD-without-PhD" era.

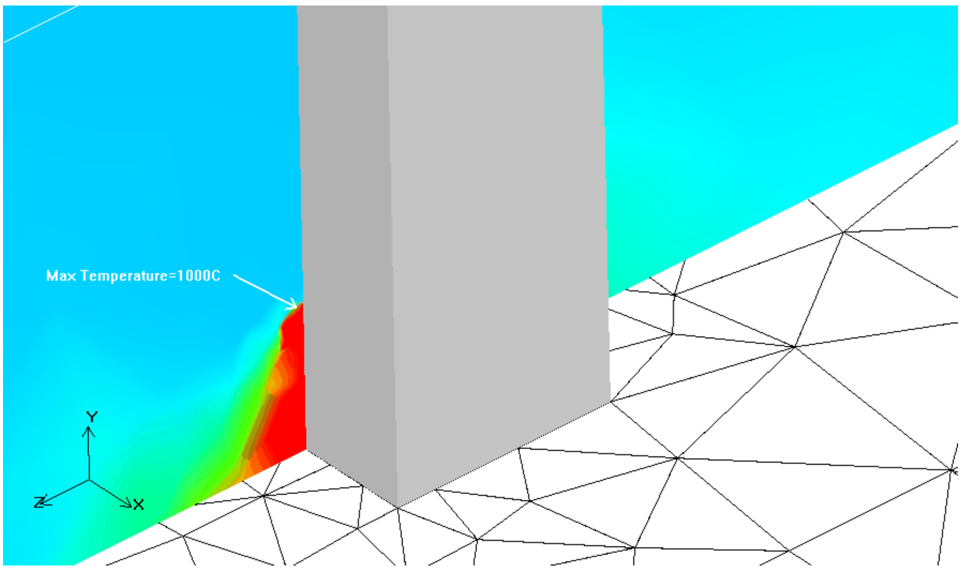

Fig. 15: The temperature distribution adjacent to the fire source

The engineer interacts with the CFD software via a simple menu system (Fig. 17), no more complex than an Excel spreadsheet. Each menu provides a sub-menu containing grouped activities, such as positioning the features of the computational domain. Setting up a typical problem might require one to two hours.

In Fig. 17 the top menu line indicates the broad stages of setting up a CFD simulation from reading in an existing file (File), setting up the geometry in the computational domain (Model is open in this view showing the next level down of specific objects, e.g. Person), defining the solution procedure (Solution), examining the results visually (Post) and reporting the CFD simulation outcomes (Report).

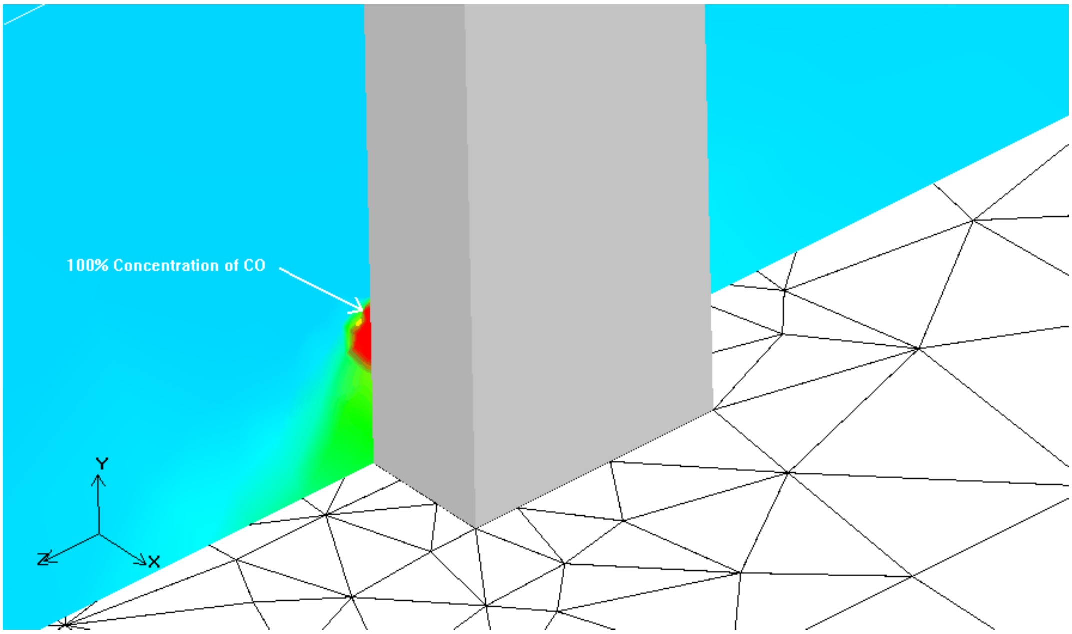

Fig. 16: Carbon monoxide concentration (mass fraction of CO) adjacent to the fire source

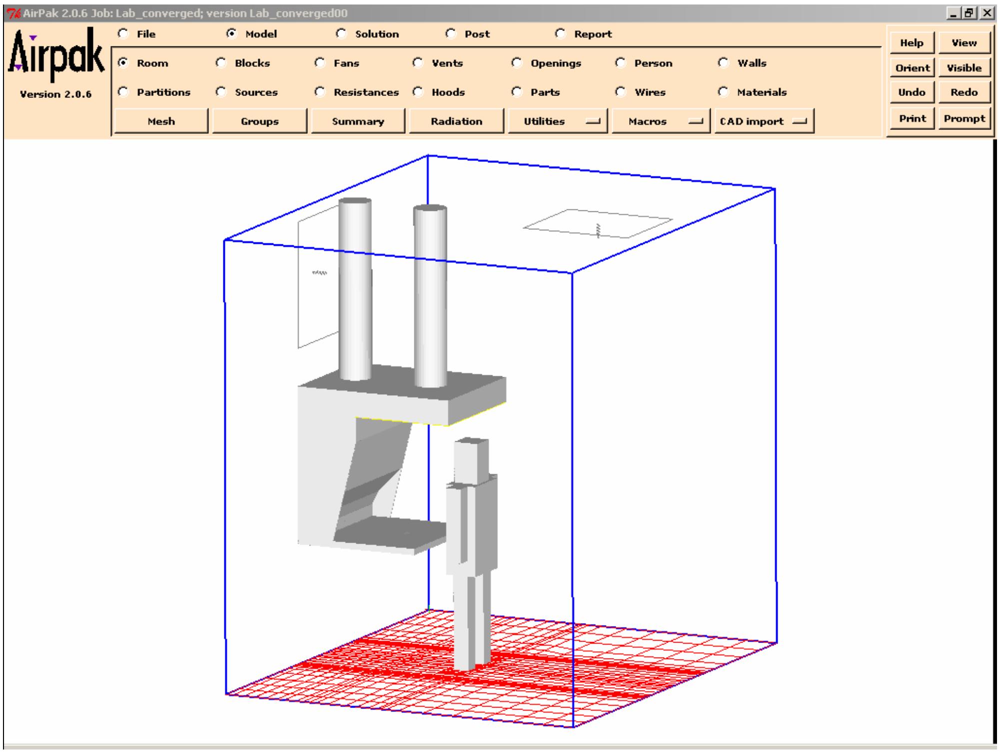

Fig. 17: Typical CFD menu system for controlling problem set up and solution

In a typical CFD project, an engineer will read in the geometry from a CAD file, or create it in the CFD package by manipulating (translating and stretching) standard objects. The computational domain could be a room, a car park or the region surrounding one or more buildings. Once the geometry has been set, the engineer may make some simple decisions controlling the grid generation. For example, clustering more grid points close to a surface or region. Subsequently the CFD package generates the grid automatically.

In the next stage the CFD package creates the solution by repeatedly solving the equations in each grid cell. This may take from 500 to 1000 iterations (repetitions) to produce an accurate solution. The modern trend is for the mesh generation and solver stages to be fully integrated, so there is no need for the user to be directly involved.

The user spends most time examining the results, both visually and quantitatively. A typical sequence would be to examine the velocity vectors in various planes to determine the overall flow behaviour, checking for areas of recirculation, stagnant air regions or high-velocity regions. A secondary stage could review the corresponding pressure distribution, as this will determine loads on various surfaces. A third stage could involve examining the temperature distribution, checking for locations of unacceptably high temperatures. If the problem involved transport of pollutants, the concentration of individual species would also be examined, looking for areas of unacceptably high concentrations.

Some CFD codes provide specific post-processing in relation to thermal comfort of human operator. It is possible to define the clothing and workload of the operator, in a way which allows an energy balance to be established for the operator and a measure of the expected thermal comfort (Fig. 18). Often the post-processing will involve probing specific points in the flow domain to obtain quantitative data, a temperature or derived quantity like predicted mean vote (PMV). The outcome of the post-processing maybe to accept a specific design or to guide the choice of design changes, which will then be examined by subsequent CFD simulations.

In some CFD software the process of varying key parameter in a systematic way can be made automatic (Fig. 19). The user specifies the starting value and increment of a specific variable. The CFD code is run automatically providing the user with a separate solution for each parameter choice. In practice on a PIII PC, six to eight cases might be run overnight, allowing the user to make a best choice for a particular design variable.

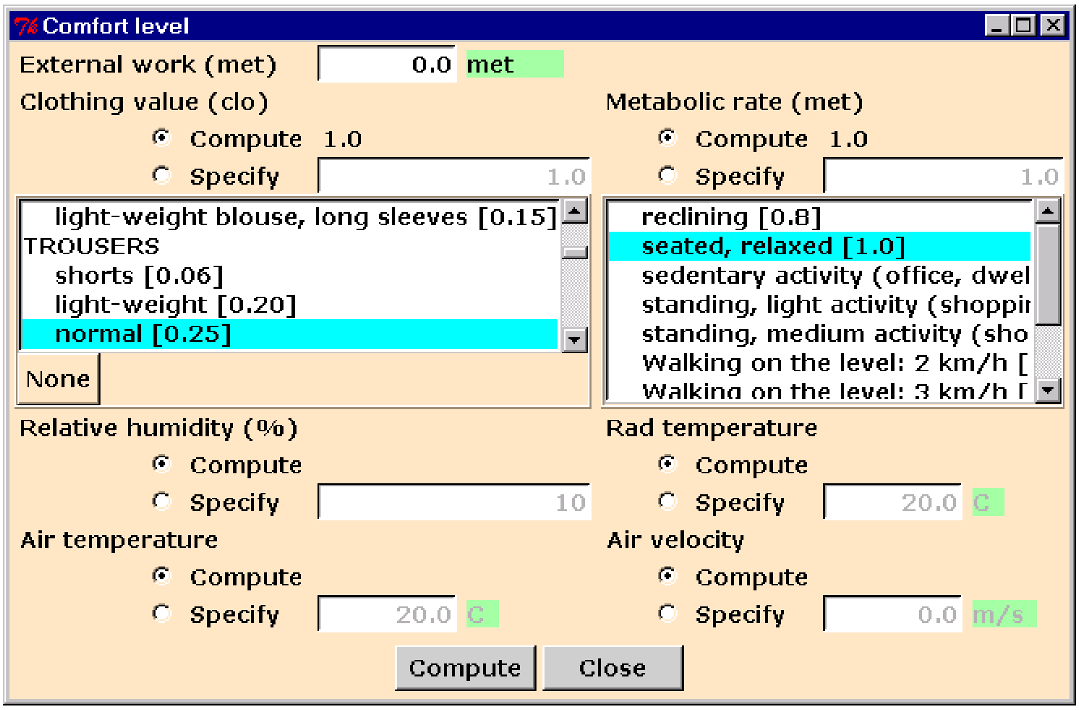

Fig. 18: Thermal comfort assessment under different work conditions and clothing

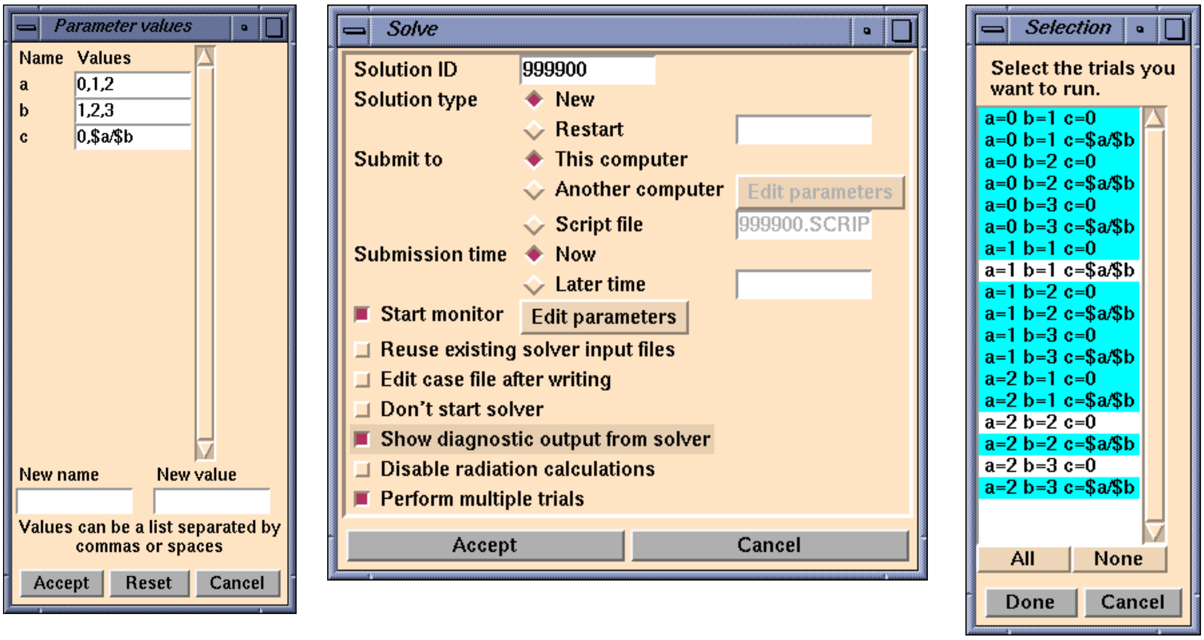

Fig. 19: Typical user controlled menus to allow parametric execution

The use of CFD provides some direct benefits to the user. Results for a given engineering problem can be obtained in a much shorter time and at lower cost than building and testing a prototype or wind tunnel model. The results are also much more detailed, allowing more precise decisions to be made, and facilitating explanation of the flow behaviour to stakeholders.

5. CONCLUSIONS

High quality commercial CFD is an effective engineering tool for the building services engineering industry. It can determine detailed flow distributions, temperatures and pollutant concentrations to better than 5% accuracy, without excessive effort by the software user, permitting engineering decisions to be made with considerable precision. An engineer with thermofluids expertise can acquire useful CFD skills within a couple of weeks, and a degree of expertise within two to three months. A specific problem can be set up in a couple of days, and a systematic fluid flow analysis undertaken within another four to five days, obtaining and analysing two specific cases per day. Typical examples provided in Section 3 indicate that both visual and quantitative data can be obtained without difficulty for most fluid flow, heat transfer and pollutant transport problems encountered in building services engineering.

REFERENCES

1. C.A.J. Fletcher, Computational techniques for fluid dynamics, 2nd edition, Springer, Heidelberg, Vols. 1, 2 (1991).

2. S. Kato, S. Murakami, S. Shoya, F. Hanyu and J. Zeng, “CFD analysis of flow and temperature fields in an atrium with ceiling height of 130 m”, ASHRAE Transactions, Vol. 100, No. 2, pp. 95-104 (1994).

3. S. Murakami, “Comparison of various turbulence models applied to a bluff body”, J. Wind Engineering and Industrial Aerodynamics, Vol. 46-47, pp. 21-36 (1993).

4. W.K. Chow, “Application of computational fluid dynamics in building services engineering”, Buildings and Environment, Vol. 31, No. 5, pp. 425-436 (1996).

5. W.K. Chow, “A discussion on using computational fluid dynamics in building services engineering”, Architectural Science Review, Vol. 42, No. 2, pp. 121-128 (1999).

6. Q. Chen, “Computational fluid dynamics for HVAC: Successes and failures”, ASHRAE Transactions, Vol. 103, No. 1, pp. 178-187 (1997).

7. Q. Chen and J. Srebric, “Application of CFD tools for indoor and outdoor environment design”, International Journal on Architectural Science, Vol. 1, No. 1, pp. 14-29 (2000).

8. W.K. Chow, “A comment on studying the ventilation requirements for buildings in the Hong Kong Special Administrative Region”, International Journal on Architectural Science, Vol. 1, No. 1, pp. 1-13 (2000).

9. H.R. Baum, O.A. Ezekoye, K.B. McGrattan and R.G. Rehm, "Mathematical modelling and computer simulation of fire phenomenon", Theoretical and Computational Fluid Dynamics, Vol. 6, pp. 125-139 (1994).

10. S. Murakami, S. Kato and J. Zeng, "Combined simulation of airflow, radiation and moisture transport for heat release from a human body", Building and Environment, Vol. 35, pp. 489-500 (2000).

11. S. Murakami, "Overview of turbulence models applied in CWE-1997", J. Wind Engineering and Industrial Aerodynamics, Vols. 74-76, pp. 1-24 (1998).

12. E.R. Galea, D. Berhane and N.A. Hoffmann, "CFD analysis of fire plumes emerging from windows with external protusions in high-rise buildings", In: C. Franks and S. Grayson (editors), Proceedings of the 7th International Fire Science & Engineering Conference: Interflam 96, pp. 835-839 (1996).

13. A.Eghlimi, B. Penglis, I.F. Mayer and R. Roomina, "Atrium smoke analysis with computational fluid dynamics", Proceedings of AIRAH 2000 Conference, Melbourne, March (2000).

14. P.O. Fanger, Thermal comfort, McGraw-Hill, New York (1973).

15. ISO 7730, Moderate thermal environments – Determination of the PMV and PPD indices and specification of the conditions for thermal comfort (1994).

16. J. Toftum, A.S. Jorgensen and P.O. Fanger, "Upper limits for air humidity to prevent warm respiratory discomfort", Energy and Buildings, Vol. 28, No. 1, pp. 15-23 (1998).

17. P. Wargocki, D.P. Wyon, Y.K. Baik, G. Clausen and P.O. Fanger, "Perceived air quality, sick building syndrome (SBS) symptoms and productivity in an office with two different pollution loads", Indoor Air, Vol. 9, No. 3, pp. 165-179 (1999).