A IVC
#10241

MODELLING AIR MOVEMENT

a BEPAC Newsletter supplement Winter 1995/6

In this issue

Interested in Weather Data?
See Eric Keeble's article on page 25 and details of meeting at CIBSE on 15 May - booking form enclosed with this Supplement

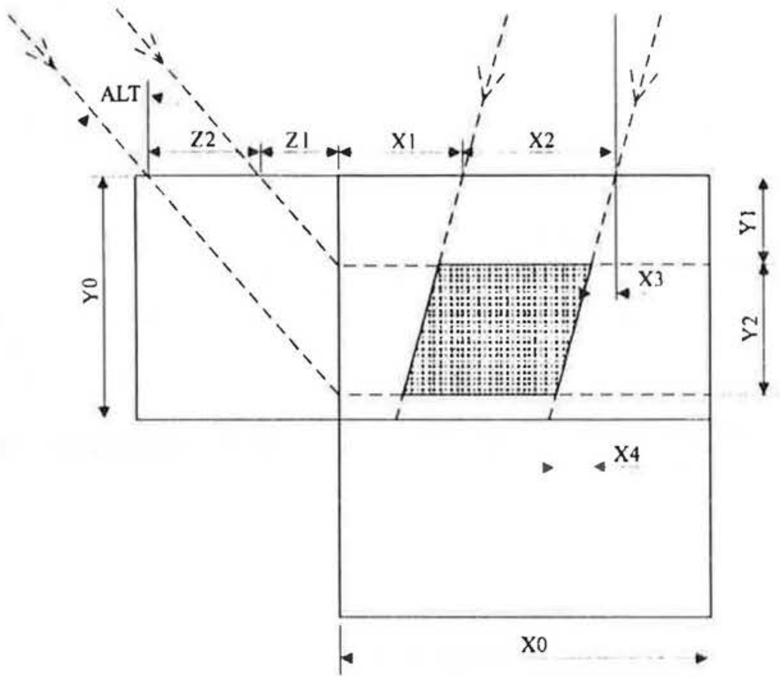
Modelling convective flow and heat transfer induced by sun patches

Peter Arnold and Geoff Hammond, University of Bath, Andy Irving, Rutherford Appleton Laboratory, and Chris Martin, Energy Monitoring Company

Recent work funded by the EPSRC has shed light on the way in which convective heat transfer should be treated in building energy simulation programs when applied to passive solar buildings. It examined the evolution of transient buoyancy-driven air movement and heat transfer in modern low-energy dwellings stimulated by external climatic changes (particularly solar gain, but also factors such as long-wave radiation and ambient dry-bulb temperature). The dynamic response of convection processes within a thermally light-weight room was assessed with and without an attached sun-space (or 'conservatory'). The relevance of the work is two-fold; firstly, the transient buoyancy-driven convective flow problems were addressed at a fundamental level; and secondly design criteria or guidelines for naturally-ventilated enclosures (rooms and conservatories) are being deduced. These will then aid the design of passive solar buildings, with obvious benefits in terms of energy efficiency and environmental protection. The grant was awarded as part of the Research Council's Clean Technology Initiative, in recognition of the importance the results will have in extending and/or validating design methods used for passive solar buildings. Prior to the project formally starting, discussion took place with both British Gas and the ETSU Passive Solar Programme in order to ensure that the work programme met the requirements of the wider community.

Over recent years there has been some controversy concerning the most reliable way to estimate internal surface convective heat transfer coefficients (h_c) for the purpose of building energy simulation. The CIBSE Guide currently recommends fixed values depending on the orientation of the surface. This implies buoyancy-driven (rather than forced) convection over smooth surfaces, and takes no account of variations with surface-to-air temperature differences or size of the surface. High values of the former induce transition from laminar to turbulent flow. In 1983 Farshad Alamdari and Geoff Hammond derived, in what might be regarded as a first principle manner, rather more elaborate formulae for h_c that account for transition effect as well as orientation. However, they were careful to provide advice to users on the still restricted range of conditions over which the new correlating equations were valid. These expressions (or their simplifications) have now been incorporated in most of the well known UK dynamic thermal simulation programs. Nevertheless, from time to time other researchers have cast doubt on the reliability of their use in buildings. Under close examination these assertions have been found to be either an incorrect use of the Alamdari and Hammond formulae, or their application in situations where they are invalid (such as forced convection heating). Part of the present study was therefore aimed at confidence building in the use of these correlations; in this case for passive solar design purposes.

In order to achieve the project aims, a collaborative team with differing, but complementary, expertise was engaged in the work. These included participants with expertise in thermo-fluid dynamics (Peter Arnold and Geoff Hammond at Bath), test cell facilities and measurement instrumentation (Chris Martin of the Energy Monitoring Company: EMC), and expertise in multivariate time series analysis (Andy Irving at RAL). An innovative design of a test room was utilised in which the temperature of the room surfaces was controlled, except for the south-facing, glazed facade that is subject to natural climatic variations. This advanced EMC Test Room 3000 was designed, fabricated and commissioned at Cranfield during the course of the present study *(and was described in two articles published in the Supplements to Issues 11 and 12 of the BEPAC Newsletter in Autumn 1994 and Spring 1995 respectively - Ed.)* The resulting sun patch formed on the side-walls and/or floor (see Figure 1) was tracked over 30 days of field measurements.



Case 1 Sun Patch Wholly on Floor

Figure 1. Typical sun patch position

The solar input was measured by employing a net radiometer on the south-facing vertical facade. Other climatic variables were monitored from the EMC local weather station. Surface heat exchange was determined using (a) measured surface temperatures and heater mats, and (b) Mayer ladder arrays. Bulk air properties such as velocities and temperatures were also measured using three novel GILL 3-axis ultrasonic anemometers. They were employed to measure both the speed and direction of the room airflow, and were interfaced to a PC so that data could be logged at regular intervals over a long period.

The data has been analysed using two approaches. Firstly the data was derived as

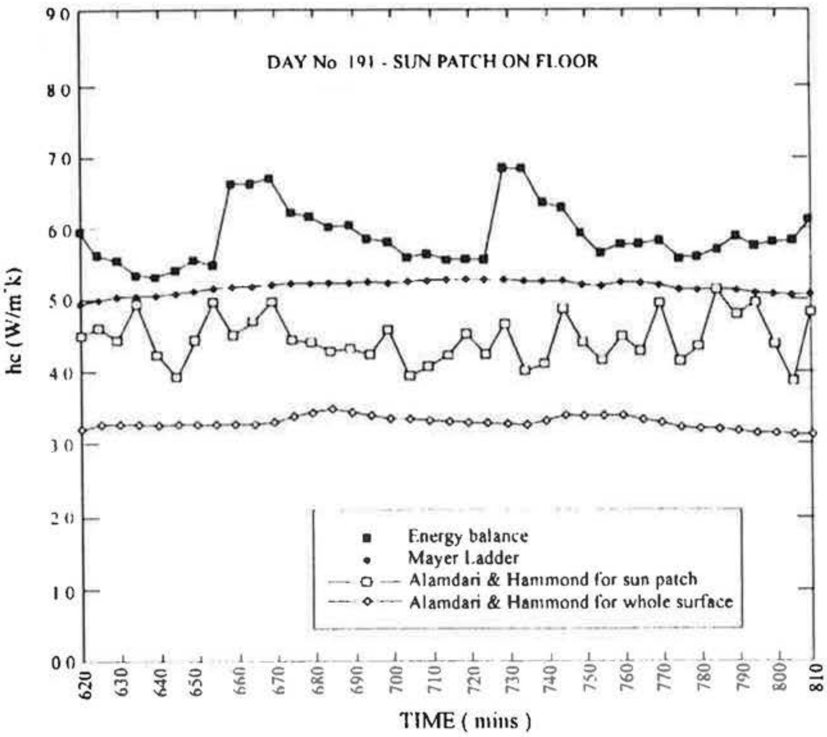


Figure 2. Measurements of sun patch convective heat transfer coefficients (h_c)

conventional single-sample thermo-fluid experiments, with the results compared to the correlating equations of Alamdari and Hammond (1983). The latter correlations have been shown to produce widely differing results depending on whether they are applied to the whole building surface (as is commonly done in building energy simulation programs), or separately to the sun patch (see Figure 2 for some preliminary, unfiltered comparisons). For horizontal surfaces at least, applying the correlations to the sun patch seems to produce better agreement with measured values. Ignoring buoyancy-driven convection from the sun patch is likely to have significant implications on building thermal models since not only is the heat transfer coefficient higher than for the surrounding surfaces, but the convective heat flux also is greater due to the larger surface to air temperature difference. Building energy modellers would still need to make their own judgement as to whether it is necessary to simulate the sun patch separately from its surroundings. This is because the additional complexity and higher computing costs are not trivial. Consequently, modellers need to assess whether these are worthwhile paying in order to gain physical realism in terms of simulation of the convection processes.

The second method of analysis used two time series analysis techniques: the analysis of the heat transfer and fluid flow were treated separately. EMC collected suitable experimental data to investigate the nature of buoyancy-driven convection above a sun patch and to examine the effect of a conservatory on the resulting fluid flow (see Figure 3). Initially the heat transfer was considered, the data being analysed to determine the linear and non-linear, dynamic and steady-state surface heat transfer coefficients. The surface heat transfer coefficient provides an important thermal link between the mass of the building and the air volume. In practice it has an influence on room temperature control, thermal comfort and room preheating in intermittently heated buildings.

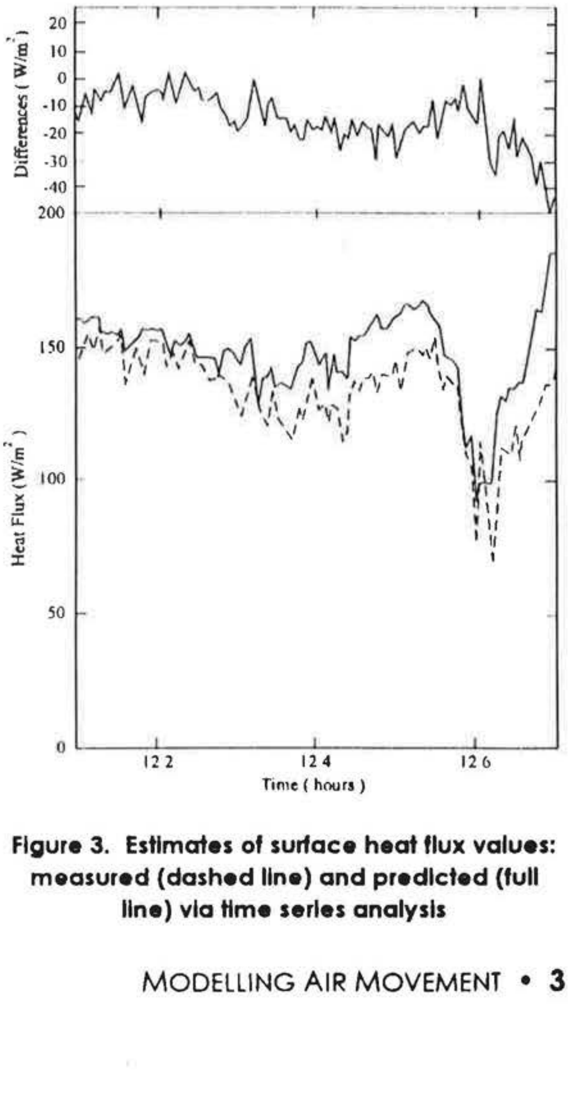


Figure 3. Estimates of surface heat flux values: measured (dashed line) and predicted (full line) via time series analysis

Computational air flow modelling techniques were also developed to aid the interpretation of measured data. The incompressible Navier-Stokes equations were numerically solved using the SIMPLE and SIMPLEc algorithms, and a modified algorithm aimed at buoyancy-driven flows was developed using non-linear FAS multigrid acceleration. This numerical solution procedure was extensively validated for both accuracy and performance on a buoyancy-driven test problem. The results indicate the modified algorithms have significant performance advantages over other segregated methods, particularly when used in conjunction with multigrid acceleration. They represent the complex flow induced by the sun patch well, and a typical example of the corresponding surface heat flux is shown in Figure 4.

The major outcome of the project has been guidelines on the treatment of buoyancy-driven convection in passive solar buildings. These are based on the use of the extended correlating equations of Alamdari and Hammond (1983), which have been implemented in the main commercial building energy simulation programs (including APACHE, ESP and TAS). This will lead to more reliable use of the correlations, which has only been made possible by the interaction of the experimental and computational modelling aspects of the study.

The work on which this article is based was supported by EPSRC research grants GR/F91988 and GR/G29571. The authors are grateful to Keith Horsley (formerly an undergraduate at Bath, and now with Hoare Lea and Partners, Bristol) who produced Figures 1 and 2. Details of this work are currently being prepared for journal publication, and readers requiring further information should contact Geoff Hammond in the first instance:

Professor G P Hammond, School of Mechanical Engineering, University of Bath
Bath BA2 7AY. ☎ 01225 826168, fax 01225 826928, e-mail: ensgph@Bath.ac.uk

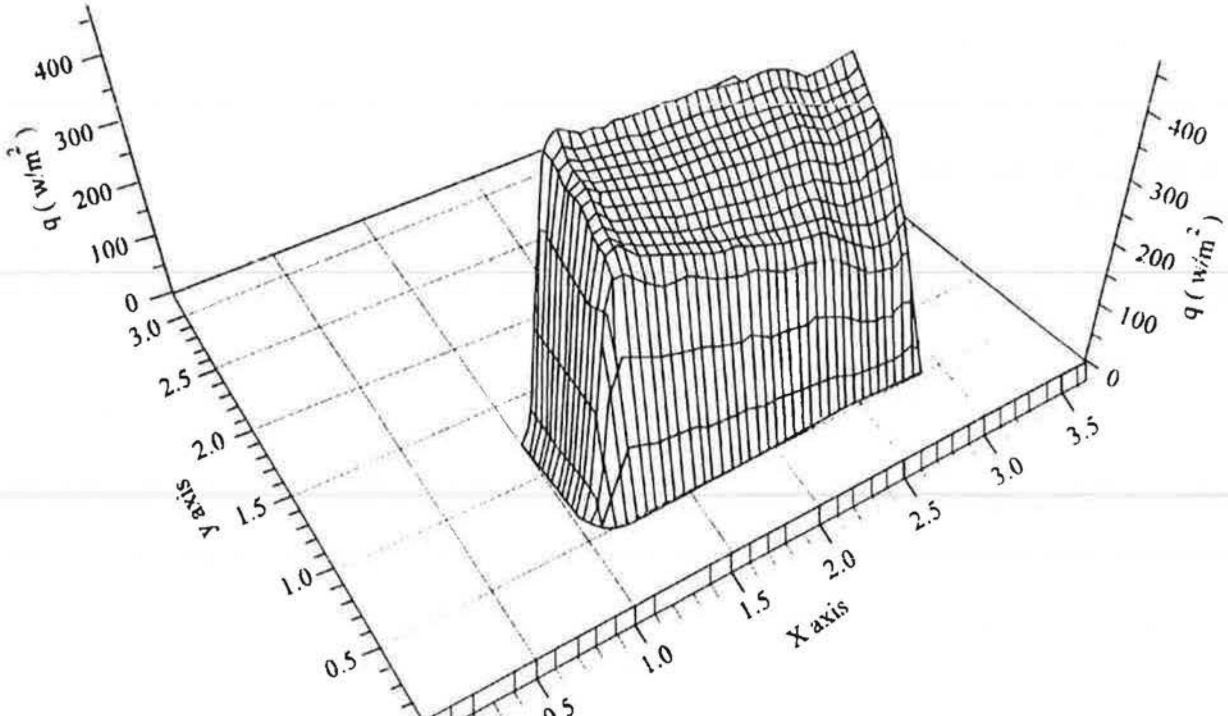


Figure 4. Floor surface heat flux map obtained from turbulent flow modelling

CFD airflow modelling in buildings

F Alamdari, BSc, MSc, PhD, CEng, FCIBSE
BSRIA, MicroClimate Centre, Old Bracknell Lane West, Bracknell, RG12 7AH, England.

Introduction

There is a growing interest among architects, building services and environmental engineers, and building owners/occupants in the prediction of air movement, thermal comfort, ventilation effectiveness, smoke and airborne contaminant movement in buildings before being built or occupied. Not surprisingly, this interest has grown with the development of design standards in indoor air quality, thermal comfort and safety, and the higher expectations of the building users.

Basically, the prediction of airflow and related processes in buildings may be obtained by either physical modelling using full size or scale mock ups, or computer modelling techniques. The latter approaches employ either macro- or micro-climate mathematical modelling:

- Macroclimate or multi-zone models in their most complex form allow the calculation of bulk air flows into and within buildings based on a combination of wind pressure, stack effect, zonal temperature differences and purpose designed mechanical ventilation systems. Although they do not provide detailed air distribution patterns, they may be used to estimate ventilation rate, infiltration and trace the general pattern of air flow within a building[1,2].
- Microclimate or CFD models allow the detailed prediction of airflow and related phenomena in and around buildings. The following intends to briefly review the background and application of these models in the context of the built environment.

What is CFD?

Building airflow and associated processes such as temperature and pressure distribution and contaminant concentration are governed by the principles of conservation of mass, momentum, thermal energy and chemical species. These conservation laws may each be expressed in terms of 'elliptic' partial differential equations*, the solution of which provides the basis for a microclimate flow model. Based on computational fluid dynamics (CFD) techniques these models solve, numerically, the governing conservation equations in order to generate field values for the velocity components, the static pressure, the temperature and concentration of chemical species.

The governing time-averaged equations for the turbulent flow, thermal energy and chemical species may be represented by a common form, using tensor notation[3].

$$\partial(\rho\phi)/\partial t + \partial(\rho u_j \phi)/\partial x_j = \partial\{\Gamma_\phi (\partial\phi/\partial x_j)\}/\partial x_j + S_\phi \qquad (1)$$

where u_j (u_1, u_2, u_3) are the time-averaged (mean) velocity components in co-ordinate direction $x_j(x_1, x_2, x_3)$, ϕ are any of the dependent variables, Γ_ϕ are the effective diffusion coefficients, S_ϕ are the sources or sinks, ρ is the air density, and t is time. Mathematical expressions for the diffusion coefficients and source terms for each variable can be found elsewhere[4].

In turbulent flows, the closure of the equation set can be obtained by an appropriate turbulence model. A two-equation turbulence model, namely the k-ε model[5], has been mainly used in the context of the built environment. This model defines the turbulent exchange coefficient for momentum from two additional variables, the turbulence kinetic energy and its dissipation rate, that characterise the local state of turbulence.

The finite-volume CFD solution of the above equations requires some discretization of the flow domain into a finite set of cells or control-volumes formed by computational grid. The integration of differential equations over each control volume leads to a set of algebraic equations in the following finite-volume form:

* 'Elliptic' is a term used in the classification of governing differential equations and corresponds to a two-way computational concept in which the conditions at a given location in that co-ordinate are influenced by changes in conditions on either side of that location.

$$(a_p - S'_p)\,\phi_p = \Sigma_n\, a_n\, \phi_n + S_p \qquad (2)$$

$$a_n = D_n \alpha(/Pe_n/) + <0, \pm C_n>$$

$$a_p = \Sigma_n\, a_n$$

$$S_\phi = S_p + S'_p\, \phi_p$$

where indices p and n denote the central computation grid point and its neighbouring points respectively, C and D indicate the strength of convection and diffusion respectively, $\alpha\ (/Pe_n/)$ is a weighting function, and the symbol $<>$, denotes the greater of the enclosing terms.

These algebraic equations are strongly inter-linked with no obvious equation available for the static pressure which appears in the momentum equations. The solution procedure employed by most finite-volume CFD models is the SIMPLE (semi-implicit method for pressure linked equations) calculation procedure or a modified version of this[3]. This involves an initial guess of the field values and then an iterative solution and correction procedure. The iterative solutions continue until the imbalance or error in the equations is sufficiently small to be considered negligible.

What does CFD offer?

In general CFD simulation can offer the detailed 'field' prediction of fluid flow and heat transfer. Taking the advantages of the speed and power of new computers, CFD models offer a powerful and versatile computer-aided-engineering design tool.

In their most general form CFD codes can *potentially* be applied to a wide range of fluids and related phenomena in complex three-dimensional situations, including: steady and transient; laminar and turbulent; buoyant; incompressible and compressible; convective, conductive and radiative heat transfer; mass transfer; chemical reaction; combustion; multiple streams; and multi-phase fluids.

Although most of the commercially available 'general-purpose' CFD codes offer (or at least intend to offer) the majority of the above capabilities, 'special-purpose' CFD software only includes those features which are most appropriate to the purpose.

In the context of the built environment, CFD models may be used as an efficient tool for indoor and outdoor thermofluid analysis:

- *Indoor airflow simulation and thermal comfort*

  The environmental parameters that are significant in thermal comfort are air temperature, radiant temperature and air velocity. These parameters are mainly influenced by air distribution in building spaces. CFD allows the performance of naturally and mechanically ventilated, and air-conditioned spaces to be analysed by predicting the values of these parameters in the entire space. These values may then be used in thermal comfort models to analyse the level of the occupants' satisfaction with their environment under the predicted field conditions. Figure 1 shows the predicted airflow and thermal field within an office space incorporating a displacement ventilation system.

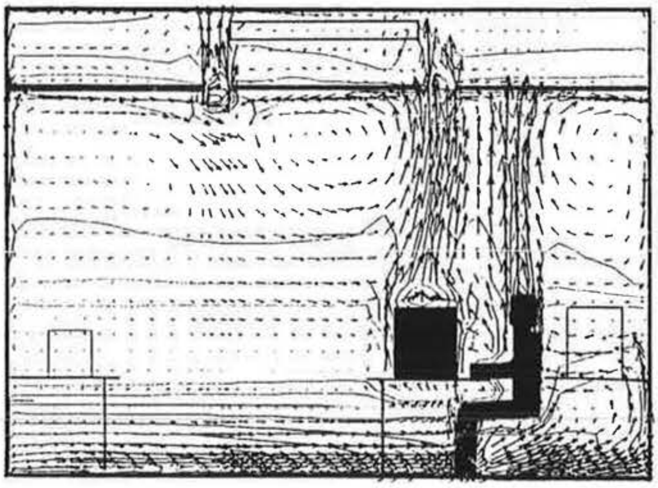


Figure 1. FLOVENT special-purpose CFD code used to predict air movement and thermal field in a room with displacement ventilation

- *Ventilation effectiveness*

  Indoor air quality and the effectiveness and performance of ventilation systems in removing body odours, environmental tobacco smoke, organic compounds, ozone released from photocopiers and printers, fumes, smoke as a result of fire and products of combustion, gaseous from internal and external pollutants, etc. can be analysed. Figure 2 displays the predicted smoke movement within an atrium.

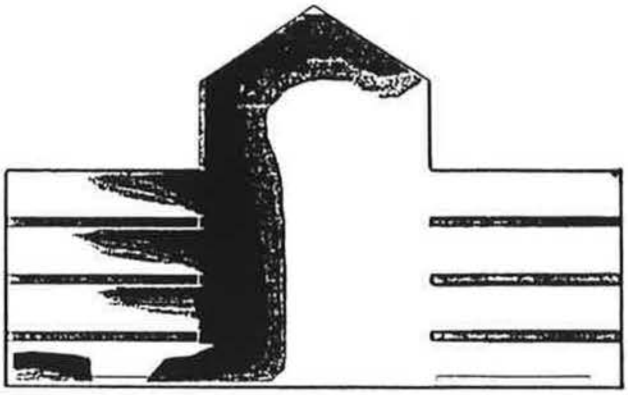


Figure 2. FLOVENT special-purpose CFD code used to predict smoke movement in an atrium

- *External airflow and pollution*

  CFD wind tunnel simulations may be used to assess external aerodynamics. For example, where exhaust stacks emit potentially hazardous contaminants to identify the likely dilution and passage to surrounding districts or to predict natural ventilation rates for critical applications, such as offshore platforms. Figure 3 shows the predicted air flow around buildings.

What does CFD require?

Domain discretization

The first step in the numerical solution of Equation (2) is the discretization of the flow domain. Domain discretization must conform to the boundaries in such a way that boundary conditions can be accurately represented and allow the computational cells to be small in regions with steep property gradients, such as near the walls and obstacles. Domain discretization is provided by a computational grid generated using an appropriate co-ordinate system.

Although most building geometries may be discretized by using simple rectangular cells and adopting a simple Cartesian co-ordinate system, some difficulties may arise in specifying angled or curved surfaces. Those finite-volume CFD codes employing only rectangular cells represent such surfaces by a best 'step-wise' fit to a grid and blocking off unwanted cells (see for example figure 4). Others may include a boundary-conforming co-ordinate system to generate a grid to fit irregular boundaries.

The accuracy of the solution obtained by CFD simulation relies heavily upon the degree to which the solution depends on the grid arrangement Therefore, in buildings with interconnected zones with various geometries, irregularities and features of different sizes (for example small diffusers in large spaces), it is desirable to adopt a flexible three-dimensional body-fitted grid generation with local embedded mesh refinement capabilities, combination of unstructured and arbitrary interfacing meshes (see Figure 5). Although this enhances the accuracy of the solutions for such complex geometries it increase the complexity and hence reduces the user friendliness characteristics of CFD.

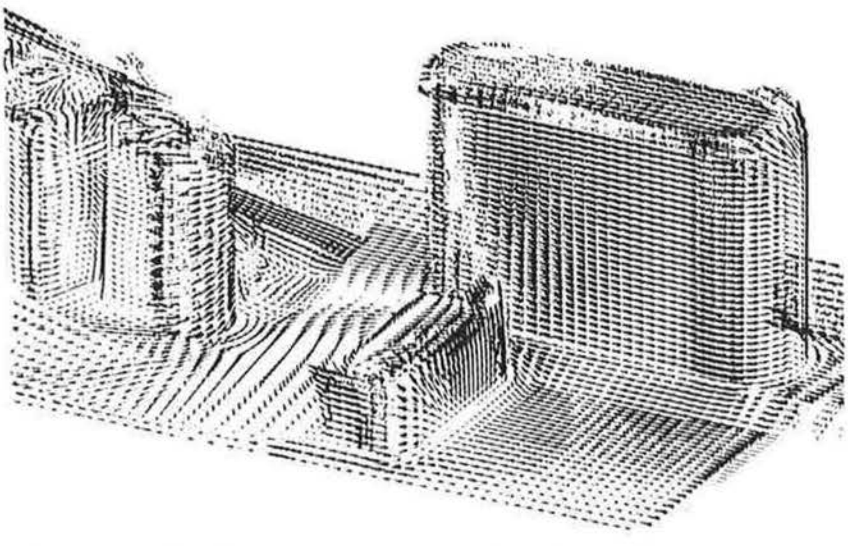


Figure 3. STAR-CD general-purpose CFD code used to predict airflow around buildings *(courtesy of Computational Dynamics Ltd)*

Boundary conditions

These are problem specifications, the conditions for which CFD simulations are required. These boundary conditions could be a specified value of variables or value of the associated flux, or a relation between the variable and the flux.

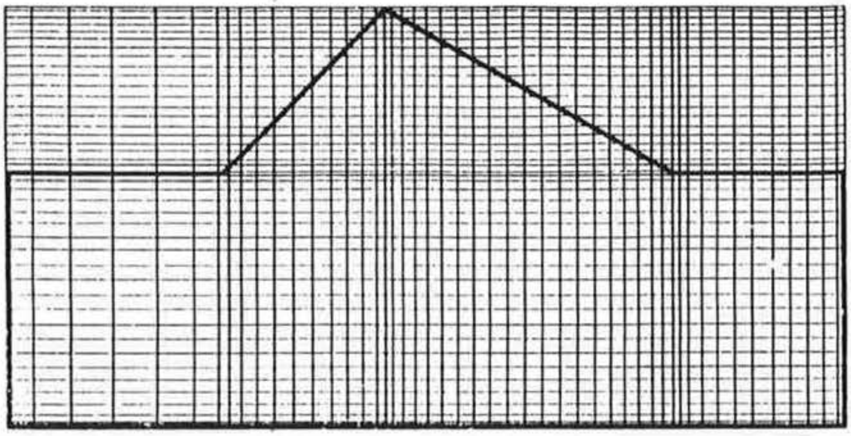


Figure 4. Simple Cartesian computational grid

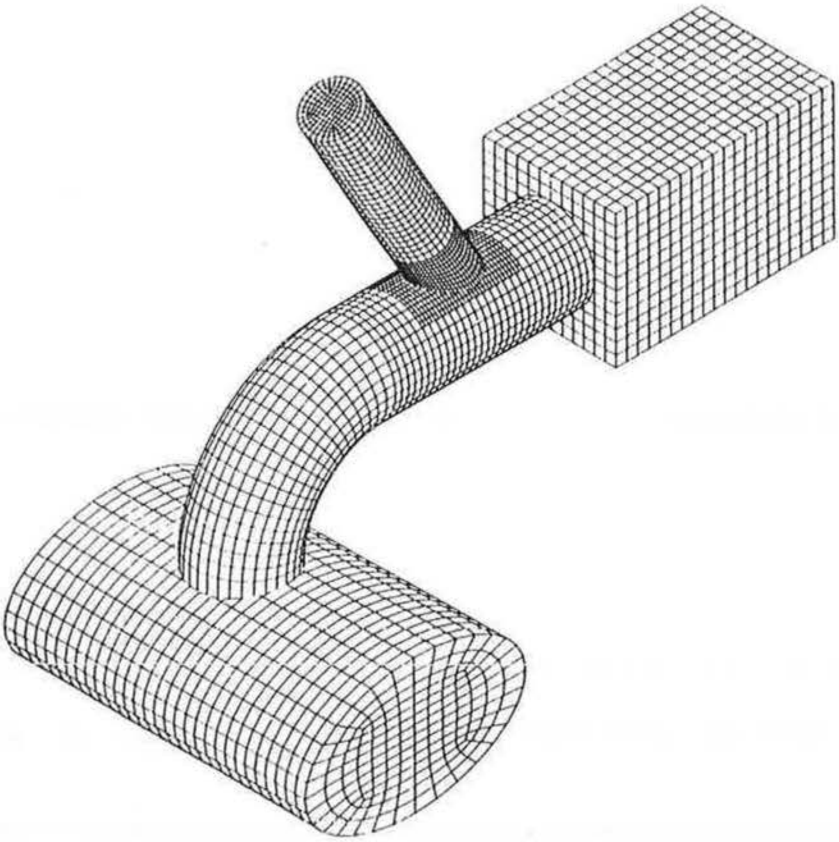


Figure 5. Body-fitted unstructured grid with local mesh refinement *(courtesy of Computational Dynamics Ltd)*

Special-purpose CFD software may incorporate a built-in library of boundary conditions for common building elements (ie window, external and internal walls, etc), HVAC systems (such as supply, extract unit, heater, fan-coil unit, etc), internal features (for example furniture, people, equipment, etc). This eases the input data required for simulations.

Solution control

CFD provides the numerical solution of a set of non-linear, interlinked equations in an iterative manner. In this method, it is necessary to examine how the current solution approximates to the exact solution of equations at the end of each iteration, for each variable. In principle, the iterative solution should handle any non-linear and interlinked equations. In practice, however, the use of some form of under-relaxation is found to be necessary. This practice reduces the magnitude of current error level in the field, while preventing the divergence in the iterative solution of the equations. The problem here is that there are no general rules for describing the optimum values of under-relaxation as they are problem dependent and can only be found by experience.

What does CFD provide?

The advances in computer graphics have allowed CFD software to provide full colour displays of field environmental parameters, which make it possible to view the geometrical features of the problem, the distribution and resolution of computational grid, monitoring the convergence and display the results. The latter includes plots of velocity vectors, contours and/or cell-filled of speed, temperature, pressure, and concentration of species on pre-defined 2D sections within the flow domain and on 3D domain surfaces.

Visualisation aids such as rotation, zoom, multiple field displays, streamline tracking, cursor pointing and display data, text annotation and editing, and video effect displays are almost becoming standard for most CFD codes.

What are the limitations?

Geometrical aspects

Geometrical aspects of buildings could become extremely complex. Architectural features, internal layouts and furniture, equipment, people, air terminals, etc. often create a flow domain which is a real challenge for computational grid generators. For this reason often some compromises may be necessary depending on the scale of geometrical complexity in practice. Further due to the large flow domains and the existence of small, but important, boundary and internal features the size of the grid may become enormously large in some applications. For example, air terminals (ie grilles and diffusers) in building spaces dominate the room air diffusion. It is therefore important to represent these devices accurately. But, the complex geometrical configurations of diffusers means the requirement of local mesh refinement and large grid cells. These devices are often represented roughly, therefore, the reliability of simulated air diffusion in rooms is questionable. There are several methods for simulation of diffusers, for example based on the wall-jet theory[6,7], or near-field measurements[8].

Boundary conditions

In buildings, the time-dependent interaction between the external conditions, building fabric and internal conditions, means that the domain boundary conditions may vary with time. Since CFD models require extensive iterative solution on very small time scale, often some compromises are made on the scale of transient interaction or indeed boundary conditions are often assumed to be steady-state, based on the analytical or experimental results, except when CFD is used to predict the dispersion time of gases or smoke; for example, in evacuation time analysis.

In turbulence flow, viscous and turbulent stresses are of the same order of magnitude near the wall. This forces the values of turbulent transport properties to fall to their laminar values and the result is a steep, non-linear variation with distance from the wall in dependent variables and their gradients. Most CFD codes use the so-called 'wall-function'[5] to bridge the steep property gradients. These are not appropriate for recirculating flows and will give rise to error in, for example heat exchange coefficients near walls.

Turbulence

The transient convection, conduction and radiation heat transfer processes in and around buildings give rise to very complex fluid flow problems. The airflow around buildings are usually turbulent flows, while within buildings are mainly in the transition range. Even in the turbulent regime, there is currently no universal turbulent model available that can reflect the behaviour of the full range of complex turbulent flows observed in buildings.

Turbulent flows may be modelled by one of the three approaches: direct simulation, large-eddy simulation and turbulence transport models. Direct simulation involves direct solution of the time-dependent governing turbulence equations without assumptions. The number of computational grid points required in such calculations is beyond the capacity of today's computers. The large-eddy simulation assumes, large scale motions are primarily responsible for all transport processes, such as the exchange of momentum and heat, therefore, eliminating the small-scale motions (ie smaller than the grid size). Although the modern computers can handle this model, computational time is very long due to the three-dimensional fine grid and transient simulation requirements, and currently is inappropriate for building airflow simulations. Turbulence transport models represent dynamic turbulent quantities by time-averaged turbulent field, therefore, simulate only the overall features of the turbulent flows. In practice acceptable flow fields were predicted using these models. A most popular model in this category is the k-ε model[5] and is employed by almost all CFD codes as the default option for turbulence model.

In building spaces, if the thermal and flow boundary conditions are defined correctly, the k-ε model most likely is able to predict turbulent airflows for practical applications.

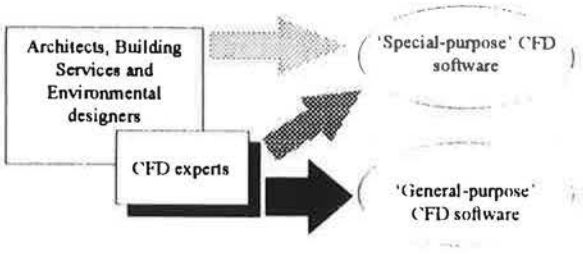
Expertise and resources

Microclimate modelling of the built environment is complex, time consuming and usually require considerable resources and expertise in the field of thermophysics and thermofluids for obtaining meaningful results. There are many factors that can influence the predicted results from CFD, and the knowledge and expertise of the user play a significant part in the accuracy of the predicted results. Indeed, different results may be expected by different users even from the same CFD software.

The execution time is highly dependent on the performance of the computer hardware and also software solver. Although, most of the CFD codes may be run on high performance PC's their application is limited. For practical situations, high performance computer workstations are essential.

In recent years advances in computer graphics allowed the provision of a new generation of 'special-purpose' CFD software which are focused to application areas, providing required functionality and using the industry terminology. Although, such tailored CFD products should, potentially, eliminate the mathematical modelling knowledge, the expertise in building thermofluid remains essential to ensure the appropriateness of the predicted results.

In any case, for complex building airflow problems the resources of skilful CFD modellers and researchers are necessary, working together with architects, building services and environmental designers.



References

1. Liddament M W and Allen C M, AIC Technical Note 11, AIVC, UK.
2. Holmes M J and Salusbury, 9th AIVC Conference, Gent, Belgium, 1988.
3. Patankar SV, Numerical heat transfer and fluid flow, 1980, Hemisphere, Washington.
4. Alamdari F, Hammond G P and Mohammad W S, Proc. CIB Int. Symp. Use of Computers for Environmental Engineering Related to Buildings. Bath, 1986, CIB/CIBSE, London.
5. Launder B E and Spalding D B, Computer Methods in Applied Mechanics and Engineering, 1974.
6. Nielsen P V, Restivo A and Whitelaw J H, ASME Journal of Fluid Engineering, 1978.
7. Alamdari F, PhD thesis, Cranfield University, 1984.
8. Salemi R and Alamdari F, BSRIA 77230 Report, 1994.

COMIS validation results - a summary

•J-M Fürbringer, •C A Roulet and #R Borchiellini
•LESO, EPFL, CH-1015 Lausanne, Switzerland
#Dip. di Energetica, Politecnico di Torino, C.so Duca degli Abruzzi, I-10129 Torino,Italy

IEA-ECB&CS Annex 23 program intends to provide a validated and user-friendly computer code, called COMIS, which simulates air flow patterns in multizone buildings. When developing such a code, which intends to model reality, it is essential to check that it implements the chosen algorithms correctly at each step in the calculation. At the other end, the claimed user-friendliness should also be checked with real users.

This evaluation task took a large part of Annex 23's resources. The result is that COMIS was checked very carefully, using up-to-date strategies and tools. The final report presents the methods used and the results of this huge validation task.

The Annex has produced both a new validation and user test strategy and new methods and computer tools for sensitivity analysis of simulation code.

Simulation results were compared with more than 50 simple test cases, for which either an analytical or a numerical solution was obtained using classical tools. Each of these test cases was created to check a particular feature of COMIS. This so-called analytical evaluation allowed the correction of several bugs which appeared in the early versions of COMIS.

Inter-model comparison with as much as 12 other simulation programs was performed by five different laboratories, using various objects. In each case, the objects were adapted or chosen in such a way that they could be modelled by the program. The comparisons showed that all the models produced the same results, within a very narrow dispersion band. This is the expected outcome, because all the models use similar algorithms and simulations were performed with identical input data. However, it also shows that, at least for the checked features, these models do not contain bugs.

The experimental comparison task conducted within Annex 23 is probably the largest ever performed for a computer code. Nine different buildings were monitored for this purpose, each building offering several cases for comparison. A sensitivity analysis was performed for each case, in order to reveal not only the uncertainties in the measurements, but also the uncertainties in simulation results which result from uncertainties on input data. These were found to be very large. This meant that the differences between the measurements and simulation results were, in some cases, not statistically significant. In other cases, however, important differences were found, showing errors in either the model or in the measurements.

The largest differences were found in the user test. Two cases were submitted to several different users and results were compared. One case was simple and clearly defined, with all essential input data provided. For this case, all users but one (who made a modelling error) provided the same results. The other case, however, was more realistic, since data were provided as usually available in practice: only building plans and some measurement results. The user not only had to design the network model, but also had to choose some essential input data, notably the pressure coefficients. Very large differences in the results were found in this instance. Most of the discrepancy can be explained by modelling errors which are attributable partly to unclear instructions in the draft User Guide (since corrected), and partly to differences in input data.

The limits of applicability of Comis were not found although they may exist. But other limitations were found by the user: the uncertainty of input data and the way a particular case is modelled have a large effect on the result. Annex 23 gives model users a good understanding of the way uncertainties in input data will contribute to the uncertainty of simulation results.

The final report on evaluation will be available soon; some draft copies are still available from the authors.

J-M Fürbringer can now be contacted by e-mail at: furbringer@nist.gov, C A Roulet at: roulet@eldp.epfl.ch and R Borchiellini at: borchiel@polito.it

CFX simulations of building ventilation and aerodynamics

Nathalie Hamill
AEA Technology

Accurate predictions of air flows in and around buildings are necessary for the successful design of heating, ventilating and air conditioning (HVAC) systems. Computational fluid dynamics (CFD) offers an economical alternative to many of the more traditional methods based on practical experience, empirical estimation and expensive air conditioning laboratories and/or wind tunnel experimental tests.

The CFX software from Computational Fluid Dynamics Services (CFDS) is used routinely by a number of major companies in the building industry. Founded on more than 20 years experience in CFD modelling, CFX (formerly CFDS-FLOW3D) is a general-purpose CFD code which boasts geometrical flexibility, advanced physical models and powerful numerical methods. A CFX simulation predicts the velocity fields, pressure and temperature at all points within or around the building, data which can then be visualised with the state-of-the-art graphical post-processors in the software. The resulting images provide the designer with a clear and detailed understanding of the efficiency of the HVAC systems, information which would be prohibitively expensive or impossible to obtain experimentally.

Furthermore, CFX enables coupled and/or decoupled solutions of the following classes of building aerodynamics problems:

- global simulation of external air flow fields for the whole building as well as for relevant parts of its surrounding area
- detailed simulation of air flows in enclosures which are exposed directly to the action of the external air flow field
- detailed simulation of air flows in enclosures which do not experience directly the action of the external air flow field.

The necessity of the coupled numerical treatment of the sub-topics mentioned above

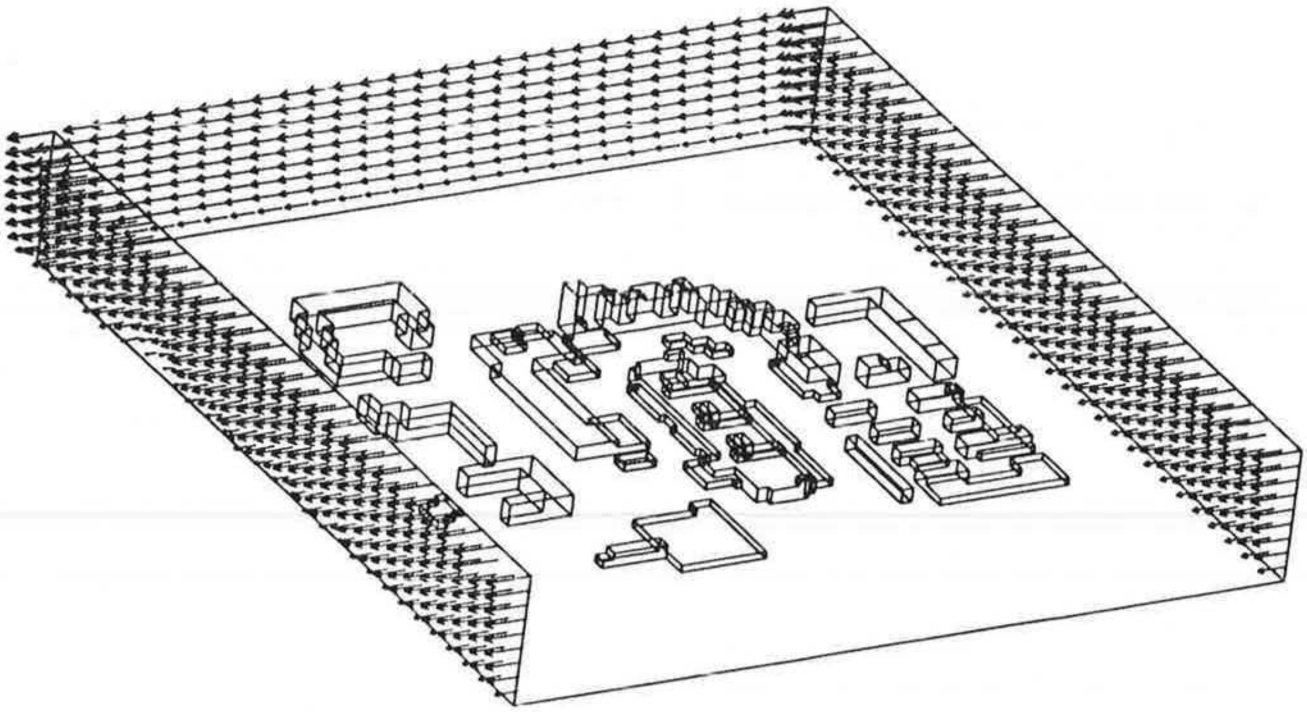


Figure 1. Computational domain of the school and surrounding buildings

depends on the intensity of the aerodynamic and heat transfer interaction effects between the building, single enclosures and their outdoor and/or indoor environment.

Numerical air flow simulations are especially useful in the design of HVAC systems for large enclosures, such as exhibition halls or atria, where it may be impossible to produce scale experiments which accurately represent both forced- and natural convection effects. In relatively complex buildings such as schools, offices or industrial buildings, experimental techniques can provide only a limited amount of data due to the difficulty of fully instrumenting the geometry. CFD however allows complete and unobtrusive analysis of all aspects of the flow. Early consideration can therefore be given to the air conditioning requirements regarding: velocity, temperature and purity of air, turbulence intensity as well as the avoidance of cold draughts in occupied zones. However, perhaps the most important advantage of the CFD-aided approach is the possibility of studying a greater number of alternative design concepts at the preliminary design stage.

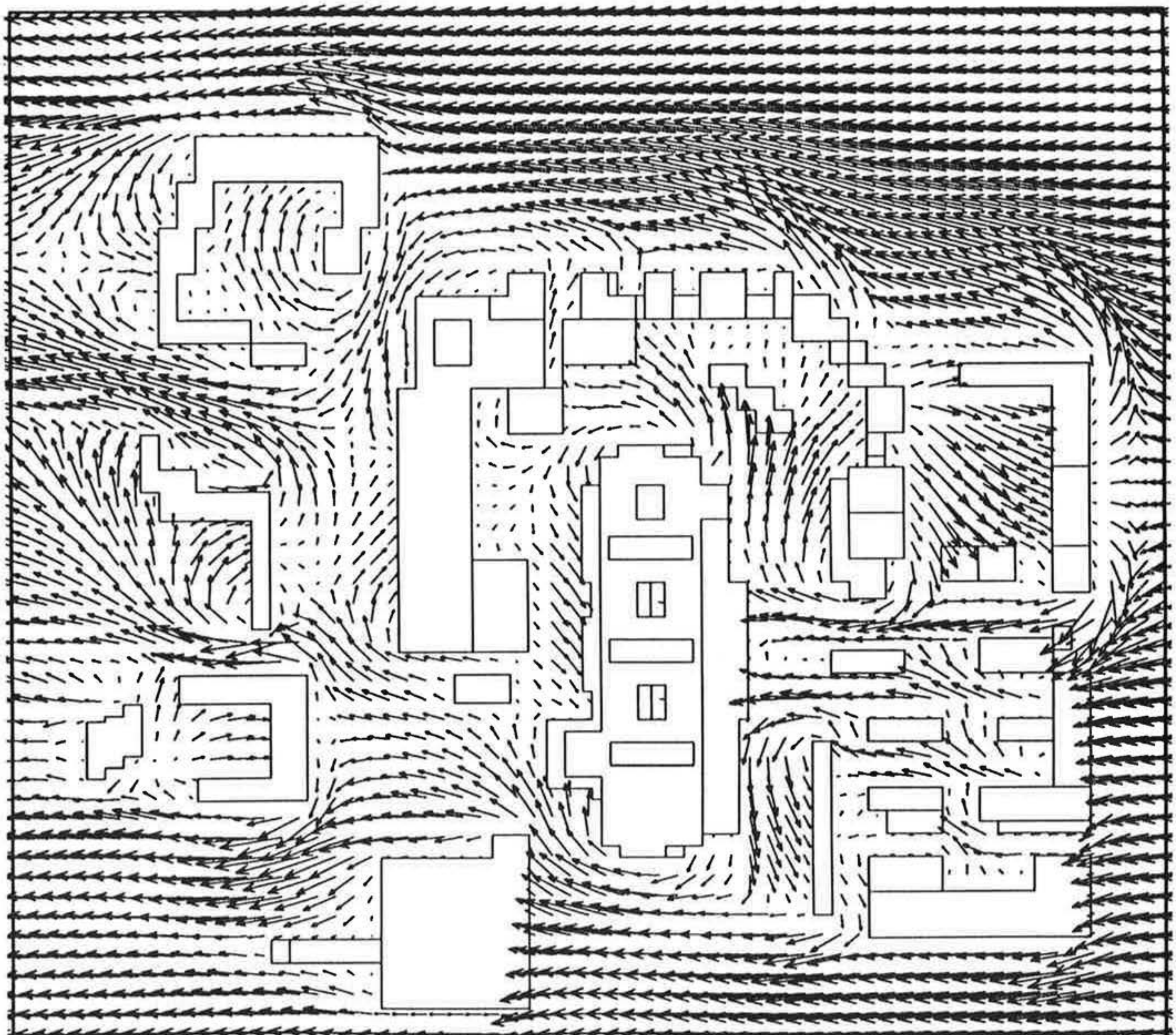


Figure 2. Wind speed at ground level around the school

ROM (Rud. Otto Meyer), in Germany, use CFX routinely for the simulation of HVAC systems. Their latest study involves the natural ventilation of an existing school building which is surrounded by several high residential buildings of between 2 and 14 storeys. These buildings strongly affect the wind induced velocity and pressure distribution within the school area. CFX was used to predict the naturally induced air exchange for some representative classrooms exposed to different environmental and operational conditions. The global computational domain of the school and the surrounding buildings is shown in Figure 1. Strong wind-shadowing effects of the surroundings, which can be observed in Figure 2, lead to relatively small dynamic pressure differences between the windward and leeward sides of the school building and lead to reduced ventilation through the classrooms.

In many practical cases, the effectiveness of natural ventilation is strongly determined not only by the interaction effects between the building and its outdoor environment but also by the enclosure-to-enclosure interactions which occur within the building. These are very important for the air flow behaviour in glazed atria with open office galleries. In an illustrative case, also performed by ROM, the open office galleries are connected directly with the glazed atrium and have windows opening onto the building's external facades. The computational mesh, which comprises the atrium together with the adjoining offices and corridors, is shown in Figure 3. It can be seen from Figure 4 that the atrium experiences intensive outdoor air flow infiltration as soon as some windows are opened. Unfortunately, the incoming external air flow causes strong draughts in the atrium's occupied zones, and this increases with the growing open window area. However, such situations can be avoided successfully during the building's operational life, provided that the critical window opening schemes are determined in advance. The ease with which large combinations of wind direction, speed and window openings can be investigated with CFX makes it particularly efficient for this type of study.

For further information about CFX please contact:

Gillian Rowe
Marketing Manager
CFDS
AEA Technology
8.19 Harwell
Didcot, Oxon OX11 0RA
tel: +44 (0) 1235 433771
fax: +44 (0) 1235 432989

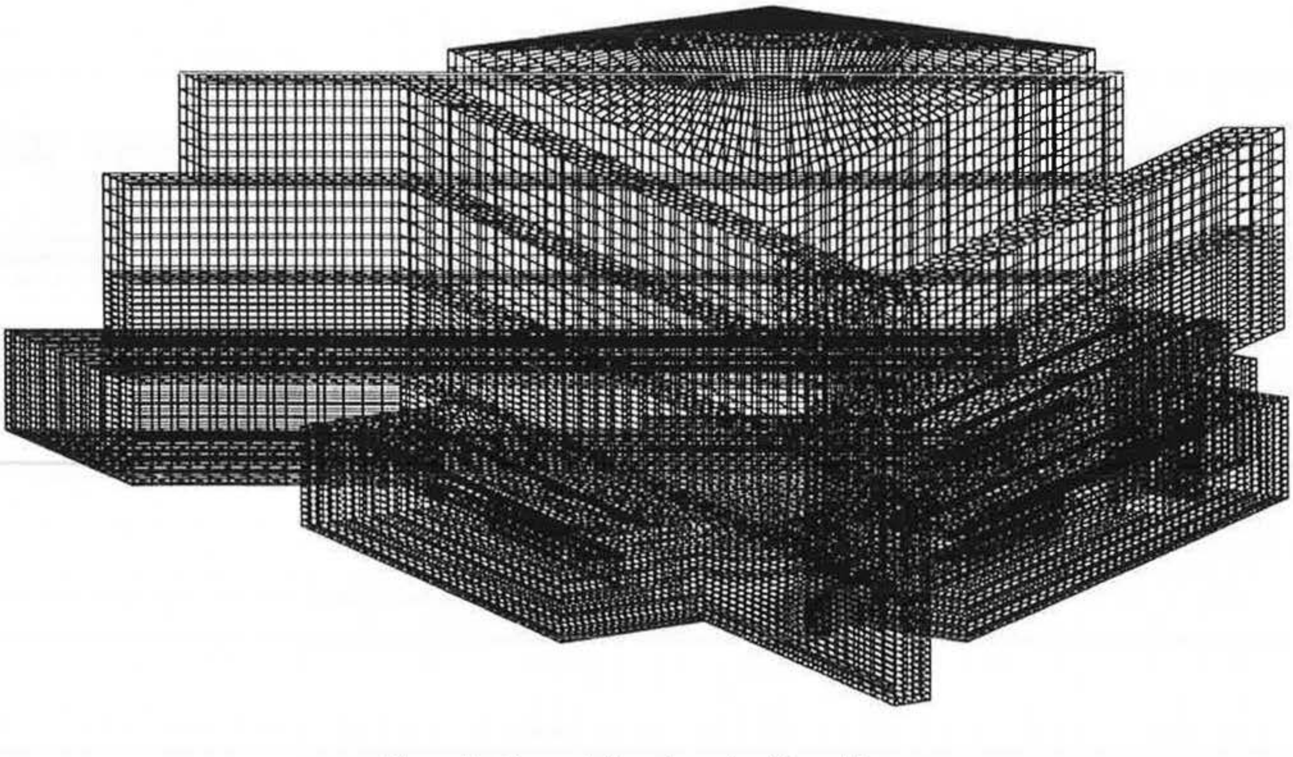


Figure 3. Computational mesh of the atrium

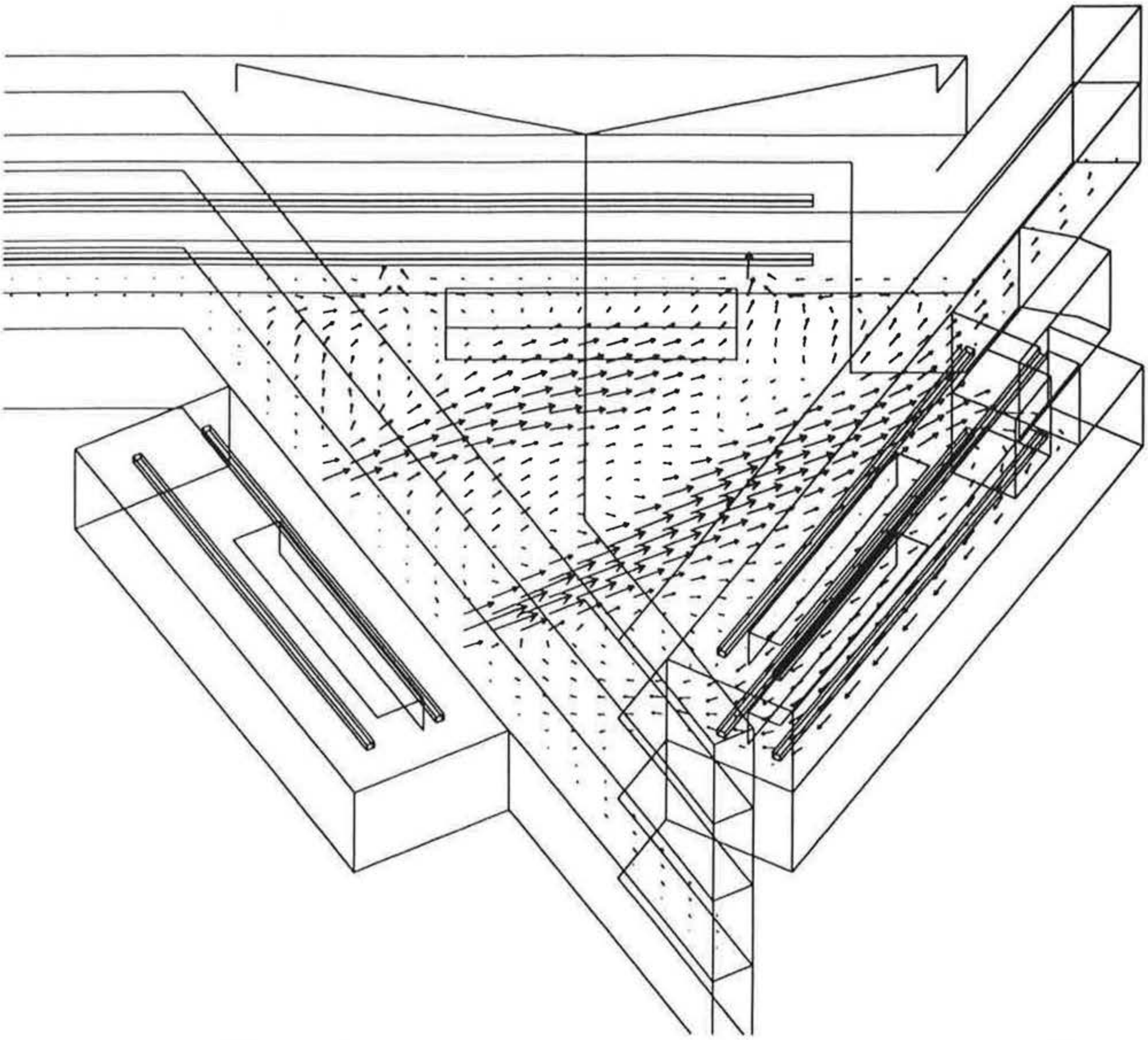


Figure 4. Velocity vectors in the atrium with some windows open

CHAM's PHOENICS-VR - a new approach to CFD air-flow simulation

Nic Cavanagh
Concentration Heat and Momentum Limited

Until recently, the simulation of air-flow and temperature, pressure or contaminant distribution in and around buildings and structures has relied on prototype experiments and/or conventional CFD (computational fluid dynamics) techniques. For the former, costly prototype models must be constructed and, unless these are built to full scale, scaling effects can add uncertainty to predictions.

For CFD, the solution methods used require highly trained engineers to set up the geometric and numerical models, to define the underlying grid ensuring that it is fine enough to produce sensibly converged solutions and to define often complex boundary conditions. Furthermore, the hardware required may be beyond the financial resources of some who are therefore deterred from using the technology.

CHAM believe an answer to all these objections is provided by PHOENICS-VR, an air-flow simulator which is easy to use and available on the PC. The code incorporates PHOENICS, the leading flow simulator developed in the UK by CHAM. It is available either as a stand-alone software product or may be accessed for a modest charge as part of an Internet based consultancy service.

The stand-alone PC based system requires a Pentium based machine with at least 16 Megabytes of RAM and VESA compatible SVGA graphics capabilities. For a typical simulation (containing 50,000 cells say) a converged solution could easily be obtained by running over-night. More complex simulations will require more time or may be solved using the PHOENICS-VR Internet service with CHAM's Parsytec X'plorer parallel machine as the CPU.

What is PHOENICS-VR?

There are a number of software packages presently available, such as the PHOENICS based FLAIR program, which are tailored to the application of CFD to problems encountered within the built environment. However, PHOENICS-VR is the first design simulator to use 'virtual reality' to address the real needs of the construction and engineering industries. It is envisaged that the exploitation of recent virtual reality tools and the advent of 'The Internet' will combine to radically change the fluid flow software and consultancy market.

With PHOENICS-VR calculations are carried out without the need to create a grid, this task being performed automatically. PHOENICS-VR may therefore be considered as a grid-less CFD analysis system which provides simulation capabilities to those who have little or no knowledge of the underlying technology. The user has limited control over the underlying grid configuration which may be defined as coarse, medium or fine. A simple algorithm is then used to calculate the number of cells in each region of the domain, where a region is identified by adjacent object boundaries, and hence the positioning of cells in the three Cartesian axis directions. In the majority of cases, this results in acceptable grid configurations, but under certain circumstances (e.g. for small gaps across which there are high velocity gradients) the grid may need to be refined manually.

The input data is defined as a virtual world representing the CFD solution domain, and comprising objects placed within that 'world' with 'boundary conditions' attached reflecting the input and output of heat, momentum (air flow) and contaminant to and from the world.

The output is a graphical representation of the air movement and temperature, pressure or contaminant distribution under a particular set of conditions. These may take the form of colour-coded vectors representing velocity or contours representing pressure, temperature and concentration. These can generally be displayed in sectional planes or 3D views.

The method of data input for the software is highly intuitive and mouse-driven. Furthermore, the relative cost of hardware is small because it is designed to run on a high-end PC, compared with many flow simulation products which require the power of expensive workstations.

Unlike other packages, PHOENICS-VR does not demand an intensive knowledge of CFD. The highly intuitive virtual reality user-interface allows the designer to set up his problem very easily, then solve it either locally or by transmitting to a remote site for processing. PHOENICS-VR allows users to try a variety of options in the most cost effective manner - speeding the analysis process and permitting optimal design.

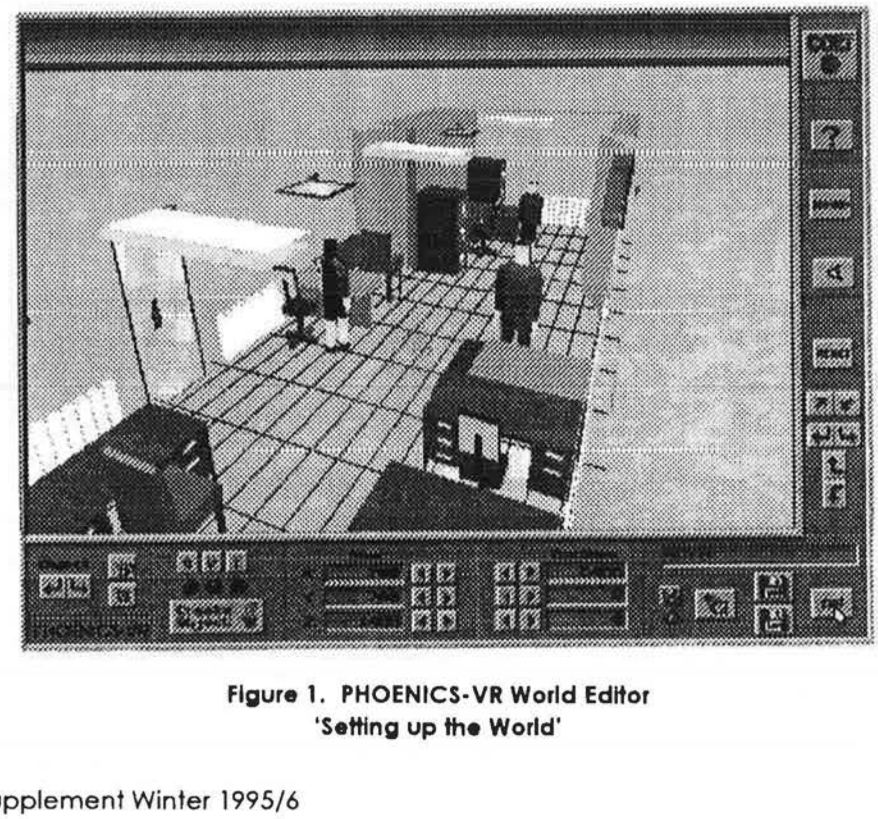


Figure 1. PHOENICS-VR World Editor 'Setting up the World'

The PHOENICS-VR graphical front-end, shown in Figure 1, allows the user to create 3D objects with easy to use dialogue boxes using an extensive library of clip-art, and assign to these the attributes required by the CFD solver. These attributes include size, position, temperature, heat output, material type and pollutant source concentration. The model can be viewed very effectively in the VR world using 'walk through' facilities provided by both the VR World Editor and the VR Results Viewer, as shown in Figure 2.

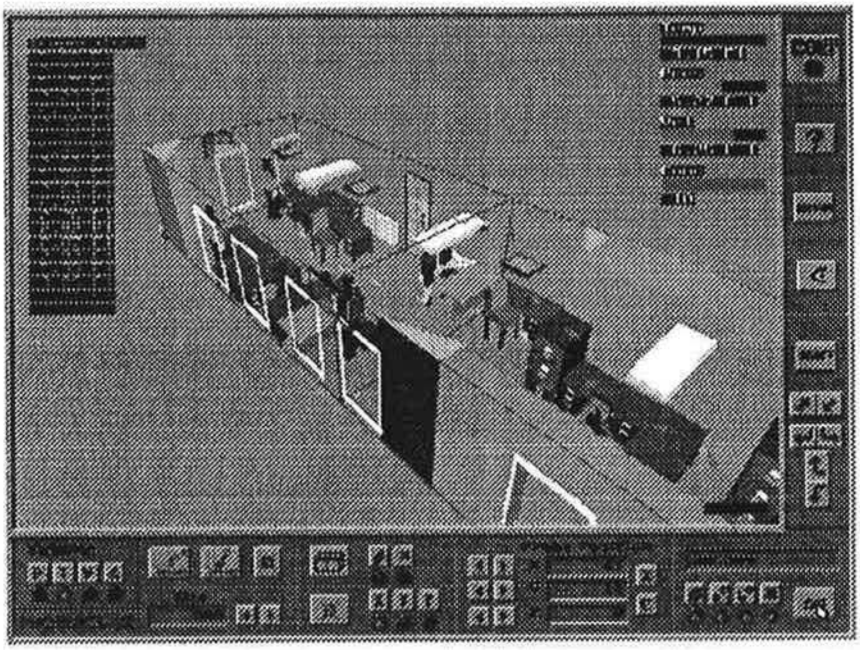


Figure 2. PHOENICS-VR Results Viewer 'Walking through the World'

CHAM's Internet support service facilitates the use of PHOENICS-VR by enabling users to get on-line to this new easy-to-use flow simulation technology at a much lower cost with the back up of application experts. Users can be anywhere in the world but still have immediate access to experts to support and assist them.

The bottom line is that designers needing to know about temperature, air velocity and other 'flow-related' quantities will be able to ask the questions and receive the answers in their own terms. The PHOENICS-VR package makes the question and answer sessions as easy as they can possibly be. The designer builds his world, walks around it and through it until he is satisfied that it represents what he wants, then sends it off for CFD analysis, conducted locally if he has the software, but otherwise at a remote site by CHAM.

When his 'world' comes back, he can enter it again to explore its thermal and fluid flow characteristics, placing 'virtual' thermometers, flow detectors and pressure gauges wherever he needs them. At critical sections 'slices' may be taken and vectors representing velocity or contours of temperature, pressure or concentration may be viewed. Using these facilities 3-D pictures of temperature, pressure, contaminant concentration and air velocity may be built up, as shown in Figure 3.

Additionally, a sensor may be dropped at any point in the 3-D solution domain. The temperature, pressure, concentration and air velocity are automatically displayed at the sensor position. Streamlines may also be displayed originating at the sensor position and, by dropping a number of sensors within the virtual world at points of particular interest, a real-time 3D visualisation of the air movement, along with temperature and contaminant distributions, in and around your building or structure is possible.

Moreover, if the calculations have been computed at a remote site, the user can tap the expertise available there regarding how much trust he should place in his results.

The PHOENICS CFD code, which provides a general purpose capability for simulating fluid flow, heat transfer, mass transfer and chemical reaction processes, is constructed in such a way that it can be readily adapted to many applications. The additional VR pre- and post-processing facilities provided by PHOENICS-VR make it ideal for modelling internal and external flows in buildings which are of interest to those designing energy efficient buildings.

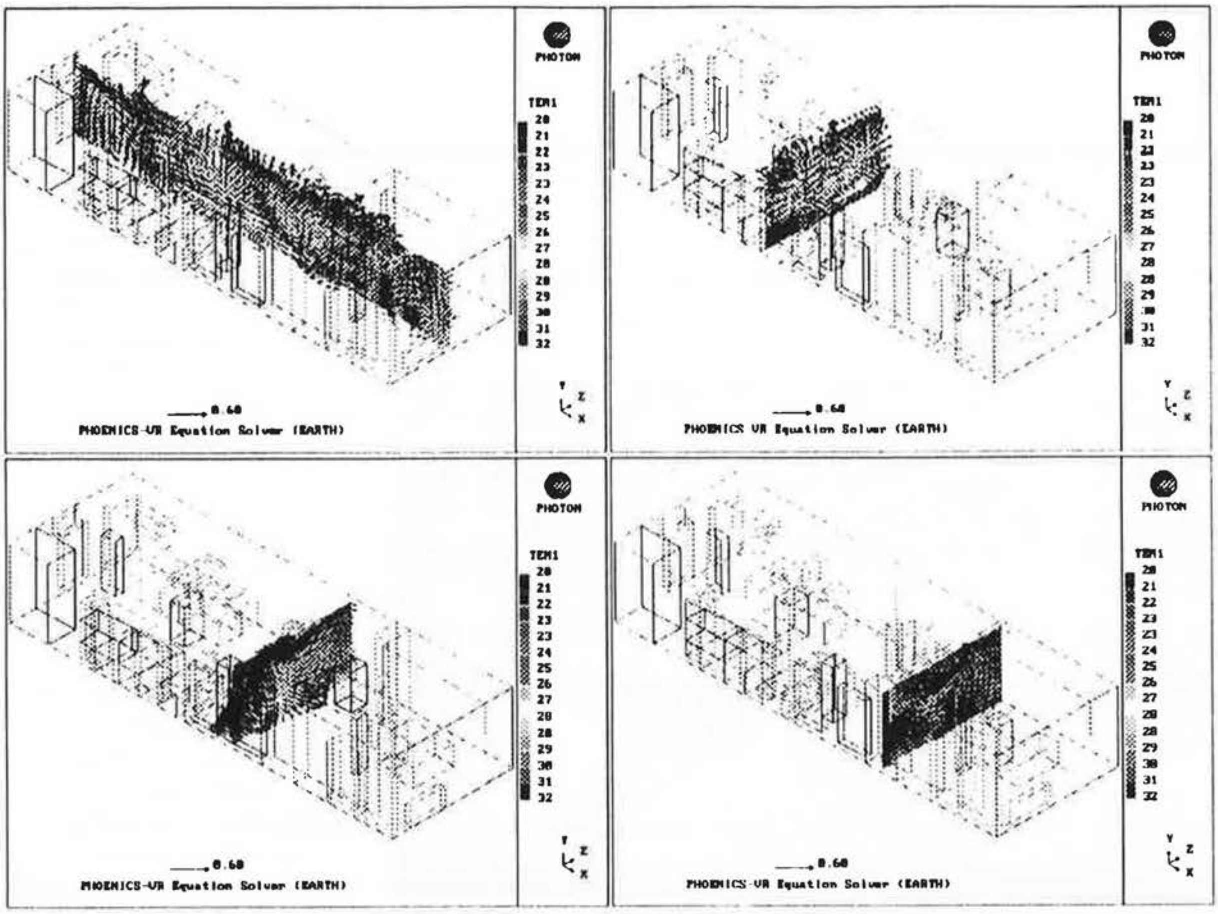


Figure 3. Typical temperature contours and velocity vectors produced using PHOENICS-VR 'Exploring the results world'

The whole concept of PHOENICS-VR and the on-line service makes the complete design process more approachable for the average computer literate engineer or architect and considerably reduces costs. This helps in de-skilling the CFD analysis making it a more practical and speedy tool, viable for a wide variety of applications without compromising the desired functionality. With PHOENICS VR INTERNET engineers can get on line to this new easy-to-use flow simulation technology at a much lower cost than either a software license or a conventional consultancy contract *AND* be safe in the knowledge that CFD experts are on hand to provide assistance where necessary.

PHOENICS-VR air-flow simulation for energy efficient building design

For engineers or architects concerned with designing offices and buildings which are energy efficient in terms of heating, ventilation and air-conditioning, PHOENICS-VR can be used to simulate the effects of a selection of designs on, for example, the natural ventilation characteristics of the building. The use of mechanical air-conditioning and ventilation systems may then be minimised.

Energy efficiency can only be achieved for designs within which a healthy and comfortable working environment is maintained. A number of similar schemes may therefore be investigated with a separate simulation, a number of 'what-if?' type simulations being run for each case. The best of these schemes (from an energy consumption view-point) within which the desired criteria for levels of thermal comfort and indoor air quality are satisfied may then be identified and selected.

Subscription to the PHOENICS-VR Internet service is available for around £1,000 per annum and the costs of running the core package are incurred only when the program is needed for a particular project. It seems likely that this method of utilising CPUs will be on the increase as remote information exchange increases and improves. However, for those who wish to purchase the complete standalone system (including the fluid dynamics solver) licences are available on either an annual or perpetual basis.

For more information please contact Dr Nicholas Cavanagh, PHOENICS-VR Project Manager, at:

Concentration Heat and Momentum Limited
Bakery House
40 High Street
Wimbledon Village
London SW19 5AU, UK
Telephone: +44 (181) 947 7651
Fax: +44 (181) 879 3497
24 Hour answering machine:
+44 (181) 947 6872
email: njc@cham.demon.co.uk

Flovent

A video on optimising cleanroom design

Diane Weeks, Flomerics Limited

A new 12-minute video graphically illustrates how Flovent V1.4 software from Flomerics, the specialists in computational fluid dynamics (CFD), can be used to optimize the design and performance of cleanrooms.

Flomerics joined forces with the Building Services Research and Information Association (BSRIA) to develop both the Flovent software and its new modelling services. The new video demonstrates how this new service was used by SmithKline Beecham (SB) to verify and optimize a proposed design upgrade for existing drying room facilities - the first time such technology has been used in this way.

The problem faced by SB was that the ceilings in its existing drying rooms were too low to permit safe and efficient extraction of potential airborne contaminants. The proposed solution was to install an "active ceiling", some two-thirds of the area of the actual ceiling, above the working equipment in the room. Using Flovent, it was demonstrated that the proposed position of the active ceiling meant that the single operator working in the room would still be exposed to unacceptably high levels of contamination from the chemical being processed. However, by moving the position of the active ceiling so that the operator worked directly beneath it, exposure levels could be reduced to a minimum.

It was also deduced from the flow of airborne contaminant - concentrations of which are shown graded by colour in the Flovent model - that the operator should not stand between the exposed tray of product and the extractor vents, and that the tray should always be kept below chest level for maximum safety.

Thus, Flovent not only helped SB to make crucial design modifications prior to implementation, with consequent important savings in time and money, but also identified vital changes required in operating procedures.

In addition, Flovent presents results in a form easily understood by the engineers involved in commissioning and installing such facilities, and costs a much less than traditional alternatives, with prices starting at £2,765/$4,425.

Flovent will run on almost any computer system and comes complete with documentation, manuals and technical support. It can be supplied on a project or annual licence basis.

Further information can be obtained from:

Diane Weeks, Flomerics Limited, 81 Bridge Road, Hampton Court, Surrey, KT8 9HH, UK. Tel: +44 (0)81 941 8810; Fax: +44 (0)81 941 8730.

Flomerics' thermal management software helps British Telecom cut costs

Hassan Moezzi and Mark Seymour
Flomerics Limited

British Telecom (BT) has halved the cost of cooling its new telephone switching installations, using Flomerics' advanced computational fluid dynamics (CFD) software Flovent and Flotherm.

BT began by examining every aspect of its existing cooling systems, questioning the need for refrigerants, ventilated floors and ceilings. Then, taking the European Telecomms Standard (ETS 300 019 Class 3.1) for cooling switching installations as a benchmark, various theoretical models were analyzed using Flomerics' Flovent and Flotherm software to demonstrate the air flow and the build up of heat within the systems.

Subsequent Flovent analysis of a sample switching installation enabled BT to design a simple cooling system and cabinet suite layout which not only cut predicted lifetime cooling costs by over 50%, but also halved the capital cost of each cooling system, from £30,000 to £15,000 for a typical installation with 25,000 connections.

In addition, the simplified design has meant that less skill and fewer man-hours are required for system maintenance. An added bonus was that simulated designs using Flovent eliminated any risks to customer services which could have resulted from using live test sites.

The new design is expected to be implemented in over 80% of new switching installations - a radical step which, says BT, would not have been possible without the confidence inspired by developing and testing the system using CFD modelling. Future plans include further modelling using Flomerics CFD software aimed at extending cost savings to transmission and other systems.

A video from Flomerics describing the above application and illustrating the air flow within a typical switching installation is now available.

Further information can be obtained from:

Hassan Moezzi and Mark Seymour, Flomerics Limited, 81 Bridge Road, Hampton Court, Surrey, KT8 9HH. Tel: +44 (0)181 941 8810 Fax: +44 (0)181 941 8730.

HGa air movement case studies

Natural ventilation in an urban office refurbishment

Working in a design group with the architects Short Ford Associates, HGa undertook a computational fluid dynamics (CFD) analysis of the final design scheme for the refurbishment of a four-storey city centre office block. The scheme employed full-height external ventilation stacks to drive natural crossflow ventilation on each floor.

Previous studies using bulk air analysis had predicted that the volume of air flow induced naturally through the building could avoid the need for the installation of a new air-conditioning plant. Consideration of air movement was part of an overall energy and environmental assessment looking at the likelihood of summer overheating. CFD was now deemed necessary to show visually how the air moved through the space. This was considered an essential part of convincing tenants and owners of such buildings of the acceptability of avoiding air-conditioning.

The scope and objectives of the study were to:

- develop a greater understanding of the applicability of CFD to the design of fresh air supply schemes driven by natural ventilation
- identify potential internal air flow problems in the naturally ventilated offices
- identify the level of detail required for the specification of the components of a natural ventilation scheme for offices, including window opening positions, dampers, and stack terminations
- provide informative graphic representations of natural air flow in the offices
- provide more reliable data on thermal comfort within the space than was available from a dynamic thermal model
- carry out an independent assessment of the applicability of CFD for future design advice.

FloVENT was successfully employed to investigate the viability of naturally ventilating parts of the final design. The analysis confirmed that the concept of the natural ventilation scheme proposed was sound but that the detailed implementation would not have been satisfactory as it stood. The main problem was the need for the resistance of the opening at the top of the stacks to be reduced so that the air on the top floor would be induced to flow out of the stack rather than out of the window on the opposite facade. It was also shown that the inclusion of cellular offices would not prevent effective natural ventilation.

Wiltshire Music Centre

Working in a design group with the architects Feilden Clegg Design, HGa have assessed the implications for internal conditions of the outline proposals for the Wiltshire Music Centre. The aim of the brief was to provide a low energy building which as far as possible provided fresh air and avoided overheating without recourse to mechanical ventilation or air-conditioning.

The focus for the initial phase of the project was the rehearsal auditorium which seats up to 300 people and accommodates up to a one hundred piece orchestra. It was proposed to naturally ventilate the auditorium provided that such a strategy could be shown to provide sufficient draught free fresh air in the winter and deal with the level of heat gains during the summer. The other space considered in the analysis was one of the tuition rooms. These are predominantly shallow plan and located at the perimeter of the building to benefit from daylight and natural ventilation.

In order to establish if the proposal could achieve the broad aims of the brief HGa undertook a detailed analysis of the energy and environmental performance of the proposed design.

The study comprised three parts:

- investigation of daylight strategies
- detailed analysis of air flow through the auditorium, using CFD, both as a basis for formulating a ventilation strategy and to provide air change rates for thermal simulations
- prediction of the internal air temperatures during performances corresponding to the previously calculated air change rates, using SERI-RES.

This approach enables the combined effect of individual design decisions to be seen and indicated:

- the tuition rooms could not be considered well daylit as the average daylight factor predicted was 2%. This highlighted the need to maximise surface reflectances
- the need to glaze the roof of the lantern to allow uniform lighting over the stage area, thereby preventing the need for task lighting for the orchestra
- how increasing the size of the outlet openings or opening the balcony doors would increase the air change rate in the auditorium
- the need for night venting to cool the structure thereby reducing the occurrence of peak daytime temperatures during performances
- this work could be extended to identify an outline control strategy for the ventilation system.

The results of this initial assessment will influence the design of the building and its services.

Contact Theatre Manchester

HGa worked within a design team with the architects Short and Ford investigating the feasibility of extending and improving the existing premises of the Contact Theatre, Manchester. Included in the brief was a requirement to remedy the overheating problem in the main auditorium. The main heat sources are the lights and the audience. The existing building, completed in 1964, incorporates a mechanical ventilation system which is now considered inadequate. The initial proposal for the refurbishment included air conditioning.

The key question was whether natural ventilation dissipate the internal heat gains, particularly during the summer, and incorporate sufficient control to provide adequate draught free ventilation during the winter.

HGa undertook to determine the air change rates achievable using CFD analysis under winter and summer conditions. These results were then used in the thermal simulation work. Flovent also enables designers to view the distribution of the incoming air within the space. This was important in this instance since the raked seating created occupied zones at various heights.

Air Flow Analysis Method

- a 2D 'slice', default 1m deep, of the auditorium was modelled
- areas of inlets and outlets were proportioned to the openable area per metre length indicated in the outline proposals
- four cases were modelled, high and low level seating for both summer and winter.

Results

- summer time air flow rates were found to be limited by the size of the openings to the supply stacks. These are to be enlarged to provide the higher air flows necessary for summer cooling
- the proposed natural ventilation scheme was shown to be capable of delivering sufficient fresh air for occupant health in winter, but the manner in which the incoming air will be heated requires further investigation
- during winter occupant discomfort was indicated for those sat at low level due to high air speeds, >0.2m/s and a temperature gradient of 7°C across the occupied zone.

The analysis results enabled the architect to convince the client of the scheme's merits, and have since been used by the theatre company in their own promotional acitivities.

Action

- natural ventilation strategy accepted in principle
- size and number of inlet stacks reviewed
- rearrangement of the seating at low level so that people will not sit too close to the perimeter heating and the inlet air grilles.

Leeds City Office Park

The work was an extension to a previous study of Peter Foggo Associates' proposals for an office building which British Gas Properties intend to build at Leeds City Office Park.

The previous study involved the analysis of likely summertime temperatures, daylight levels and the ventilation scheme. The proposals included a 'mixed-mode' ventilation scheme comprising an optimal combination of mechanical and natural ventilation.

The combination of mechanical and natural ventilation has two advantages:

- it is not as energy intensive as the conventional solution, VAV air-conditioning
- it has lower environmental impact. Carbon dioxide (CO^2) emissions are reduced and the system does not require any refrigerant.

Interest in natural ventilation has increased due in part to concerns about occupant satisfaction with internal conditions, with its implications for productivity, and because of increased awareness of the environmental impact of energy use.

However, the strict performance guarantees which can be offered with air conditioning cannot be given with natural or mechanical ventilation. By their very nature, internal conditions within non-air-conditioned buildings vary, corresponding to changes in the ambient.

In this instance the CFD analysis has been used not only as a tool for aiding the designers in the refinement of their initial ventilation concepts, but also as a means of illustrating to potential tenants the merits of a 'mixed-mode' ventilation scheme.

The main focus of the study was to determine the ventilation effectiveness in perimeter and core zones on each floor of the building under various seasonal operating modes.

As the interior space is likely to be a combination of open plan and cellular offices it is important not only to determine the overall air change rate for the building, but also to investigate the relative air movement through the separated spaces.

The results of the analysis showed that:

- under winter conditions, with mechanical extract from the atrium, and assuming a balanced air supply to each floor, the air flow through each zone was reasonably uniform
- under summer conditions with the extract flow driven by the stack effect there is a greater flow-imbalance between zones on the same floor which simple balancing of the floor diffusers will rectify
- opening all the outside windows did not dramatically alter the flow to each floor from the case above and did not create a backflow from the atrium into the office space as might be expected.

Pharmaceutical laboratories, Ahmedabad, India

In conjunction with the architects Short Ford Associates, HGa was involved in the development of the design proposals for a pharmaceutical complex in the Gujarat region of west India for clients Torrent Pharmaceuticals.

Dominant Features of the Local Climate

Most workers will be local with naturally ventilated homes. Consideration of comfort criteria in this particular region of India suggests an air temperature of 32°C with local air movement from electric fans would be acceptable.

Influence of Climate on the Design

A typical cross section of the proposed design showed a three storey building with office and laboratory space each side of a central concourse. By enabling daylight to enter the laboratories and offices indirectly via the concourse, windows in the external walls could be minimised thereby reducing solar gains and the requirement for cooling. With regard to the ventilation strategy the main ideas to be developed included:

- natural ventilation using the predominant south westerly wind
- evaporative cooling in tower to lower air temperature and assist inward air movement
- night purging
- ensure cooled air flows in contact with thermal mass to gain most benefit.

The basis of the ventilation strategy in such a climate could be to either minimise fresh air intake during the day to a minimum consistent with occupancy or to maximise the movement of cooled air through the building to dissipate heat gains. Bearing in mind the very high internal heat gains anticipated, $75W/m^2$, the latter strategy was chosen.

Application of CFD

The scheme modelled has both the supply and extract of air within a central concourse. It was assumed that the supply air temperature would be lowered from the ambient 40°C to 30°C by evaporative cooling. This would assist inward air movement. Windows in the offices and laboratories facing the concourse enabled the circulation of air within these spaces. Low level windows for inlet, high level for extract. It was also assumed that windows to the ambient were closed. From the first and second floors extract was directly into a duct leading up the side of the concourse.

The CFD simulation illustrated the air flow pattern within each zone, and quantified the relative bulk air flow rates through the offices and laboratories on each floor of the building. The analysis indicated that it should be possible to achieve at least 10 air changes per hour through all the spaces which subsequent dynamic thermal simulation work showed would enable comfort conditions to be achieved in May when the peak external temperature is 43°C.

Comparison of air conditioning systems using CFD

David Walshe & David Baker
Roger Preston & Partners

This paper presents a case study in the use of CFD to compare the effectiveness of two air conditioning control strategies in terms of the comfort and air quality achievable. The strategies analysed were mixed flow ventilation and displacement ventilation.

The analysis was performed using ARIA software, a CFD code utilising the k-e turbulence model with buoyancy calculation. The grid for each model was Cartesian; 250,000 nodes in size. Each model took approximately two days to converge using a Silicon Graphics Indy workstation.

A 36m x 36m x 12m high bay was modelled. For the mixed flow model, air was supplied at 16°C and 13 m/s from the sides of the central riser, at a height of 7m. Casual gains for the mixed flow model were modelled as patches on the floor and ceiling. For the displacement ventilation model, air was supplied at 20°C and 0.5 m/s from the sides of the riser, with supply grilles between 0 and 2m high. Casual gains for this model were modelled explicitly,as the location was found to affect the displacement ventilation performance.

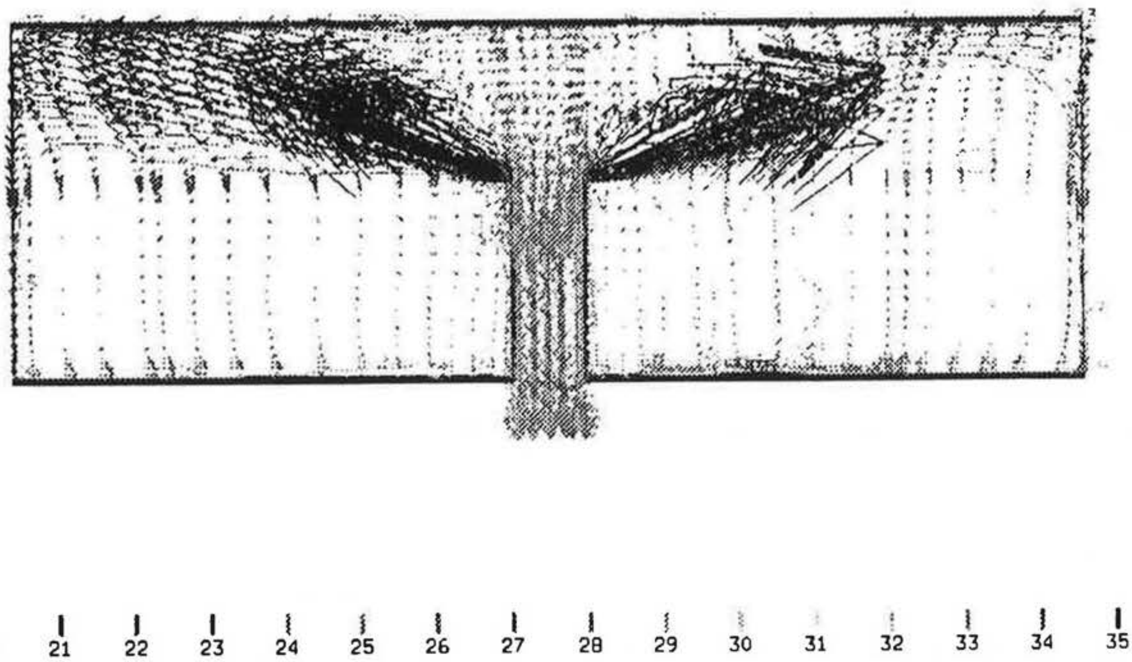


Figure 1. Mixed flow ventilation

People were modelled in groups standing, lights and equipment were modelled on the sides of the riser. In both models, air was extracted through the central riser at a height of 7m.

Fig. 1 displays a cross-section of the mixed flow scenario. The air is mixed throughout the space to the desired condition of 24°C. The room air is induced into the supply jet stream, and re-circulation occurs. It was therefore apparent that airborne contaminants would be re-circulated through the space.

Fig. 2 displays a cross-section of the displacement ventilation simulation. The

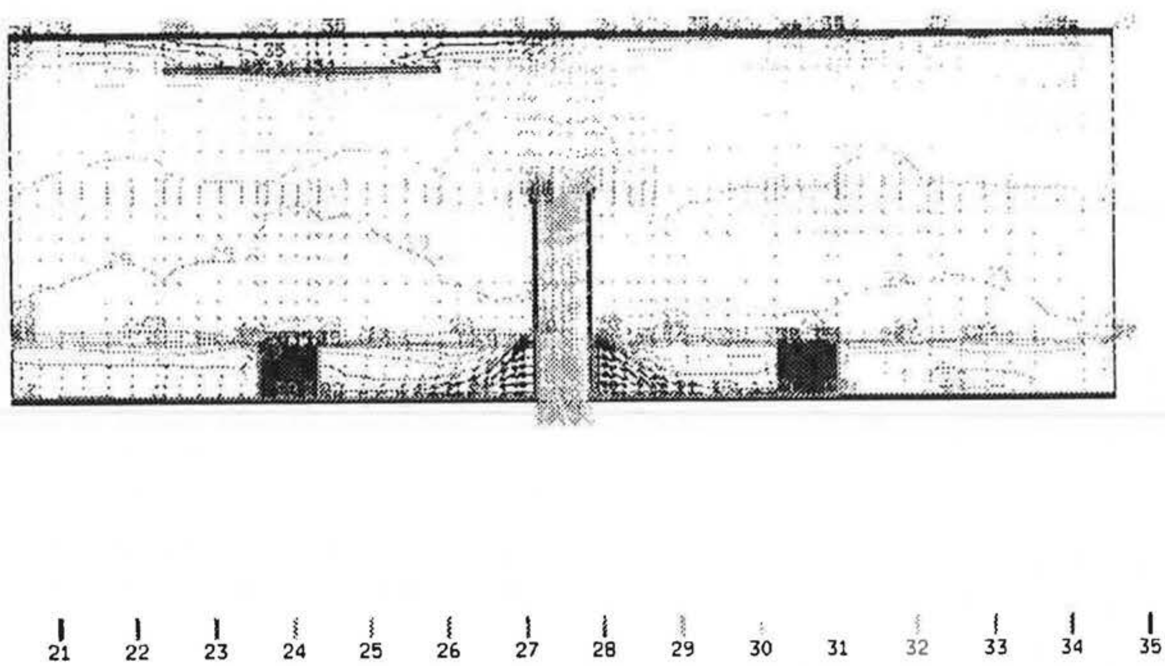


Figure 2. Displacement ventilation

cooler supply spills across the floor, and rises through the heat plumes created by the casual gains. There is considerable temperature stratification between floor and ceiling level, causing air to rise by buoyancy. While air temperatures in excess of 30°C are apparent at high level, the air within the occupied zone is between 22 and 25°C. Furthermore, the displacing nature of the airflow ensures that most airborne contaminants are removed from the occupied space.

It was therefore concluded that displacement ventilation offered better comfort conditions than the mixed flow system, while consuming less energy. CFD proved a suitable analysis tool in comparing the systems, offering a detailed visual view of the anticipated environment conditions.

Climatic data for simulation and design

Eric Keeble, consultant; Geoff Levermore, UMIST; Andy Wright, EA Technology

Accurate, current and recent weather data are crucial to the design of comfortable, energy-efficient and environmentally friendly buildings. This has never been more important than at present as building services engineers and other building design professionals face the challenge of designing buildings with little air-conditioning and using natural ventilation and natural lighting to their full. In addition, the evidence for global warming is now sufficient for the issue to be taken seriously by both clients and the design professions. It is therefore timely to look more closely at the impact of weather on building environmental performance - a challenge being met in work to produce a new CIBSE Guide to Weather and Solar Data.

The existing CIBSE Weather Data Guide, Part A2[1], issued in 1982 focuses almost entirely on manual methods. The selection and use of hourly weather data for computer simulation, in the form of the original CIBSE Example Weather Years, are dealt with only in outline. Now the need is to provide harmonised and compatible weather data for both manual and computer-based methods, recognising that the latter offer the power to explore the sensitivity of designs in terms of energy, environmental peaks, load profiles, etc. There remains a place, however, for data meeting simpler needs, such as characterising the climate of different places or allowing exploration of alternatives during the earlier stages of design.

What do we need from Weather Data?

Defining weather data for building services calculations has grown from simple beginnings to the much more sophisticated possibilities provided by large hourly datasets and simulation packages. Manual data increased in range and complexity over the years, with computers enhancing our capacity to analyse and reduce raw meteorological data from the late 1960s. Criteria for preparing the new Guide include a range of current considerations, for example:

- the increasing use of simulation by designers to examine the response of buildings at various points on the weather load spectrum, eg typical or average, extreme, semi-extreme, etc, including combinations of weather elements that may be critical for the building under consideration;
- providing for expressing performance in risk-related terms, such as the percentage of time that internal temperature exceeds a given value in summer: current practice is sometimes a compromise, such as repeating a worst day without exact knowledge of the risk represented;
- providing better bases for analysing the interaction of weather with fabric and controls in the widened ranged of built forms and design approaches being adopted today, eg use of thermal mass, intelligent controls, naturally ventilated and mixed-mode operation, intermittency, etc, taking account of time of year and the range of weather combinations found in different years;
- providing computer-based tools both to derive conventional weather statistics in a flexible way (eg by varying base temperatures or exceedences), and to explore weather effects and impacts in the

(1) CIBSE Guide, Part A2; Weather and Solar Data; CIBSE, Balham, 1982

context of specific buildings or circumstances;

- improving the weather data available for cooling load assessment by defining extreme hourly sequences, such as 1-, 3-, 5-, and 7-day events, probabilities linked to the upper quartile of summer months or the use of repeated 1-day extremes;
- giving guidance on the likely extent and influence of climate change;
- providing more data on wind speed and direction, and reflecting this in the selection of new hourly weather data;
- improving the quality and range of solar data, especially through revising the prediction methods for vertical and slope irradiation;
- more generally, extending the range of choice available to designers rather than specifying one set method, especially where this helps meet responsibilities for demonstrating the overall adequacy of designs in terms of client expectations;

Selecting weather years and other hourly data sequences has been the subject of research both in the UK and abroad in recent years. The methods, most based on frequency distribution statistics, are being explored with the aim of achieving known risk levels across a wide range of building types[2]. The existing 3 sets of UK weather years (CIBSE[3], European Community[4] and ETSU/Solar[5]) will be augmented by newly selected years for representative sites across the UK. In addition, several approaches to selecting short, extreme sequences of hourly data for summer and winter design are being compared.

Meeting these needs is a major undertaking, which is being aided by financial support from both DOE and CIBSE. The DOE contribution is provided under the *Partners in Technology* programme, in recognition of the value of better weather data for the industry's competitiveness, and as a means of aiding judgements on the necessity for air-conditioning. The Met. Office has also made a major contribution of raw hourly weather data for the exploratory and development work in the project. This is essential to allow the quality of different selection methods and biases for hourly data to be fully explored. The options for availability of such data to end-users have still to be negotiated.

The Weather Data Toolkit

A vast amount of information is locked up in datasets of hourly weather. Only a fragment of this can be made available in hard-copy form, whereas an infinite number of permutations of time periods, sites, variables, ranges, correlations, etc are possible. Users can unlock this storehouse of information only by using the computer. While the computing power available to everyone has increased enormously in the 16 years since Part A2 was published, analysis of weather data using general packages such as spreadsheets and databases involves considerable work, particularly where large amounts of data and complex algorithms are involved. Data analysis programs such as statistical packages are expensive, take time to learn and are not normally available to designers.

It was therefore decided that the new Guide should include a software Toolkit specifically for weather data analysis, which will:

- run on standard Personal Computers and be easy to use;
- allow users to define filters for time and variable ranges;

(2) Keeble E J, Levermore G L: Example Weather Years and the CIBSE Guide; Proc. CIBSE National Conference, Eastbourne , Oct 1995 (in course of publication)

(3) Hitchin E R, Holmes M J, Hutt B C, Irving S, Nevrala D; The CIBS Example Weather Year; Building Services Engineering Research and Technology 4 (3), 119 -124; 1983

(4) Lund H, Eidorff S: Selection Methods For Production Of Test Reference Years; Final Report (short version) EUR 7306; Thermal Insulation Laboratory, Technical University of Denmark, DK-2800 Lyngby, Denmark, for Commission of the European Communities; 1980

(5) Loxsom F: Reduced Solar/ Meteorological Data Sets For Kew: Part 1: Selection And Testing Of A Test Reference Year For Kew: Research in Building Group, Polytechnic of Central London, for ETSU, Harwell; June 1986

- include ready-made, product-specific algorithms (eg, for calculating degree-days);
- handle large datasets (eg, 10 years' hourly data);
- generate derived meteorological variables;
- include built-in Standard Query Language (SQL) for complex queries;
- export in ASCII or common database formats.

In the Toolkit, files of ASCII weather data (or existing database files) are read into a database. Typically hourly data would be read in, but any constant interval which is a whole number of minutes can be used. The database can then be queried using simple dialogs to define filters, time masks, etc, with output directed to files or text on the screen. An example of a query would be "all the hours Monday to Friday, 09:00 - 17:00, May to September, when the temperature exceeds 20°C and the wind speed is below 2 m/s".

From the names of the variables read in, the Toolkit can work out what derived data are available (eg, humidity from atmospheric pressure and wet- and dry-bulb temperatures), and optionally add them to the dataset. Development is now under way using Borland's Delphi, by E A Technology under contract to UMIST and CIBSE. It is hoped to release a test version to selected users in late February 1996. If interested, please contact Dr. Andrew Wright, Tel: 0151 347 2364, Fax: 0151 347 2570, Email ajw @eatl.co.uk.

Consultation with the Industry

The CIBSE Team will soon have fleshed out much of the content of the new Guide, including selecting prototype weather data for a number of sites in the UK and evaluating the Toolkit in prototype form. The next step will be to hold a CIBSE "*Bastings*" - the traditional form of consultative meeting at which members and other interested parties, especially BEPAC members, can learn in detail about the new Guide at draft stage, and contribute comments and ideas to help finalise it to provide maximum benefit for the industry. The *Bastings* on the draft Guide to Weather and Solar Data is scheduled for:

Wednesday, May 15, 1996

at CIBSE, Delta House, Balham

The programme will be based on presentations by the Volume J Chair, Geoff Levermore, and the Chairs of the 3 Task Groups writing the new Guide: Weather Data for Design and Simulation, Wind and Precipitation Data and Solar Data. There will be ample time for discussion and contributions. A nominal charge will be made for attendance.

Interested readers can obtain further information and register to attend by contacting Anne Gibbins at CIBSE, 222 Balham High Road, London, SW12 9BS (Fax: 0181 675 5449). You may also contact Geoff Levermore (Tel: 0161 200 4257, Fax: 0161 200 4252, Email geoff.levermore@umist.ac.uk) or Eric Keeble (Tel/Fax: 01923 441599), Email eric.keeble@umist.ac.uk who will be glad to discuss the work.

NEW CIBSE GUIDE TO WEATHER AND SOLAR DATA

aims to provide:

- methods of selecting typical, intermediate and extreme hourly weather data for computer simulation;
- computer-readable weather data and a flexible data analysis toolkit;
- updated versions of many of the manual data in the 1982 guide;
- clear information on climatic risks, to support the analysis of options for client decision-making;
- data for the effective design of naturally ventilated buildings;
- data to support strategic systems choice, with special regard to energy efficiency;
- data for an up-to-date climatological period;
- guidance on climate change during the design life of buildings and systems;
- better coverage of the UK, plus revised world weather data;
- conformity with common data definitions agreed with ASHRAE, CEN and other bodies, wherever possible.

Editor's note:

This Supplement on Air Movement was held over from the Autumn 1995 issue of the BEPAC Newsletter awaiting the arrival of late contibutions. Subject to consideration of the additional mailing costs which will arise, we hope from now on to issue Newsletters and Supplements alternately at quarterly intervals.

This newsletter supplement is edited and produced by:

BEPAC Administration
16 Nursery Gardens
Purley on Thames
Reading RG8 8AS

phone/fax: 01734 842861
e-mail: 100572.3163@compuserve.com