AIVC
#13,611

Proc. CIBSE A: Building Serv. Eng. Res. Technol. 21(4) 241–247 (2000) Printed in Great Britain B592

Summary Computational fluid dynamics has a wide range of application in the study of room air distribution. The application is providing valuable guidance for those interested in such areas as comfort, productivity and sick building syndrome. This paper gives a comparative review of some of the work undertaken in the field and highlights some of the modelling assumptions noted within the literature. It is apparent from the review that the use of the CFD methodology is helping to develop our understanding of internal ventilation flows, yet it is also apparent that many investigations are currently employing modelling assumptions that will hinder the development of generic guidance from such investigations.

Use of computational fluid dynamics to aid studies of room air distribution: A review of some recent work

M B Russell BEng (Hons) and **P N Surendran** MSc PhD MCIBSE CEng
Department of Aerospace, Civil and Mechanical Engineering, University of Hertfordshire, Hatfield Campus, College Lane, Hatfield, Hertfordshire AL10 9AB, UK

Received 6 October 1999, in final form 5 July 2000

List of symbols

N_c	Air quality number (ε_c/PD)
N_t	Thermal number (ε_t/PPD)
PD	Percentage of people dissatisfied with the quality of the indoor air
PMV	Predicted mean vote
PPD	Percentage of people dissatisfied with the thermal environment
t_{ai}	Internal air temperature (°C)
t_{mrt}	Mean radiant temperature (°C)
t_c	Dry resultant temperature (°C)
ε_c	Effectiveness of containment removal
ε_t	Effectiveness of heat distribution or removal

1 Background

The importance of internal air flow and its interaction with building occupants and their surroundings should not be underestimated. Internal air flows affect human thermal comfort, rates of heat transfer and the transportation of odours/pollutants throughout the ventilated enclosure. The combination of these factors affects the potential occurrence of 'sick building syndrome', the energy consumption and, ultimately, whether the ventilated enclosure is 'fit' or 'unfit' for its intended purpose.

Historically, our knowledge of internal ventilation flows, as with much of the parent subject of fluid mechanics, was obtained experimentally. These investigations include studies of bulk air movement together with studies of fundamental phenomena such as the characteristics of ventilating jets.

Although many benefits are associated with the experimental approach, such as safety and the ability to undertake 'what if' analyses, it should also be recognised that many issues are also raised from such studies. Typically, these include the cost of setting up the experimental facility and the need to ensure that appropriate boundary conditions can be established and applied to the experimental prototype. In room air investigations this normally includes the turbulence properties of amplitude and frequency of the inlet air together with the correct simulation of those factors that affect heat transfer to and from the enclosure. In addition to prescribing surface temperature/surface heat flux, it is also necessary to incorporate the surface properties of emissivity and absorptivity (for studies where radiation is important) and surface roughness (for studies where convection is important).

For transient investigations, the experimental prototype also needs to re-represent adequately the thermal storage properties of its real-world counterpart or at the very least include strategies that sufficiently describe the delay and attenuation of the likely external energy flows.

In addition to these 'configuration' issues, problems also arise with the data acquisition. Data-acquisition equipment needs to be able to capture the range in time and length scales of the turbulent eddy cascade, while also being unobtrusive to the flow field. The experimental practitioner will also have to be aware of the likelihood of 'additional' flow features such as wakes and flow separation created as a consequence of inserting 'probe type' data-acquisition equipment directly into the flow field, such as hot wires or thermistor-based anemometers.

The use of computational fluid dynamics (CFD) within the 'built environment' community is growing. The range of problems being studied with the aid of the CFD method is varied and is limited only by imagination. Examples of this variety include investigations of the computational parameters, mesh resolution, domain size, convergence criteria and computing time, in the study of wind effects around buildings. This paper[1], in addition to presenting results of CPU time, normalised error, etc. also presents predictions of recirculation lengths and surface pressure coefficients around a rectangular building geometry. Further work on external flow has been undertaken by Shao *et al.*[2]. In that paper CFD was used to investigated the magnitude of the pressure coefficients on a tall rectangular shaped building. Implicit in the investigation is the notion that CFD is not the only aid for computational air flow investigations. Indeed, the purpose of the investigation was to develop data sufficient for use by the alternative zonal air flow modelling approach.

The CFD predictions were compared with the results obtained from scaled investigations undertaken in a wind tunnel. The CFD model gave good prediction agreement on the windward face but less so on the lee face. This, the authors suggest, is due to the more complex flow field characteristics present on the lee face and the use of the k–ε model. Notes on the usefulness, the ease of setting up the problem and the ability to define an approaching velocity profile are made within the paper. All of which, the authors note, would be more difficult within a wind tunnel.

Other applications of CFD in the built environment include investigations of air flow in large enclosures. These include the studies of Chow[3] and Schild *et al.*[4]. Chow's study demonstrated the usefulness of CFD in aiding investigations of the ventilation regime in railway waiting halls and underground car parking spaces. Schild's study looked at the problems of modelling atria, with their strong coupling to the external ambient, and present results from a series of alternative modelling approaches. A further study of the application of CFD to the modelling of atria is presented in reference 5.

Studies of the components of typical ventilation systems have been undertaken by Gan and Riffat[6]. Here the CFD methodology was applied to study the pressure losses across close-coupled ductwork fittings, for which the predictions were compared with results obtained from experimental studies. In this study the investigation also reviewed the influence of applying different mesh structures, turbulence models and discretisation schemes. It is worth noting that some of the modelling combinations gave rise to the prediction of a negative zeta factor—indicating a pressure gain rather than pressure loss. The results of this study reinforce the usefulness of CFD but clearly demonstrate that it must be used with care.

Finally, the continuing drive for low-energy buildings has seen the novel application of CFD to predict building component behaviour. To aid the understanding of the 'Termodeck' flooring system, an emerging feature of low-energy buildings, a series of CFD-based investigations has been undertaken by Winwood *et al.* The initial study[7], for which the findings were presented in 1994, introduced modelling simplifications that were imposed for computation time savings. These included the use of a constant eddy viscosity, whose value was selected from experimental data, and rectangular air paths in lieu of the actual circular ones. Furthermore, the actual air path geometry did not match that of the real component, with the dead ends in the real air paths not being modelled. Although a qualitative realism was observed in the predictions, the overriding assumptions and simplifications were likely to impact on the quantitative validity of the results.

A latter paper[8] by the same investigators in 1997 made improvements to the initial study in that a two-equation (k–ε) model of turbulence was used, together with modifications to the air paths to better reflect the real world slab. Good agreement between the computational study and experimental data was presented. The confidence gained in the comparisons enabled the investigation to explore the effect of varying the initial design parameters. The latter study showed the effect of air flow rate, thermal conductivity, slab lengths and number of active cores (ventilation paths) on the performance of the components.

A trawl of the HVAC-oriented literature will show, however, that much of the CFD application within the built environment/HVAC field appears to be oriented to room air distribution studies. This is not surprising, since room air distribution systems are often the major interface between the room's occupant or process and the attendant environmental control system. Getting this right is the essential task of those concerned with comfort, productivity, sick building syndrome and energy conservation.

Previous reviews of the CFD methodology and its application to building-related studies come in many guises and include the work of Jones and Whittle[9] and of Li[10]. In Jones and Whittle's work, in addition to an historical overview of CFD investigations (1970–1991), the emphasis is around the CFD methodology itself and the modelling issues associated with building air flows. The paper discusses mesh generation, turbulence models and numerical methods/solvers. Li's review covers both the CFD methodology and an overview of recent work.

The intention of the present paper, however, is not simply to indicate those broad areas being investigated but to provide a critical review of some of the recent work, highlight the different modelling assumptions applied and discuss how these assumptions may influence the resultant predictions. This paper has at its heart not a review of the CFD method *per se* but rather the results of the CFD application to room air distribution studies.

With reference to the HVAC-oriented literature, it seems apparent that two distinct types of study are reported.

— In one type CFD is applied to evaluate the performance of room air distribution systems and their influence on the ambient flow field, for example reviewing the effect of varying supply volume flow rates or reviewing the influence of the location of the supply/exhaust diffusers on the flow field. These studies are termed here 'CFD application-oriented studies' and examples are discussed in section 2.1.

— In the other type room air distribution problems act as a vehicle for reviewing and evaluating the performance of alternative CFD modelling approaches, for example comparisons between various turbulence models or discretisation schemes. These studies are termed here 'CFD modelling-oriented studies' and examples are discussed in section 2.2.

The CFD methodology, the need for its application and its general application are discussed in numerous generic CFD texts and papers and these discussions are not repeated here; see, for example, references 11–13.

2 The review

2.1 CFD application-oriented studies

Gan, in 1995, used CFD to study the ventilation efficiency and thermal efficiency (ventilation effectiveness) of 13 different room configurations[14]. The differences in the cases studied were in the location of the supply and exhaust openings, opening size and hence imposed inflow/outflow velocities and thermal conditions—heating (winter) or cooling (summer) mode.

The space used in the study was 4.9 m × 3.7 m × 2.75 m ($L \times W \times H$); one of the shorter walls contained a double-glazed window 3.7 m × 1.5 m ($W \times H$), and the other surfaces were considered adiabatic. A parallelepiped obstacle was used to simulate an occupant and was prescribed a constant heat flux of 70 W m^{-2} and a CO_2 production rate of 4.72×10^{-3} l s^{-1}. The heat flux was considered to act over the whole 'body' area, while the contaminant was generated at 'head' level.

In addition to the parametric study, a major thrust of the paper was the use of a radiation model embodied within a CFD code, essential for any mixed mode heat transfer problems, including human thermal comfort. The results of the study demonstrate that differing room air configurations influence the ventilation effectiveness and also that different preferred options emerge for the heating and cooling modes. No comparisons are given with regard to the CFD predictions and any experimental results. The paper points to the fact that the air flow model had previously been validated for both naturally and mechanically ventilated rooms and as such appears to offer this as the basis for model validity. References are given within the text regarding the validation study, yet the paper does not provide an overview of the code's successes and failures.

Calculation of ventilation effectiveness for heat distribution/removal (ε_t) and contaminant removal (ε_c) has also been undertaken by Awbi[15]. In addition to these ventilation parameters, Awbi further establishes the ratios of ε_t and ε_c to the predicted percentage of people dissatisifed with the thermal environment (PPD) and to the predicted percentage of people dissatisfied with the quality of the indoor air (PD) to establish further performance indices. These and other performance figures are calculated for a ventilated enclosure subjected to both displacement and mixing ventilation regimes. The ratio ε_t to PPD is the thermal number (N_t) and the ratio of ε_c to PD is the air quality number (N_c).

Plots of velocity, predicted mean vote (PMV), N_t, N_c, ε_t, ε_c and age of air are given against the inlet jet momentum. The broad result of higher values of ε_t for a displacement system under cooling mode is in agreement with the comparable cases of Gan's study (cases 10–13). Interestingly, the value of effectiveness of contaminant removal (ε_c) given by Awbi does not concur with the broad picture presented in Gan's paper. In Gan's work higher values of this parameter (Gan used the same definition to calculate contaminant removal effectiveness but uses different nomenclature (n_c) from that of Awbi) are found with displacement ventilation over a mixing system. Awbi's results gives comparable values of ε_c for both mixing and displacement ventilation regimes. The differences between their results are significant and could impact on the ability to discern any generic ventilation guidance/traits.

Although the cases compared were both essentially displacement regimes, in Gan's study the supply and extract openings were located on opposing walls, whereas in Awbi's investigations they are co-located on the same surface. Such proximity of openings could lead to short-circuiting, which in turn would lead to reduced levels of ε_c.

The likelihood of short-circuiting is physically plausible and can be seen from the following reasoning. With Awbi's arrangement, the supply jet, located directly above the exhaust opening, will spread as its enters the enclosure. Further, the convective accelerations of the turbulent eddies associated with the incoming supply jet will, by definition, be moving fluid randomly throughout the domain. For completeness, these convective accelerations are of the form

$$u_j \frac{\partial(\rho u_i)}{\partial x_j}$$

in the Navier–Stokes equations. The above equation is presented in the compact tensor form, where u_i represents the velocity in the i direction (m s^{-1}), x_j represents the space domain in the j direction (m), and ρ is the fluid density (kg m^{-3}). Both phenomena—the spreading of the jet and the convective accelerations—will ensure that some of the incoming fresh/young air meets directly with the extract device, and hence its ability to mix with the ambient fluid and dilute any contaminants is hindered.

A displacement ventilation regime generally attempts to establish a piston/plug type flow regime within the enclosure. Here the unidirectional flow ensures that poorer quality air exists at distances progressively further from the incoming air. Such arrangements are clearly best served if the turbulence characteristics of the ambient flow are reduced or damped, for even in a displacement flow regime the convective accelerations, associated with turbulence, will rapidly mix air that was once remote. Such undirectional flow is best established if the supply and extract location assist each other, as in Gan's work.

Another study of ventilation efficiency and the ability of the ventilation system to remove contaminants is reported by Chung and Dunn Rankin[16]. This study looked at the influence of two alternative exhaust locations (high-level and low-level), and the effect of a half-height partition. An interesting observation emerged from that study in that the authors note that 'the partition has a greater influence on changing indoor air quality than does the ventilation configuration arrangement'. Qualitative validation of the results is made via flow visualisation using a model room, lasers and smoke seeding, while quantitative validation is undertaken by comparing the predictions of contaminant decay rates with those measured in the same model room.

Three locations were chosen for the numerical comparisons: in front of, on top of and behind the partition. The paper notes the 'excellent agreement' between predictions and experimental data for the first two locations but notes only a 'similar decay trend but different decay rate' for the location behind the partition. It is likely that such a location would be exposed to a more complex flow pattern—separation, circulation and re-attachment, and so on—and the authors suggest that one possible reason for the discrepancy is that 'the k–ε model is not quite adequate for our case'. The flow field predictions indicate that the difference in flow pattern for laminar and turbulent calculations is more pronounced behind the partition than it is in front of or on top of the partition. Such observations clearly demonstrate not only that turbulence models are case-dependent but also that their application has differing sensitivity through the calculation domain. In this study, isothermal conditions were imposed and all thermal effects were neglected.

It is interesting to compare these observations with those made in the earlier reference to Shao *et al.* In their external flow study, similar observations regarding the results and the appropriateness of the turbulence model were also made. Recall that in this study the CFD model gave good prediction agreement on the windward face but less so on the lee face, which it was again suggested is due to the more complex flow field characteristics present on the lee face and the use of the k–ε model.

Specific CFD studies relating to comfort include the work of Gan[17] and Murakami *et al.*[18]. In Murakami's work a computational thermal manikin is used that is humanoid in geometry. Five cases were presented, which include variations in the type of flow geometry (stagnant, upward and downward), differing levels of turbulent intensity and different boundary conditions prescribed to the manikin (fixed heat transfer (20 W m^{-2}) and uniform surface temperature (33.7°C). In addition to providing field plots of velocity and turbulent kinetic energy, the paper also presents details of the local convective heat transfer and convective heat transfer coefficient on the manikin.

Although a radiation algorithm is not employed in this study (as noted earlier, an important feature of any thermal comfort investigation), the investigation highlights areas where different rates of convective heat transfer occur on the human body. Further, the differing rates of heat transfer are clearly seen to be a function of the prevailing flow configuration: i.e., higher values of convective heat transfer were seen to occur when a downward flow regime was imposed.

The paper also describes the predicted flow features close to the manikin and suggests that localised separation occurs as the geometry of the body changes: i.e., from the chest to the head there is likely to be a complex flow field that will also inform the rate of heat transfer. It is difficult to imagine how a parallelipiped representation of a body (as used in Gan's work) would allow such localised detail to be captured, but the overhead in developing a complex manikin shape is the necessity to deviate from a simple Cartesian mesh to one of an unstructured mesh arrangement.

Unfortunately, without the inclusion of a radiation exchange algorithm, it is difficult to see how such studies will adequately inform comfort investigations.

A study by Lu *et al.* in 1997 specifically looked at the influence of a convective heat source on both the flow field and particle distribution in a ventilated enclosure[19]. The heat source was a radiator located below a double-glazed window. The room dimensions were 4.74 m $\times$ 3.45 m $\times$ 2.7 m ($L \times W \times H$), the radiator was 1.1 m $\times$ 0.6 m ($W \times H$) and the window was 2.2 m $\times$ 1.6 m ($W \times H$). The contaminant source, for which a range of particle sizes (1–20 μm) were modelled, was generated in the centre of the enclosure.

Here the CFD results show a wall jet developing above the radiator, due to the positively buoyant plume, and the jet's influence on the ambient flow field. The ambient field is driven by the localised entrainment of the jet. Qualitative and quantitative comparisons are made with experimental observations and the authors note that 'generally the numerical results show good agreement with the relevant measured values'. Some discrepancies are observed and the authors suggest that these could be a function of either the numerical predictions or the experimental data acquisition. On the numerical side, the suggestion is that the mesh could be refined or an improve turbulence model could be used in lieu of the standard k–ε model. On the experimental side, the suggestion is that limitations of the instruments may be to blame. There is no mention in the paper of any attempt to substantiate the numerical claims by reviewing the effect of using differing mesh structures or the inclusion of a different turbulence model.

As with many other CFD results, the paper makes use of internal air temperature (t_{ai}), which will again be extremely misleading to those interested in comfort studies, given the obvious implication of the heat sink and heat source on the space's mean radiant temperature (t_{mrt}) and hence dry resultant temperature (t_c). The dry resultant temperature is currently offered by the Chartered Institution of Building Services Engineers (CIBSE) as the principal design comfort index[20] and is a function of t_{ai}, t_{mrt} and air velocity. Contaminant (particle) transport follows that of the general air pattern in that the particles are drawn towards the jet through the entrainment. The larger particles are seen to deposit on the surfaces while the smaller particles remain in suspension. This, the authors suggest, means 'that occupants sitting close to the radiator run a higher risk of passive smoking than those sitting the other side of the pollutant source'.

More specific studies of particle distribution and deposition have been undertaken for a single-zoned enclosure[21] and for a two-zone enclosure separated by a low-level opening[22]. The geometry and conditions of the enclosures were similar for both studies. The enclosure dimensions were 5.0 m $\times$ 3.0 m $\times$ 2.4 m ($L \times W \times H$). Ventilation was provided by high-level supply and low-level extract devices located on opposing walls.

The single-zone study looked at the deposition and distribution of spherical particles ranging in diameter from 1 to 10 μm under ventilation rates of one and two air changes per hour; the mass density of the particles was set at 850 kg m^{-3}. The two-zone study looked at the deposition, distribution and migration of spherical particles under ventilation rates of 11.566 and 12.708 air changes per hour. In this study the particles ranged in size from 1 to 5 μm and their mass density was set at 865 kg m^{-3}.

Notes on particles deposition and suspension are similar in both studies and concur with the earlier observations of Lu *et al.*: the larger particles are seen to deposit more quickly than the smaller ones, and some of the smaller particles remained suspended for periods sufficient for inhalation by the occupants.

Within the two-zone, comparisons are made with available experimental data. The authors claim that 'the numerical results show reasonably good agreement with the measured data especially in the first 10 minutes'. A review of the results suggests that deviations in the predictions and the experimental data become more apparent as the time period increases.

For both the single-zone and the two-zone studies the domain was considered isothermal and the particles were not considered to rebound from any surfaces.

Investigation of air supply parameters on indoor air diffusion is reported by Chen *et al.*[23]. In this study, the ventilation regime was essentially a displacement configuration, i.e. low level supply and high level extract for which a series of differing configurations were investigated, including rooms of differing sizes, changing the effective supply inlet area, altering the diffuser shape, changing the level turbulence intensity of the supply air and changing the supply air rate and its temperature. The study presents predictions on velocity, smoke concentration and percentage of people dissatisfied owing to drafts. Furniture in the form of tables (2), filing cabinets (2), bookshelves (2), occupants (2), computers (2) and a glazed component were included in the study. The occupants, the computers and the glazing were prescribed a constant heat flux to stimulate heat gains.

The inclusion of furniture in the study is an important addition and one that moves such investigations further towards reality. Tables and chairs, as well as creating flow features that would not otherwise have been there—wakes, stagnation points and shear layers, etc.—are also likely to create areas that may capture and create reservoirs of highly polluted air. Hence, any meaningful investigation of ventilation effectiveness, air age, etc. should include such flow obstructions.

It is unfortunate that no attempt to model the additional fans that would be located within real-world computers was mentioned in the paper. It is recognised however, that the correct implementation of the internal fan into the numerical study is a non-trivial task (the internal flow through a computer is likely to be a study in its own right), but this additional flow feature would undoubtedly create localised and possibly bulk air movement that might influence the overall results and hence any conclusions drawn from them.

Proc. CIBSE A: Building Serv. Eng. Res. Technol. 21(4) B3–B6 (2000) Printed in Great Britain

Blue Pages

Correspondence

Sir,

A new degree-day model for estimating energy demand in buildings

I read this paper[1] with particular interest because of my involvement with the development of the admittance method and the CIBSE Energy Codes. The authors, Dr Day and Dr Karayiannis, have to be congratulated for both their papers on the weaknesses of the current degree-day model and the need for an improved model based on a more accurate estimate of the daily mean internal temperature of a building.

However, while I accept their claim that the main disadvantage of using the admittance model is the need to estimate the pre-heat time, I do not believe that their derived model has a significant advantage over the admittance model (see page 174, middle of column 1 and end of column 2).

From my reading of their paper, the designer has to make two estimates of the behaviour of his system, rather than the one needed for the admittance-based approach:

— the depth of mass that plays an active role in the calculation of c, the thermal capacity of the fabric used in equation 8 (as noted by the authors at the top of page 175);

— the value of the plant capacity Q_p used in equation 10.

If the plant capacity is derived using the current CIBSE *Guide Book A*, it is more than likely that this will be based on the admittance method. Therefore, the use of the derived model requires the designer to have a good understanding of the admittance method.

Also, to understand the role that the depth of mass plays in the value of c for short-term diurnal storage, the designer would be advised to read some of the earlier papers on the admittance method. The paper by Millbank and Harrington-Lynn[2] is of particular value.

However, whichever approach is adopted, I agree with the authors that an accurate derivation of the daily mean internal temperature is essential for an accurate estimation of the energy demands of a building. This requirement was one of the major reasons for the adoption of the calculation method used in the CIBSE Energy Codes 1 and 2.

References

1 Day A R and Karayiannis T G A new degree-day model for estimating energy demand in buildings *Building Serv. Eng. Res. Technol.* 20(4) 173–178 (1999)

2 Millbank and Harrington-Lynn *Building Serv. Eng.* 42 38–54 (1974)

J Harrington-Lynn
16 Widgeon Way
Garston, Watford
Herts WD2 4RG
UK