APPLICATION OF CFD TO NATURALLY VENTILATED BUILDINGS : A GUIDE FOR PRACTITIONERS

R R Cohen*, R M Davies*, M Seymour** and J P Leppard**
* HGA Ltd, Burderop Park, Swindon, SN4 0QD
** Flomerics Ltd, 81 Bridge Road, Hampton Court, Surrey KT8 9HH

Computational fluid dynamics (CFD) modelling is increasingly being used as a tool for predicting ventilation rates and air flow patterns as part of the building design process. The potential benefits of this form of modelling are that designs can be optimised to make the most efficient use of ventilation, and so to increase air quality and decrease energy use. Although CFD has shown itself to be a powerful tool in the nuclear, aeronautical and electronics industries for over two decades, its reputation has been built on extensive work specific to those fields. It is recognised that there is a need to establish guidelines for and validate the application of CFD to building design, and in particular to schemes involving natural ventilation. This paper reports the results of studies undertaken in order to produce a guide for practitioners on how to use CFD to examine natural ventilation in buildings. The work addressed the key unresolved issues which face practitioners when they use CFD as a design tool for naturally ventilated buildings, including how much of a building and its surroundings are modelled, the treatment of openings and surface heat transfer and the level of detail required to represent internal furniture and heat sources.

INTRODUCTION

For practitioners the issue is a simple one; CFD is a powerful tool yet a complex one, how can it be utilised to provide design solutions successfully with the limited hardware resources commonly available to the practitioner, ie a PC, and with the limited time available, both in terms of project duration and staff budgets? This paper describes a research programme undertaken to develop guidelines for engineers and designers of naturally ventilated buildings which should help them to apply CFD appropriately to their buildings.

The research study used a proprietary CFD program FLO*VENT*® to investigate the different modelling alternatives open to the CIBSE member or other building professional so that they can judge the necessary level of detail required to use CFD to answer their design questions. The key to successful modelling is to be able to make simplifying assumptions but still to allow the model to provide performance information to the level of accuracy required.

The research programme considered the effect of different modelling approaches when applied to a generic cellular office and atrium, and also in three real design case studies. The main objective was to derive guidance for the practitioner on the effect on accuracy of simplifications. This paper concentrates on the results obtained for the simple office model and the case studies.

Using the generic office model, the following issues were addressed:

- How much of a building and its surroundings need to be modelled, that is:
  - the validity of modelling in 2 dimensions rather than the more rigorous 3
  - whether to include in the model a volume of outdoor air
  - whether to include more than one storey of the building in the model
- What level of detail to include for various window geometries;
- How to represent surface heat transfer;
- How much detail to include for furniture, people and equipment

In the three case studies, the following alternative approaches were examined:

- A two dimensional and a three dimensional representation of a building
- A representative floor of a three storey building and a model including the whole building
- Models with different representations of ventilation openings

The remainder of this document outlines the models on which the parametric variations were undertaken, and the guidelines identified.

METHODOLOGY

Cellular Office Model Description

The cellular office dimensions were taken as the following typical values:

- 3m wide single outside wall, 4m deep
- 3.75m floor to floor height, 3m floor to ceiling height
- Window in the centre of the outside wall 2m wide by 1.75m high, cill height 1m above finished floor level

The analysis assumed single sided ventilation through the window with no infiltration.

Base Case Model

In the base case model, the various elements of the office were represented as follows:

- the window was represented by a 100% clear hole-in-the-wall
- a 1m deep space of outside air was included in the model
- surfaces were adiabatic (surface heat transfer is zero)
- temperatures of all surfaces were fixed at the same value
- the space was taken as empty ie no furniture, equipment or people are included
- heat gains from people and equipment were included as a uniform volume source

Results Appraisal Method

A series of single parameter variant CFD runs were undertaken on the cellular office model, each one testing the effect of a specific change in how a particular element in the model was represented. Converged solutions were tested to ensure that they were not grid dependent ie that a finer grid in the model would not produce a different solution. The number of grid cells used in the various variants ranged from some 20,000 to 100,000.

A methodology had to be developed for comparing the results of each run, given the difficulties of comparing the individual temperatures and air velocity vectors calculated by the model for each grid cell. On a qualitative level, air velocity vector plots with superimposed temperature contours for 2D slices through the model could be compared by eye. For a quantitative comparison, it was decided to compare the predicted bulk air flows both through the window and across vertical planes set at 1m, 2m and 3m distances from the window, which indicate the penetration of the outside air into the depth of the room.

RESULTS OF GENERIC STUDIES ON A CELLULAR OFFICE

Summary of Results

The results for each run are summarised in Table 1.

Table 1 Summary of results for model of cellular office

Variant	Description of single parameter variant	Bulk air flow through window	Bulk air flow through window	Flow across plane 1m from window	Flow across plane 2m from window	Flow across plane 3m from window
		ach	m^3/sec	m^3/sec	m^3/sec	m^3/sec
0	Base Case	16.1	0.161	0.203	0.182	0.108
1A	2D model, open area scaled	15.4	0.154	0.207	0.131	0.050
1B	2D model, slice across the central axis	24.5	0.245	0.313	0.252	0.113
1C	2D model, slice across the central axis	16.3	0.163	0.208	0.168	0.076
2	Space outside building cuboid not included	17.1	0.171	0.209	0.193	0.118
3A	3 storey model of variant 4D - ground floor	3.6	0.036	0.085	0.071	0.049
3B	3 storey model of variant 4D - middle floor	3.5	0.035	0.087	0.071	0.049
3C	3 storey model of variant 4D - top floor	3.5	0.035	0.088	0.078	0.060
4A	Bottom hung window	2.2	0.022	0.106	0.083	0.056
4B	Bottom hung with flanged sides	1.8	0.018	0.106	0.082	0.050
4C	Top hung window	2.3	0.023	0.063	0.059	0.059
4D	Centre-pivot window	3.7	0.037	0.085	0.070	0.048
4E	Left-opening window (opening into room)	2.3	0.023	0.079	0.068	0.040
4F	Left-opening window (opening outwards)	2.9	0.029	0.071	0.056	0.033
4G	Sash window - detailed	5.9	0.059	0.112	0.092	0.083
4H	Sash window - modelled as plates	5.2	0.052	0.109	0.087	0.072
4I	Modelling window as hole with FAR= 22.9%	7.8	0.078	0.136	0.103	0.070
5A	Fixed surface htc, wall 'U' = 0.45 W/m^2K	17.6	0.176	0.216	0.203	0.129
5B	Fixed surface htc, wall 'U' = 4.3 W/m^2K	22.2	0.222	0.275	0.269	0.211
5C	Computed surface heat transfer coefficient	20.2	0.202	0.246	0.230	0.155
6A	Inclusion of the main planes of furniture	20.8	0.208	0.260	0.264	0.171
6B	Detailed representation of furniture	16.1	0.161	0.191	0.182	0.108
6C	Detailed representation of furniture & equip.	13.9	0.139	0.181	0.118	0.099

Discussion of Results

2D Model (Variant 1)

In this variant, a 2D model was created, nominally 1m wide (and other dimensions as per the 3D model). Three possible methodologies for obtaining bulk air flow predictions from a 2D model were tried:

1A. Scaling the open area in the 2D model by the ratio of the room volumes in the 2D and 3D geometries (so this 2D model had a window 1m wide by 1.167m high)

1B. Taking a 1m slice along the central axis of the room (so this 2D model had a window 1m wide by 1.75m high)

1C. Taking the same 1m slice along the central axis of the room and, subsequent to the CFD calculation, multiplying the bulk air flow rates by the ratio of the open area to room volume in the 3D and 2D geometries (a factor of 2/3 in this case)

The volume flow rate results for the 2D models in Table 1 have all been multiplied by the ratio of the room volumes in the 3D and 2D geometries (ie a factor of 3) to obtain flows comparable with the 3D model.

The results in Table 1 indicate that modelling in two dimensions may be acceptable for predicting bulk ventilation rates, with case 1A underpredicting the 3D model by 4% and case 1C overpredicting by just 1%. The simplistic case 1B, where no adjustments for relative window size were made, overpredicted by over 50%.

However, neither of the better 2D models came close to the 3D model's predictions of air flow rates 3m into the space. Typically, a 2D model would be unsuitable for predicting the flow patterns unless the ventilation openings, geometry and heat gain are almost 2 dimensional in nature.

Not Including the External Environment (Variant 2)

A "worst case" no wind condition is often assumed when analysing naturally ventilated building designs with CFD. In these circumstances, it is questionable whether it is necessary to include outside air in the model. In this variant, the 1m deep space of outside air was excluded from the model. This caused a 6% overprediction in the bulk air flow through the window and increases of 3 to 9% in the predicted flows at 1m to 3m from the window. This result suggests that only for a very accurate model would it be important to model the external environment under no wind conditions.

Modelling More Than One Storey (Variant 3)

In order to guide the practitioner on how to model a multi-storey building (without an atrium), a set of runs was undertaken to test the effect of including more than one storey in the model (to be worthwhile, outside air must be included in such a model). The key factor is how much of the air leaving one floor is entrained into the flow entering the floor above. This will depend strongly on the window type. Sash, top hung and centre-pivot window geometries showed significant re-entrainment, whilst bottom hung did not. This indicates that the latter might be reasonably represented by a single storey model whilst the former might need a multi-storey approach.

Table 1 shows the results for a centre-pivot window assuming a still environment outdoors: in this case the bulk flows in the single storey model were within 5% of those on each floor of the multi-storey model. Within the rooms, only the flows on the top floor 3m into the space differed significantly from the equivalent flows in the single storey model.

Due to re-entrainment, temperatures in the multi-storey model increased on each successive floor. The validity of such a result depends on whether the external air conditions are still. Note that in a 2D model, the hot air leaving one floor would be forced to enter the floor above; a 2D multi-storey model is therefore likely to give unreliable results for the internal temperatures on upper floors.

Modelling Window Geometry in Detail (Variant 4)

The following 8 different window types were modelled explicitly:

- Bottom hung window
- Bottom hung with flanged sides
- Top hung window
- Centre-pivot window
- Left-opening window (opening into room)
- Left-opening window (opening outwards)
- Sash window - detailed model
- Sash window - modelled simply as plates

Each window was set so that the open area was 22.9% of the base case window area (a hole in the wall). An alternative approach, using a uniform generic resistance to flow (ie a mesh) also with a free area ratio of 22.9% was also tested.

The results, shown in Table 1, indicate that the bulk air flow rate is significantly affected by the geometry of the window. The bulk air flow with a generic resistance (case 4I) was 48% of that with a hole in the wall (the base case). Comparing explicit window geometries with a generic resistance, the bulk air flows varied from 23% (with a bottom hung window with flanged sides) to 76% (for a detailed sash window) of the result for the generic resistance.

It is noteworthy how the detail of the window geometry affected the bulk air flows: for the sash window (cases 4G and 4H) incorporating finer detail, which enables flow between the upper and lower sashes, increased the flow by 13%, for the left opening window (cases 4E and 4F) changing the opening direction made a 26% difference and for the bottom hung window (cases 4A and 4B) adding a side flange reduced the bulk flow by about 20%.

Flow penetration, as represented by the flows at 1, 2 and 3m into the space also varied significantly with the geometry of the window.

Treatment of Surface Heat Transfer (Variant 5)

There are three options for the treatment of surface heat transfer:

1. Applying a fixed heat flux (in W/m²) - this was used for the base case model with the heat flux = 0
2. Using a fixed heat transfer coefficient (in W/m²K) related to the free stream temperature of the air passing over the surface, which means a coarse grid is appropriate
3. Allowing the model to compute the surface heat transfer coefficient (htc) using wall functions, which means a fine grid is appropriate

In general, the first method is only suitable for heated surfaces from which the heat output is known (eg a 'hot plate' or an electrically heated window). The second method relies on empirical values for surface heat transfer and cannot reflect any dependence of the surface htc on the temperature difference between the surface and the air, whilst the third relies on the accuracy of the wall functions which may not be perfect. In theory, the third method should be the most reliable but it has a major practical disadvantage in requiring a very fine grid (in the range 2 to 7mm).

The results in Table 1 show that the base case model (where surface heat transfer is neglected) underestimated the bulk air flow by 20% compared with the computed htc method (case 5C). Using a fixed htc for all surfaces instead of a computed htc reduced the bulk air flow by 13% assuming a 'U' value for each boundary of 0.45 W/m²K (case 5A) and increased the bulk air flow by 10% assuming a 'U' value for the boundaries of 4.3 W/m²K (case 5B).

These results indicate that using a fixed htc with a coarse grid is significantly preferable to ignoring surface heat transfer altogether. In most circumstances, the relatively small enhanced accuracy of the computed htc method is unlikely to be able to justify the very considerable increase in computing time which would be required for the very fine grid.

Model Detail for Furniture and Heat Gains (Variant 6)

The results in Table 1 show that the representation of furniture, equipment and people made a significant difference to the bulk air flow. Interestingly, a relatively detailed representation (case 6B) gave very similar bulk air flows as the base case in which the space was devoid of any physical objects. In a run which included only the desk surfaces (case 6A) the bulk air flow was nearly 30% higher than the base case. It was also noticeable in this run that the flow pattern was significantly changed. In the most detailed representation (case 6C), where heat gains emanate from the occupants themselves rather than being included in a volume heat source, the bulk air flow was 14% lower than the base case and 33% lower than the run with desk surfaces only.

These results indicate that including room contents explicitly can significantly affect bulk air flow rates as well as penetration. However, often room layouts are not known when the CFD analysis is being undertaken eg at an early stage in the design, and to include internal details would increase computing time.

CASE STUDIES

Concert Hall

Background to the Project

The brief required a concert facility with seating for 350 people and a stage for a one hundred piece orchestra. It was desired that the auditorium would be naturally ventilated during the summer provided that such a strategy could be shown to avoid unacceptable overheating.

In the proposed design fresh air for the stage area entered via two vertical builders work ducts at each end of the stage. From these the air would feed into a plenum running along the back of the stage and thence through a series of low level grilles onto the stage. Fresh air for the audience was designed to come primarily from a large louvred opening which would supply a triangular cross section plenum running beneath the raked seating. The air would enter the auditorium through grilles in the step risers under each row of seats. Air extract from the auditorium was via a high level 'lantern' with a glazed roof and louvred openings on each side, positioned vertically above the stage.

CFD Studies

CFD was used to predict the bulk air flow rate through the auditorium in the worst case, no wind, situation and to examine fresh air distribution between the audience and orchestra.

For the original CFD study the decision was taken to model a 2D 'slice' in line with the intake point beneath the seating, as this allowed more time for sensitivity runs. This model represented 1/15.6th of the auditorium volume. All air path openings were modelled as 1/15.6th of the actual opening area with a loss coefficient of 1.04 and a free area ratio of 0.5 to represent the combined resistance of grilles and acoustic baffles.

The reliability of the 2D model was tested by creating a full 3D model.

Summary of Results

The 2D model overpredicted the total bulk air movement through the space by 7% compared with the 3D model. The predicted flow to the orchestra was some 36% higher in the 2D model whilst the flow to the audience was 5% higher. These results corroborate the conclusions of the cellular office test case that in appropriate circumstances it is possible to predict bulk flow reasonably well using a 2D modelling approach but that air flow patterns within the space are difficult to capture.

3 Storey Office Building

Background to the Project

The project involved a speculative 3 storey office building, rectangular in plan some 17m deep and 50m long. Floor to ceiling height was 2.8m. Initial proposals included a fan coil air conditioning system. However, in response to a request from the client for a 'low energy' approach, revised proposals were put forward incorporating a mechanical displacement ventilation scheme.

The brief specified internal gains as $10W/m^2$ for lighting, $23.6W/m^2$ for equipment and $10W/m^2$ for people.

CFD Studies

CFD was used to examine the adequacy of natural ventilation for a 6m deep perimeter zone in summer. The worst case "no wind" situation was modelled in which the air change rate would be induced by buoyancy forces alone.

For the original CFD study the decision was taken to model a 3D 7.5m wide slice of an open plan area of a single representative storey of the offices. This original model was compared with a model comprising a 3D 7.5m wide slice through all three floors and including a 1m depth of outside air at each facade.

Summary of Results

Exclusion of the outside air increased the predicted bulk air flow rates to the space by 13%. The bulk flows to each floor of the three storey model were within 9% of the flow to the single storey model. However, modelling three floors highlighted significant differences in predicted temperature profiles for each floor. This occurred because air exhausted from a lower floor rose adjacent to the building and was entrained into the floor directly above. In reality, such a flow pattern might be mitigated by fairly slight wind movement. For the cases considered, the temperature profile predicted for the single floor model was very similar to that for the ground floor of the three storey model.

These results again confirm the conclusions from the cellular office test cases that bulk air flows are approximately the same in a multi-storey model as in a single storey one, but that internal temperatures will increase with height up the building through entrainment of air from lower floors.

Office Building with Central Atrium

Background to the Project

The project involved a speculative new office building which sought to avoid the need for air conditioning. Windows could be opened in summer to allow occupants to add natural ventilation to the 'background' mechanical ventilation which supplied fresh air to the offices from floor nozzles. Air extract was via a network of ceiling ducts connecting the offices to an atrium and was driven by buoyancy forces created in the atrium.

CFD Studies

CFD was used to predict the bulk air change rate in the offices induced by the stack effect in the atrium.

For the original CFD study, a 3D model of the atrium was created. The louvred openings of the ducts into the atrium were modelled as holes with a 50% free area to allow for the louvres. Louvred outlets in the atrium roof were similarly represented. This treatment of the openings was compared with an alternative in which each opening was represented by a hole with an area equal to the actual open area. The louvred openings of the ducts to the atrium were modelled explicitly (ie by a series of individual holes) whilst the louvred outlets in the atrium roof were modelled as holes with areas 50% smaller than the overall louvred area.

Summary of Results

The original model with openings represented by their full louvred area and a 50% free area ratio predicted a bulk air flow about twice as high as the alternative model in which the openings were represented by holes half as large but with the same actual open area. As with the results from the cellular office test cases, this result indicates that a detailed representation of openings should be employed if resources allow; alternatively, it is likely to be more accurate to use unobstructed openings with the correct open area rather than correctly sized openings with a uniform generic resistance to flow (ie a mesh) with a free area ratio equal to the open area ratio of the actual opening.

CONCLUSIONS

A methodical series of single parameter sensitivity studies using a cellular office test case, corroborated by case studies on real building designs, have shown that significant care needs to be taken when making simplifying assumptions for modelling natural ventilation. The study specifically identifies the most appropriate methods for representing windows, choosing the dimensionality of the model and deciding other issues such as whether to include the outdoor environment and how to represent internal detail. A more complete version of the guidance in this paper is available in a fully illustrated Practitioners' Guide.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the 50% financial contribution for this work provided by the Department of the Environment.

80