ABSTRACT

Computational Fluid Dynamics (CFD) software is increasingly being used to predict the effects of wind on buildings and on the people in and around them. It is well suited to studying the effects of wind speed on pedestrian comfort within and around buildings. The technique is known as Computational Wind Engineering (CWE). This paper presents examples of how the authors have used commercial CFD codes for CWE to study the inclusion of new buildings in a campus at the University of Cambridge and inform the design team of its effects on human comfort. Another example is shown where comfort was studied, involving the analysis of complex fabric structures. Key issues of interest to designers are highlighted and areas requiring further work are identified.

INTRODUCTION

Computational Fluid Dynamics (CFD) software is increasingly being used to predict the effects of wind on buildings and on the people in and around them. Large buildings significantly affect airflow and influence ground level wind speeds around their base. This can lead to the development of very high wind speeds around the base of a building, rendering certain areas unsafe for pedestrians. On a smaller scale, moderate wind speeds may make areas uncomfortable and therefore unsuitable for, say, a street café or park bench.

A number of methods exist for assessing human comfort, many developed independently by different researchers during the 1970s and Melbourne (Melbourne 1978) found that there was remarkable agreement between them. The BRE have performed a review of this area and BRE Digest 390 (BRE 1994) summarises the techniques used to assess wind speed and methods for relating wind speed to human comfort. They found that people generally accept occasional high wind speeds as a fact of life, but object to particularly accelerated winds at specific locations.

For certain activities, such as sitting at a café, the absolute wind speed is most important and in these cases the number of days during which a critical wind speed will be exceeded may be the most useful measure of serviceability.

The technique that is most commonly employed at present is wind tunnel testing in a Boundary Layer Wind Tunnel. Anemometers or flow visualisation techniques (most commonly scouring methods) are used to assess wind speeds and identify areas where the wind speed is acceptable for different activities.

Consultants are now starting to turn to CFD techniques for this work. The BRE (BRE 1994) and others have assessed the potential of CFD for prediction of wind speeds around a complicated urban environment. CWE has been used to resolve planning issues related to the effect of massing building within city landscapes.

CFD CODE USED

The CFD code used for these studies was the commercial package, CFX versions 5.3 and 5.4. Structured and unstructured meshes were used and the turbulence models most generally employed were of the k-ε family, in particular the RNG k-ε model.

VALIDATION

The first step towards validation of CWE prediction has been carried out by comparing the CWE results to wind tunnel tests or full scale measurement (Easom and Wright 2001). Full scale testing was carried out at the Silsoe site (Hoxey et al. 2000) where a regular full-scale 6m cube structure has been monitored over a long periods of time. The data has been compared with CWE and the results show close agreement when the RNG k-ε turbulence method is used. Wind tunnel tests on the other hand widen the scope for validation as non-regular buildings can be measured and results compared with CWE simulations. In Comparing the results of CFD simulation with wind tunnel testing, the work carried out by Miles and Westbury is useful (Miles and Westbury 2002). The CFD package used was CFX 5.5 with an unstructured mesh of tetrahedral and prism elements. They
compared results for the standard k-ε, RNG k-ε and k-ω models, again finding that the RNG k-ε model gave the best results.

**METEOROLOGICAL DATA**

Meteorological data are obtained from the closest weather station to the location studied. The data sometimes need to be adjusted for the site altitude and topography. The data set used is the hourly mean velocity frequency distribution. With agreement with the client, a percentage of velocity exceedance is chosen and used to produce a velocity profile for the model.

**CASE STUDY 1: SIDGWICK CAMPUS, CAMBRIDGE UNIVERSITY, UK**

This project involved assessment of the impact of proposed new buildings on environmental conditions. The microclimate affecting the Sidgwick Avenue site was simulated to study the urban airflow regime and assess the impact of proposed new faculty buildings.

The study provided a context for reporting building performance as issues arose throughout the design process. A full 3D CFD model of the development was constructed to investigate the particular effects of the most onerous wind directions.

A site survey was carried out to facilitate appropriate model generation and a model built using the CFX software.

Locations within the model can be assessed for comfort and safety. Both are related to the pedestrian, the former relates to the activity and the second to the level of distress experienced. The methodology developed at BRE has been used here. These criteria have been widely accepted for this type of study and are comparable with international guidance. The criteria have been developed around the Beaufort scale, extending its applicability to environments in and around buildings, as follows

<table>
<thead>
<tr>
<th>Wind speed category</th>
<th>Limit</th>
<th>Activity</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0-4 m/s</td>
<td>Pedestrian sitting for a long time, Entrance doors</td>
</tr>
<tr>
<td>B</td>
<td>4-6 m/s</td>
<td>Pedestrian standing or sitting a short time</td>
</tr>
<tr>
<td>C</td>
<td>6-8 m/s</td>
<td>Pedestrian walking / strolling</td>
</tr>
<tr>
<td>D</td>
<td>8-10 m/s</td>
<td>Business walking Including cyclist.</td>
</tr>
</tbody>
</table>

Areas of high air velocities in the models were identified and analysis performed of likely occurrences of certain conditions – their frequency and correlation with air temperatures from Met Office data. Air wind velocities were grouped in bands according to table 1, and a comfort map was produced, indicating the location where certain activities could be carried out (Figure 1).

![Figure 1 comfort map at 1m above ground level – southwesterly wind](image)

Although air is accelerated through areas of the site, analyses showed that for both on-site localities and in the surrounding area winds are likely to be below distress and safety criterion.

**CASE STUDY 2: ASHFORD DESIGNER OUTLET VILLAGE, KENT, UK**

Located on the outskirts of Ashford in Kent, the 30,000m² continuous tensile canopy is the largest in the world. A study of the Food Court of the Ashford complex was proposed. The aim was to simulate the area in order to engineer a solution to problems that were occurring with draughts in occupied zones (figures 2 and 3).

Figure 2 shows a 3D view of the solid model used for Ashford. The tensile structure was model in great detail close to the area under study, whilst further away from this area a smoothed curvilinear structure was used. The computational mesh was similarly refined in the area studied.

![Figure 2 : 3D view of CWE model](image)
This involved CFD analysis of existing structures (see Figure 4), followed by a parametric study of possible solutions, likely to be the erection of screens within the occupied areas: glass, membrane or hedges. Based on meteorological information from the Ashford area it was decided that analysis for a number of wind conditions was required.

The CFD model of the site includes all relevant structural elements. It has the flexibility to add and remove any number of elements at any stage and assess the effect on remaining structures and the environment on the site.

A parametric study was carried out to assess the efficiency of design solutions. Wind speed frequency and direction analyses with Met Office meteorological data were carried out to give the frequency of occurrence of adverse environmental conditions in occupied zones.
Figures 5, 6 and 7, show wind speed around the food court area, the speeds are given at a height of approximately 1.2 m - a representative height at which a food court customer might experience winds. This plot is intended purely for comparative purposes against the plots of the models with hedges present.

In the other two models, hedges are present up to a height of 2.25 m. They completely surround the seating areas. There is some effect of the hedges to be seen, although it only amounts to a slight reduction in air speed around the second set of seats close to the plane of symmetry down the centre of the space. There does not appear to be a reduction in air speeds in the vortex seen around the main seating area.

The second modified model saw the permeability of the hedges decreased by 4 orders of magnitude, in much the same way as was done for the simple models earlier in this report. The change in air speeds between Fig 6 and Fig 7 now becomes apparent. The air speeds within the seated areas are reduced dramatically – the only cause for concern is that the areas between the hedged regions experience channelling effects and therefore higher air speeds.

ADVANTAGES OF COMPUTATIONAL WIND ENGINEERING

The advantages of a computational solution over traditional wind tunnel testing are manifold. The following points are key to a multidisciplinary engineering practice such as Buro Happold.

- Flexibility in altering the model. The major cost and delay in wind tunnel testing is in preparing the physical model. CFD offers the option of altering and refining the model to examine more design alternatives.

- Speed of analysis. A CFD model can be built from an architectural drawing and analysed in a matter of days, whereas preparing a physical model and arranging for a wind tunnel test may take a several weeks. It is now possible to model large developments plus their surroundings, using up to around 4,500,000 cells. Using parallel processing on standard desktop machines these problems can now be solved in 2-3 days.

- No scale effects. It is impossible to build a scale model in a normal boundary layer wind tunnel that matches the full-scale effects. The reasons for this are explained by Simiu and Scanlan (Simiu and Scanlan 1996). This becomes particularly important for very large structures and developments where a very high degree of scaling is needed or where local effects must be examined.

- Model permeable structures accurately. Permeable structures such as hedges and screens can be directly represented in the CFD model.

- Integrate with design and detailing process. As design and construction are increasingly automated, with computer models of the building being passed between architect, engineers and fabricators, design methods that fit in with this process are increasingly attractive. The information generated by the modelling can be presented directly to the designer. The process is dynamic and interactive meaning that new design changes can easily be implemented and tested, improving building design and pedestrian comfort.

This can present a challenge to the modeller in that costs and time must be controlled during these design changes. Flexible appointment and payment systems may be needed to allow the optimum balance between design cost and building performance.

- Measurements can be taken at all points simultaneously. If measurements are only taken at certain key points in the wind tunnel the engineers must first assess which are the key points. For complicated structures this may be neither trivial nor intuitive and initial flow visualisation studies may be required prior to placing all the instrumentation.
REQUIREMENTS FOR FUTURE DEVELOPMENT

Whilst computational wind engineering has now developed to the point where it can be considered a useful design tool in a number of applications, there remain a number of barriers to its wider application, in particular for safety-critical applications such as calculating structural loads.

Chief amongst these is the lack of good studies comparing CFD results with full-scale measurements. Comparisons with wind tunnel tests are of little value as it is unclear whether any discrepancy is due to errors in the CFD prediction or in the wind tunnel measurements.

The problem is compounded by a lack of clear guidance on what constitutes good practice in CFD analysis for CWE problems.

CONCLUSIONS

Examples have been shown of how CFD codes for CWE have been used commercially. These examples included the use of a comfort scale to demonstrate the effect on pedestrian comfort of wind circulating around and inside open building structures for two very different projects.

Key issues were highlighted that are of interest to designers, as well as about the use and applicability of CFD for CWE.

Finally, areas requiring further work to allow the development of CWE are identified.

REFERENCES


Miles, S. D. and P. S. Westbury (2002). Assessing CFD as a Tool for Practical Wind Engineering Applications. 5th UK Conference on Wind Engineering, University of Nottingham, WES.
