A COMPARISON OF WIND TUNNEL AND CFD METHODS APPLIED TO NATURAL VENTILATION DESIGN

D.K.Alexander, H.G.Jenkins , P.J.Jones
Welsh School of Architecture
Bute Building, Cardiff
CF1 3AP, U.K.

ABSTRACT

The design of a naturally ventilated atrium was assessed using both wind tunnel and CFD methods to appraise and modify the response of the system to wind forces. The initial design was expected to be susceptible to flow reversal due to wind forces opposing and ultimately defeating buoyancy forces. Several design options were assessed by both methods.

Both the methods were able to provide good information to guide the design development. Crucially, the information and guidance from both methods was consistent; that is either method could have led the design development to a similar final result. Each method has, of course, advantages and limitations, and to some extent these are complementary.

INTRODUCTION

When designing a naturally ventilated building a knowledge of wind pressures on the external openings are often required to allow the prediction of ventilation performance. In many cases ventilation due to buoyancy alone (i.e. a low or no wind condition) will not be the "worst case" scenario, and the effect of wind must be considered during the natural ventilation design. Wind pressure data can be generated either through wind tunnel measurement or computational fluid dynamics (CFD) predictions. While advanced CFD codes, and wind tunnels, will remain in the hands of design consultants, there have been improvements in the availability of high-powered computing and CFD software targeted at the design field.

The question arises; do the low-end CFD methods now becoming available provide the same design advice that may be produced through design consultancy using more sophisticated tools?

In the course of a recent investigation [Jones, Alexander], the design of a naturally ventilated atrium was developed using wind tunnel methods to appraise and modify the response of the system to wind forces.

The opportunity was taken to compare the wind tunnel results, and in particular the direction the design improvements were being lead, with those that could be obtained by a limited use of CFD, such as may be feasible within a design practice. The CFD calculations were carried out using the commercially available code FLOVENT®.

BUILDING DESIGN

The investigation was carried out on the design of the Saga Group Headquarters. This is to be a prestige commercial office building, designed by Michael Hopkins and Partner, with Ove Arup and Partners (Figure 1). The natural ventilation strategy is intended to control the thermal conditions in the atrium space, in order to prevent summertime overheating. The office spaces are mechanically ventilated.

The designers' preferred solution was to have inlet and outlet openings on the front facade of the atrium at low and high locations respectively, leaving the top of the atrium structure free to be used as a terrace for the director's suite. The building site is located on the south coast of the UK, with the atrium facing southwards to the sea. Therefore on-shore winds are prevalent and concern was raised over the effect of wind pressures on the use of the atrium facade openings as air inlet/outlet devices. Initially wind tunnel measurements were undertaken to investigate this issue.

WIND TUNNEL MEASUREMENT

The wind tunnel at the Welsh School of Architecture is an adiabatic atmospheric boundary layer wind tunnel with a working area of 4m x 2m x 1m high. It can be configured for a number of different atmospheric boundary layer profiles. The site to be modelled in this investigation is on a rising slope, facing the sea. There is low level building to the south and east and woodland to the west. The vertical wind velocity profile chosen for these tests simulated that found in open areas, terrain category 1, figure 2. This was felt to be the best approximation of the site conditions for prevailing winds. The boundary layer simulation is adequate for models occupying the lower 1/3 of the wind-tunnel.

A physical scale model of the proposed building and its surroundings were made to a scale of 1:250. The model office building was approximately 30x30x15 cm in size. The office block model is shown in figure 3. Pressure tapping points were placed at 46 locations on one office block model, mainly on the atrium facade. Pressure measurements were made using a tunnel air speed of approximately 8m/s. This speed would allow sufficient flow around the model to provide adequate modelling of Reynolds scale effects. Averages of the surface pressures were taken at each point over 10 seconds. Due to the time scaling of the modelling process, this is equivalent to an average over approximately 1 hour at full scale. Thus short term turbulence effects are not considered in these results.

The primary data produced from these tests are mean wind pressure coefficients (Cp) for each tapping point. Cp is a dimensionless coefficient denoting the ratio of the wind pressure at a point on the surface to the potential free air wind pressure for the site at some reference height Z. For these tests the reference height was the buildings roof height. Reference pressures were measured upstream of the model building.

The pressure coefficient data can be used to estimate the mean wind pressure at a point on the facade for any wind speed from the relation;

$$P = Cp * (1/2 \rho Uz^2), \text{ where}$$

ρ is the air density, and
Uz is wind speed at height Z, in m/s,
P is pressure in Pascal (Pa).

Cp values are of course dependant on wind direction.

Initial testing of the design indicated that for the prevailing wind, the proposed vertically sloping outlet areas would have a relatively high positive pressure relative to the bottom of the atrium facade (the proposed inlet area). This wind induced pressure difference would oppose the buoyancy pressures set up inside the atrium due to solar gains, as illustrated in figure 4. This could lead to

- flow reversal at high wind speeds, a continuous air entry at the top of the atrium,

or, more importantly, to

- the wind and buoyancy forces balancing and negating each other.

This latter case would leave minimal ventilation caused only by wind turbulence. The wind tunnel measurements suggested this could occur at moderate wind speeds (i.e. 3m/s).

This was clearly undesirable and thus several design options were assessed. An obvious solution to the problem would be to move the outlet to the terrace area at the top of the atrium, where there were strong negative pressures. However, this was not suitable for architectural reasons. Therefore, a number of fixtures and devices were tested, to attempt to produce a relative negative pressure at the sloping vertical outlet.

The most promising of these was a 'wing' type of wind deflector which shielded the outlet from direct wind pressure, and promoted a smooth air flow past the outlet. Wind tunnel testing indicated that such a device would reduce positive pressures at the top surface of the atrium and so increase suction pressure at the atrium outlets. With such a device, wind and buoyancy forces worked together, through a wide range of wind directions.

A range of variants on the wing design were tested, these are shown in figure 5. The effectiveness of the device was found to be sensitive to the size and placement of the wing, and the effect could be disrupted or negated by the addition of solid shading devices or balustrades on the terrace area.

The alterations suggested through this work have been incorporated in the final design, as shown in figure 6. This correspond to case e in figure 5, with both the solar shading and balustrade in the final design being open or porous.

CFD

As part of the research exercise, the design investigation was repeated using CFD methods. The intent was to compare the design advice generated by the two methods.

Flomerics® FLOVENT® version 1.401/33 was used for this exercise. FLOVENT® had been in use in the research group for some time, but previously had not been used as a “numerical wind tunnel”. This exploratory use of the tool would arguably emulate the application CFD by members of the design team rather than by external consultants.

The simulations were carried out on a standard 90mHz Pentium® PC computer. Due to restrictions in available computer power and memory, and in access time to facilities, the CFD simulations were carried out in 2D only. That is, only a vertical section through the centre of the building was considered. This is less than ideal, as only face-on winds could be considered, but it represents a not uncommon option to the CFD user.

The CFD simulations were set up only to consider only external air flow, no internal air flow was specified, and thermal calculations were disabled. Thus the calculations simulated the adiabatic wind tunnel measurements. The CFD domain extended considerably beyond the extent of the building.

An example geometry for the building and surroundings are shown in figure 7. The feature to the right of the building represents rising ground behind the offices. The grid was set at 0.5m spacing around the building, decreasing upstream (to the left), downstream and upwards. The CFD grid contained approximately 29000 cells, well within the memory constraints of the system.

The pressure data produced by FLOVENT were absolute pressures within each cell. Surface pressures were determined from those of adjacent cells. In calculating ventilation, it is the pressure difference between openings that is important, and this was the parameter used for testing. In the data presented, the CFD surface pressures were normalised to acheive the same pressure value at the bottom of the atrium between cases.

In the simulations for each case, 2000-3000 iterations were allowed (requiring approximately 5 computer hours each case). Most cases did not achieve full convergence (as defined by FLOVENT, a continuous reduction in field residuals to below 0.5% of the total flux) but rather settled to an oscillating residual that could not be reduced through further calculation, variation of relaxation factors or other computational option. The solutions at varying points on this oscillation were subtlely different, primarily in the location of eddies, but general trends of flows were found to be similar. This phenomena was interpreted as representing the unsteady nature of the flows being simulated and accepted as such rather than as an indication of an error in code or data. Individual cases were stopped near minima of these oscillations.

COMPARISONS OF RESULTS

This exercise was not intended to provide a validation of CFD methods, but rather to test the design guidance provided by a simple application of CFD as opposed to that provided by wind tunnel testing.

The qualitative visualisation of airflows can provide the designer with indications of problem areas and clues to solutions. A comparison of wind tunnel and CFD visualisation of wind flow patterns over the atrium facade (Figures 8 and 9) indicate close similarity between the two methods. Both for instance identify the same region near the proposed outlets as having highest wind pressures (as indicated by the location of the flow splitting and stagnation point).

Quantitative data provides the designer with the information to make choices and assess the impact of alternative solutions. Table 1 summarises the numerical results from the two methods; the parameter compared is the pressure gradient; the difference in pressure coefficient (or absolute pressure for the CFD method) between the top and bottom openings of the atrium. A negative difference implies that wind pressure opposes buoyancy pressures. A positive pressure difference is therefore desirable in this instance.

The options investigated included two different sizes of the shielding wing, and considered the interference between the wing and other (non-ventilation) features under consideration for the area at the top of the atrium; balustrades and solar shading awnings. These are shown in figure 5.

Both methods highlight the same optimal alternatives; cases 5e and 5f. Both methods also highlight the sensitivity of the effectiveness of the device to the presence of other nearby features; e.g. a solid solar shade reduces the effect of the wing by deflecting the air flow downwards onto the terrace area..

Figure 10 illustrates the change in surface wind pressure coefficient (pressure for CFD) along the vertical midsection of the atrium, for several of these options. The decrease in wind pressure at the outlet position as predicted by both methods can be readily seen.

Thus the data from the two techniques appeared to provide similar trends and indicate the same design solution. That is that a shielding “wing” can produce a considerable suction pressure at the outlet position, that a larger wing produces a better effect, and that the result can be detrimentally effected by other architectural features of the design. There are some differences in detail between the methods, but for such a coarse grid CFD calculation, this agreement is promising.

COMPARISONS OF METHODS

A wind tunnel is a large piece of capital equipment and could only be seen as part of the design process through access via consultants. Testing costs for such an investigation as described may cost £5k-£10K, or greater.

However, due to its nature, a wind tunnel highlights the dynamics of a situation, showing unstable or turbulent areas, which may be unsuited for ventilation inlets or outlets. CFD analyses, on the other hand, typically determine steady or average flow solutions, and so information on instability or turbulence effects can be lost.

Wind tunnel investigations bring a further important advantage; because of the involvement with physical scale objects the analysis and results can involve the designers to a greater degree than that for a computer based method. This may allow a more ready acceptance of results and suggested alterations to a design.

Computer methods such as CFD are notionally easier to access for a design practice, but still require a considerable capital outlay for high powered computers, software and may involve recurring licence fees. In addition, the cost of training can be high. The experience described here shows however that useful design information can be derived from even a limited application. It must be noted that the potential for the generation of misleading results with untrained users is high. Ultimately there is no substitute for experience.

CFD methods can of course be enhanced to full 3D domains. The wind pressure boundary conditions can be combined with thermal transfer calculations and can be extended to determine internal flow patterns, as shown in figure 11. Due to the nature of physical scaling laws, heat transfer and internal air flows cannot be adequately modelled in a wind tunnel investigation such as described here.

CONCLUSIONS

Both the wind tunnel and CFD methods were found to be able to provide good information to guide design development. At a global or building wide scale, the information and guidance from both methods was consistent; that is either method could have led the design development of building form to a similar result. On a smaller scale, comparisons were also good, suggesting that CFD methods are suitable for the detailed design of features, though a more sophisticated use of CFD (e.g. a more refined grid) may be required. Thus the CFD method could be successfully used in a limited sense within the design office.

Each method has, of course, advantages and limitations, and to some extent these are complementary. That is whilst physical scale modelling provides a more immediate view of the dynamic nature of the external air flows, CFD methods allow the determination of internal and buoyancy driven flows.

The wind tunnel is inherently three dimensional and capable of dealing with unsteady conditions, and as it deals with physical scale models its use may involve the building's designer(s) to a greater degree. A model can be readily altered and qualitative visualisations rapidly carried with the designer present. Wind tunnels are unfortunately not readily accessible however and are not able to simulate internal flows, which is often the embodiment of the design intent.

Due to decreasing hardware costs and increasing performance, CFD methods are becoming more readily accessible, and have the advantage of being able to consider internal flows and heat transfer problems. However there is still a not inconsiderable cost involved in entering the computational field, not the least component is the necessary commitment to user training.

In natural ventilation design however, consideration of wind effects are important in the development of a successful design, and so the use one or other of these tools is of great importance.

ACKNOWLEDGEMENTS

This investigation was carried out under an EPSRC grant, reference GR/K/19129 [Jones, 1996].

The contributions of John Berry of Ove Arup and Partners and of Brendan Phelan and Ian Sherritt of Michael Hopkins and Partner in supplying the case study design is gratefully acknowledged.

FLOVENT is a registered trademark of Flomerics Ltd. The authors are not associated with Flomerics.

REFERENCES

Jones, P.J., Alexander, D.K., Jenkins, H.G., "EPSRC final report: Investigating the Effects of Wind on

Natural Ventilation Design of Commercial Buildings", EPSRC grant ref. GR/K/19129, December, 1996.

Alexander, D.K., Jenkins, H.G., Jones, P.J., "Investigating the Effects of Wind on Natural Ventilation Design of Commercial Buildings", Proceedings, BEPAC + EPSRC conference Abingdon, February 1997, pp 141-148

Condition :	**Wind Tunnel** Cp Gradient Bottom-Top (dimension-less)	**CFD** Pressure Gradient Bottom- Top (arbitrary units)
Original Design	-0.3	-0.2
Wing 1	+0.6	+1.1
Wing 1 + Balustrade	+0.2	+0.5
Wing 1 + Balustrade + Solid Solar Shade	-0.0	+1.5
Wing 2 (larger)	+1.1	+4.3
Wing 2 + Balustrade	+1.1	+4.9
Wing 2 + Balustrade + Solid Solar	+0.6	+2.8

Table 1 Comparison of wind tunnel and CFD results, in terms of trends in pressure difference between top and bottom of atrium facade.

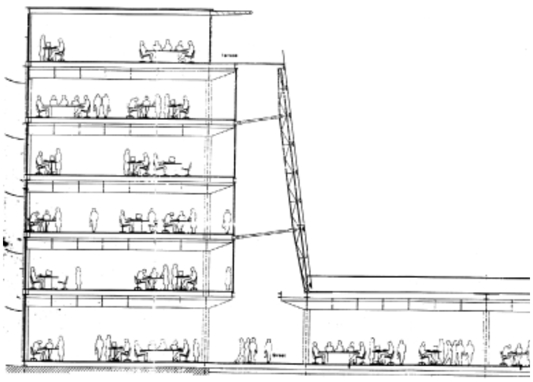

Figure 1. Initial design section showing the six levels of offices and atrium.

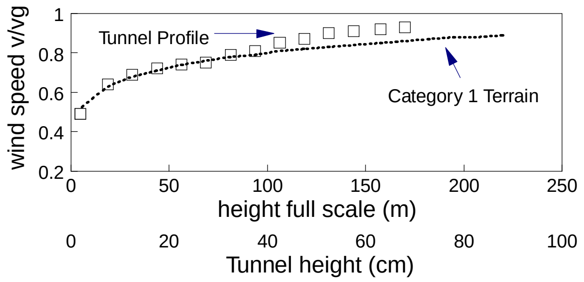

Figure 2. Boundary layer modelling in wind tunnel

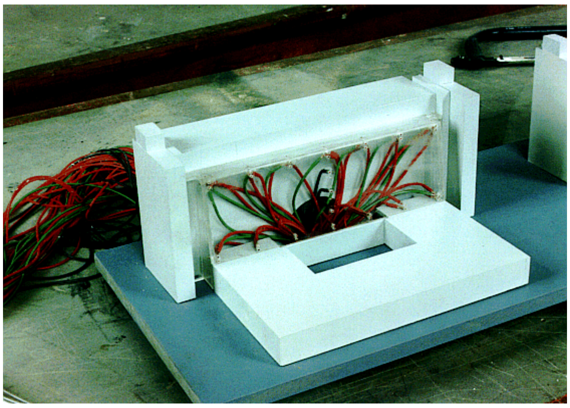

Figure 3. Wind tunnel model of design (1:250)

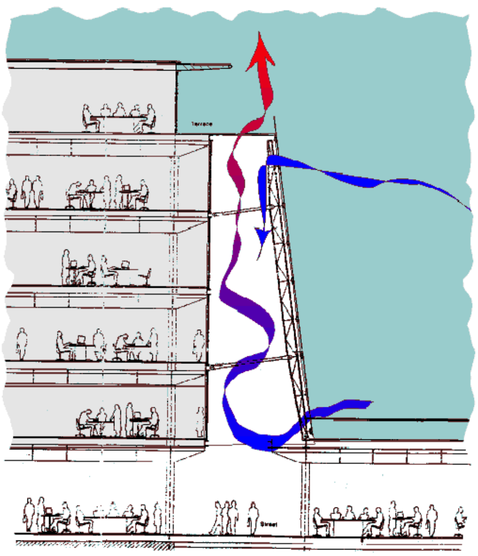

Figure 4 Possible flow regimes under wind conditions, as detected by wind tunnel tests.

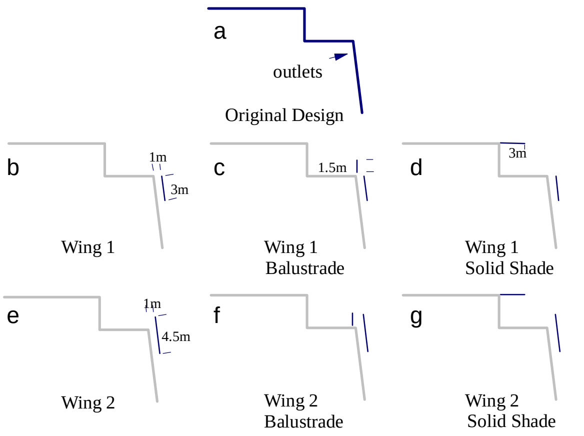

Figure 5 Shielding device configurations tested.

Figure 8 : Smoke Visualisation of wind flow over the upper part of the model atrium.

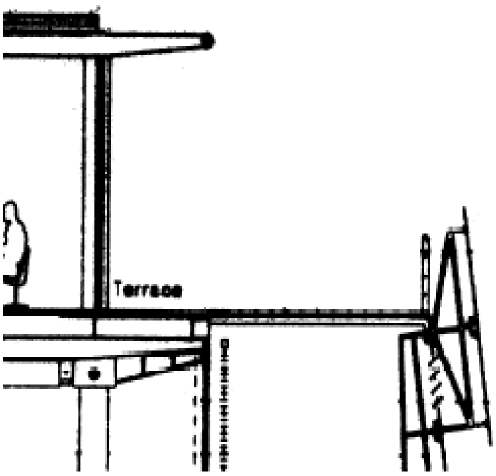

Figure 6 Final device design as adopted.

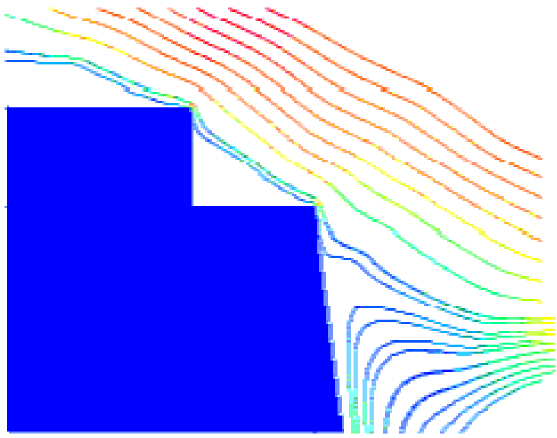

Figure 9 : CFD calculation of wind flow over the upper part of the atrium, showing streamlines.

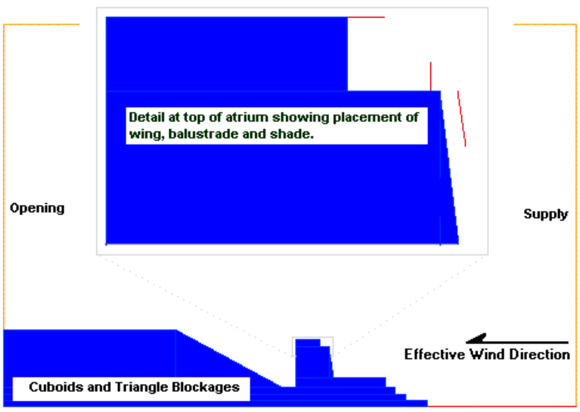

Figure 7 : CFD calculation domain and layout.

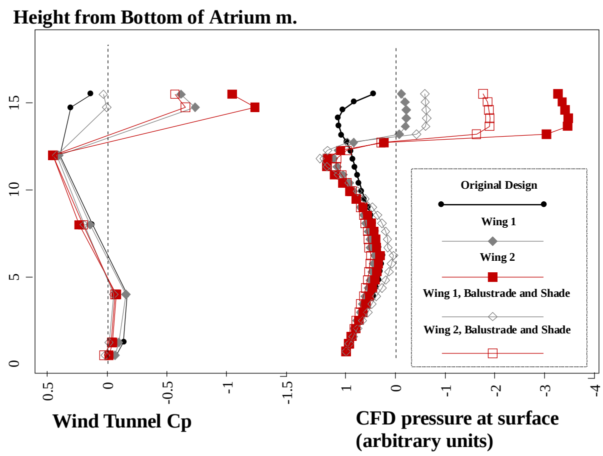

Figure 10: Predicted wind pressures on a vertical section of atrium.

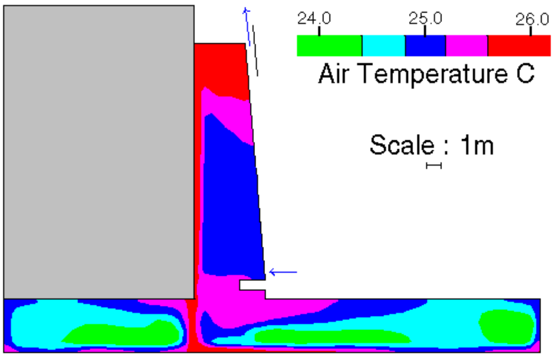

Figure 11 : Example CFD prediction of air temperature distribution in atrium.