Visualisation of air movement in buildings: an evaluation of CFD analysis in the early stages of architectural design

T.J. Kinsley, K.E. McCartney

University of Portsmouth, School of Architecture, King Henry Building, King Henry 1st Street, Portsmouth PO1 2DY, UK

Abstract

An evaluation of the potential role of computational fluid dynamics in the early stages of architectural design has been carried out. It is based on a case study of the early stages in the design of a hotel, located on the shores of the Dead Sea, which was to be cooled by wind induced natural ventilation. CFD analysis was found to make a significant contribution both at the conceptual stage, and the more detailed stages of design development.

Introduction

In a list of some 500 publications reporting on applications of CHAM’s Phoenics computational fluid dynamic (CFD) software, only 20 are explicitly related to the built environment. (CHAM, 1992). Most papers describe analysis rather than design applications. Drake and Pericleous (1989) and Markatos (1983), among others have reported on the use of CFD in building design problems. From the published examples, it appears that CFD analysis has usually taken place relatively late in the design process. It has been adopted as a technique for refining detail design, and/or to test unusual design concepts, after they have been developed to a fairly advanced stage.

The timing of the CFD introduction is significant. Architectural design usually proceeds in stages. The Royal Institute of British Architects (RIBA) publish a Plan of Work which recommends twelve stages, starting with Inception and Feasibility, and continuing with Outline, Scheme and Detail design, before proceeding to stages related to documentation and site operations. Whilst there is usually a great deal of cyclical development and revision within any particular stage, there is an attempt to establish fixed design decisions at the end of each stage. It is for example, considered extremely inefficient at the sixth stage (Production Information) when working drawings are produced, to revise early massing decisions related to height and bulk, which should have been fixed at the third stage (Outline Design). It is clear therefore that if CFD analysis is introduced late in the design process, there will be severe limitations on the scope for design revision in response to the CFD analysis results.

With this in mind, it was decided to evaluate the potential role of CFD analysis in the early stages of Outline and Schematic design as well as in Detail design.
Flomerics' FLOVENT software was used in the design of a hotel for a site beside the Dead Sea. The architect and analyst, John Kinsley, prepared the design for an Architectural Ideas competition organised in conjunction with the Passive and Low Energy Architecture Conference held in Tel Aviv in 1994, and in relation to a post-graduate course on Bio-Climatic Architectural Design (Kinsley, 1995).

**Concept Development**

With the high temperatures and intense solar radiation experienced on the site, it was clear that an innovative approach to cooling, could provide potentially large reductions in the use of fossil fuels required for conventional air-conditioning. An analysis of the climate indicated that a fairly constant wind was one of the most useful climatic resources available to assist in cooling the building. However, the direction of the wind which was relatively constant during the daytime, changed through 180 degrees, at night. This presented a very particular problem in designing the orientation of the building, and the form of the ventilation apertures.

Initial considerations focused on the concept of wind towers used in some parts of the Middle East to draw down the higher velocity winds found above the roof line of buildings (Fig. 1). The application of this concept to a larger building type, such as an hotel, could take the form of drawing wind down into an enclosed circulation space between two banks of bedrooms. (Fig. 2). Historically this type of solution was most successful in regions where there is little variation in wind direction.

In this Dead Sea location, the switch in the wind direction led to the idea of exploiting the Venturi Effect. This is based on Bernouilli's Principle which states that the pressure of a fluid decreases as its velocity increases. It is most commonly used in airbrushes, and spray guns, where the release of a gas at high velocity over the top of a tube creates a pressure difference which draws paint out of its container. It was proposed that a gull wing shaped roof, creating a constriction in the free air passage, would increase the velocity of the wind as it passed over the aperture at the top of the circulation space between two blocks of rooms. This would in turn cause a pressure difference which would induce air from the external sides of the building to move through the rooms, into the central circulation space, and then up to the roof where it would be carried away with the wind. (Fig. 3).

![Fig. 1 Traditional wind catcher depends on prevailing wind.](image1)
![Fig. 2 Wind catcher applied to large hotel using central circulation space as exhaust.](image2)
![Fig. 3 Gullwing roof creating Venturi Effect functions with wind from either side.](image3)
A more developed form of the concept is illustrated in Fig. 4, which shows the intended route for night time air movement: entering the hotel through balcony windows on the periphery, and exiting under the gull wing roof. In the daytime, balcony windows are closed to keep out the warm winds, but the Venturi Effect is used to exhaust air from the basement kitchen and restaurant, which are replenished with air which is cooled by being drawn through buried tubes in thermal contact with the underground soil which remains several degrees cooler than the daytime air temperature. (Fig 5).

Fig 4 Intended night time air circulation pattern induced by northerly wind passing through the roof.

Fig 5 Intended daytime air circulation pattern where air is drawn through underground ducts drawn by the lower pressure region created under the gull wing roof by the southerly wind.

Progress to this stage had been supported simply by an analysis of the climate and knowledge of traditional building forms and conceptual knowledge of the interaction between built form and air movement patterns, based on textbook diagrams, and reports of wind tunnel tests.

There are many questions now to answer, and hunches to be tested before it is worth developing the design further. Will the pressure difference created by the Venturi Effect be sufficient to overcome the resistance imposed by the building interior? Will the low pressure created by the roof be cancelled out by the low pressure which forms on the leeward side of a building, causing air exhausted from the windward bank of rooms to exit through the bedrooms on the 'sheltered' side of the hotel. Will the concept work if two symmetrical
Fig. 6 Sectional perspective showing design for two blocks enclosing a central courtyard.

blocks are built alongside each other to enclose a shaded courtyard? (Fig 6). What is the best geometry for the roof? What roof angle will maximise the Venturi Effect? How big should the aperture be at the top of the circulation space where the low pressure is intended to exhaust air from the building? What shape should the apertures be at the exhausts from the circulation spaces, and from the bedrooms into the circulation spaces?

Physical Model Testing in Outline Design Stage

There has long been a serious lack of design tools for predicting the effect of variation in design variables on air movement, which would enable an architect to proceed beyond this stage, without the reassurance of comparable built examples. Initially cardboard models were made and these were tested using both simple fans and in a wind tunnel. (Few architects it must be noted have regular access to expensive and bulky wind tunnel test facilities).

These initial tests indicated that the gull wing roof did appear to enhance air movement through the windward side of the building. However it did not appear possible to induce the intended air movement through the leeward side. This problem remained, even when the resistance to air flow was reduced by changing the positions of the inlets and outlets to the bedrooms, and by changing the geometry of the balustrades in order to reduce the resistance they present to the intended air stream. Attempts were made to further reduce the pressure at the roof exhaust aperture, by increasing the ratio between the opening which the roof presented to the wind, and the restriction at its centre. This also failed to generate the intended change of direction in the flow of air.

At this point further testing was clearly called for. The deadline of the architectural competition however necessitated further development into scheme design, which is shown in Fig. 6.

Following the submission of the competition drawings, a further programme of testing and development was undertaken, and at this point, the FLOVENT CFD package was made available, along with a two day training provided by Flomerics, the company who market the program. Initial tests of two
dimensional models confirmed the results from the wind tunnel tests. However, the ease of modifying the digital building model, and the speed with which 2D simulations could be carried out meant that a more exhaustive round of tests were now possible. Simulations at this stage were typically taking around 15 to 30 minutes, running on an 80486DX IBM-compatible personal computer.

After a large number of tests using the CFD program, the architect became convinced that a sufficiently low pressure was being created under the roof to ventilate the whole building. This was established by modelling the building with all inlets on the windward side closed. This demonstrated that there was sufficient pressure difference between the roof outlet and the leeward side to drive the required air flow. (Fig 7) He realised then that the introduction of a vertical screen into the central circulation space, to separate the windward, from the leeward side of the building, would enable the achievement of the intended air flow, from the periphery through the centre to the roof. This was easily tested, and confirmed. (Fig 8). The CFD testing therefore led to this crucial breakthrough, which confirmed the viability of the core design concept, and lends support to the proposition that CFD analysis can play a significant in outline design in the role of design concept testing.
One outline design problem remained unresolved. The positioning of two blocks side-by-side to create a central courtyard, caused the leeward block to function inadequately due to the wake effect caused by the windward block (Fig 9). It was clear that the relative positions of the two blocks would have to be altered, if the natural ventilation scheme was to be effective.

![Diagram](image)

**Fig. 9** The upstream block of the hotel causes a wake which prevents sufficient ventilation through the downstream block.

### Scheme Design

The following could be considered as part of the Scheme Design stage. (It is worth noting, however, that this work preceded the discovery of the effect of introducing the central screen - an essential stage in establishing the concept).

CFD analysis was used to refine the roof geometry. Here again a significant conceptual insight was gained which would directly influence the generation of any future Venturi Effect schemes. The architectural text books illustrate the application of the Venturi Effect drawing an analogy between the flow patterns observed in the classic three pipe apparatus used to demonstrate Bernoulli’s Principle, and the flows which might be expected whenever building elements which are open to the wind, converge to funnel the flow of air through a restriction. They do not emphasise that there is a major difference between the laboratory apparatus which funnels a piped flow of air, and the built environment example where air is only partially contained, and hence can be deflected away from regions of restricted flow. This became obvious in the CFD testing when parametric tests were carried out to determine the optimum roof inclination angle, and the ratio between the opening area, and the restriction.

![Diagram](image)

**Fig. 10** The gap between the flat roof and the centre of the gull wing roof is small, and the resistance to air flow deflects much of the wind over the roof.
Fig. 11 When the gull wing roof is raised, and a downturn is added to the tip of the gull wings, a greater volume of wind is channelled through. The amount of air drawn through the central circulation space of the hotel shows a corresponding increase.

Fig. 12 and Fig. 13 illustrate the increase in upward air flow through the hotel's central circulation space after addition of 300mm high wedges at the side of the aperture in Fig 13.
If the angle of the roof was very steep, or if the ratio was very large, the rate at which air could be drawn out of the central circulation space was reduced. It became clear that if resistance to the wind was raised above a threshold, then the wind would increasingly be deflected over the roof. (Fig 10) As a result, the restriction between the gull wing roof and the flat roof was increased from under half a meter to over 1.5 metres. A downturn at the tips of the gull wing was also found to increase wind flow through the gap. (Fig 11).

**Detail Design of the Wind Cavity**
The potential value of CFD in Detail design stages was exemplified in the analysis of the junction between the gull wing roof and the aperture at the top of the circulation space. The close-up images of air movement vectors around the aperture (Fig 12), led to the design hypothesis that a wedge shape on the windward side of the outlet aperture at the top of the central circulation void would enhance the intended suction effect by deflecting wind away from the aperture. The test indicated a quite unexpectedly dramatic increase in the mass flow rate through the hotel rooms, by a factor of four, due to the introduction of a wedge with a height of only 300mm (Fig 13).

**Commentary**
It was noted by the architect that the two dimensional models used for most of the analysis work could result not only in missing details, but also in misleading conclusions. He concluded, for example, that the upward flow of wind from the base of the building to the roof is unrealistic, and due to the inability of the two dimensional model to allow for wind striking the lower surfaces of the building flowing downwards and around the building as indicated in field measurements and wind tunnel simulations. When three dimensional models were adopted, the duration of the simulations extended from being measured in minutes, to being measured in hours. Clearly this problem will diminish as increased computing power reaches the designer's workbench, but in the meantime it is important to be aware of the potential for conceptual errors in interpreting 2-D results. When one also considers the potential for error resulting from the large number of often unfamiliar conventions adopted in CFD codes, one must sympathise with the opinion expressed by Fawcett (1991, p3) "The learning curve to becoming a skilled CFD user is long - typically six months of continuous use of a particular code would be a good estimate".

**Conclusions**
In this particular case study, in which it had been decided from the outset that air movement was to be a significant determinant of architectural form, there was clear evidence that CFD analysis can and did play a significant role at the outline design stage in testing the viability of the concept and overall geometry; at the schematic design stage, in decisions relating to form, proportion and dimension of building elements such as roof overhangs, and the spacing between the roof and circulation space outlet; and in detail design decisions related to components such as the fillets which redirect wind away from the outlet at the top of the circulation space.
The software and the training provided by Flomerics is sufficiently advanced to allow an architect to generate impressive results, within a few days, using two-dimensional models. However, the new, and sometimes contrasting, information that was gained when the modelling changed from two to three dimensions, was a sober reminder of the need to exercise caution. The increasing ease of use of CFD software, due to improved graphics and user interfaces, might easily lead to a situation where the inexperienced designer/analyst might be misled by the seductive CFD images. Whilst it is becoming easier to manipulate CFD software, special skills are still required in the creation of architectural CFD models, and in the interpretation of the results from CFD analysis.

References


Kinsley, T.J. (1995) Computational Fluid Dynamics as a Detail Design Tool, MSc Thesis (unpublished), School of Architecture, University of Portsmouth
