CFD INVESTIGATION OF AIRFLOW AROUND CONIC TENSILE MEMBRANE STRUCTURES

A. M. ElNokaly[1], J. C. Chilton[2] and R. Wilson[1]
[1]*School of the Built Environment, University of Nottingham, Nottingham, NG7 2RD, UK*
[2] *Lincoln School of Architecture, University of Lincoln, Brayford Pool, Lincoln, LN6 7TS, UK*
laxamhe@nottingham.ac.uk, jchilton@lincoln.ac.uk, Robin.Wilson@nottingham.ac.uk

Abstract

This paper is part of continuing research on the airflow around membrane structures. The research explores how the form and orientation of the structure itself affect the ventilation rates and the comfort level within the enclosures and in their immediate vicinity. This paper describes a study of the airflow patterns around and under conic tensile membrane structures covering open and semi-enclosed spaces using CFD (computational fluid dynamics) modelling, carried out in order to ascertain the potential of conic membranes of different forms for modifying the microclimate and improving human comfort. The reason for using CFD modelling is to give the opportunity to explore different geometries, to investigate the use of structure topology to assist in passive cooling and achieving higher comfort rates of spaces covered by membrane structures in hot climates.

1. Introduction

Since ancient days human beings have always actively shaped their habitats. According to Banham [BAN69] deployed technical resources and social organizations, "in order to control their immediate environment: to generate dryness in rainstorms, heat in winter, and chill in summer, to enjoy the built environment through acoustic and visual privacy…" Fanger [FAN70] stated that creating thermal comfort for man is a primary purpose of the heating and air-conditioning industry, and have a significant influence on building construction.

Air motion is one of the main factors that influence indoor thermal comfort [ASN68] and modifying the movement of the air around the human body helps in controlling the thermal comfort level [ALL98]. However, in terms of both physical principles and practical engineering applications, air movement is probably the least understood aspect of environmental design [JON92]. When air speed increases over the body and clothing surfaces, it can increase the convective heat losses with the surrounding air. In summer, when there is perspiration it can increase the evaporative losses at the skin surface and hence enhance the cooling sensation. The recommended upper limit of indoor air movement is usually 0.8 m/s, this will maintain comfort in a space at an air temperature 2 ºC warmer [ALL98], then that recommended in the absence of airflow. It can thus be said that in hot climates under conditions of high temperature and high humidity, discomfort can be greatly reduced by increasing the airflow [HUT01]. It is worth noting that under these hot conditions higher air speeds can also be pleasant [HUT01].

Natural ventilation can also help in reducing internal gains and minimizing temperature increase within buildings. Higher air movement within buildings is achieved by having large openings to the outside, which is a traditional cooling technique used in moderate or hot climates. In this technique the indoor temperature becomes equal to that of the outdoor, with very high air change rates. This strategy has to be coupled with efficient solar shading in order to reduce the surface radiation within the building, as it is mostly applied within lightweight structures.

The prediction and engineering of indoor environments is however a complex subject. Physical modelling has traditionally provided a means of evaluating critical spaces, although in some cases it might be rather time consuming and the capital cost of running the experiment may be prohibitive that can preclude its use [BRO01]. Recently, computational fluid dynamics (CFD) has been used to evaluate and optimise design, and it appears to offer a competitive and flexible alternative.

Knowledge of the airflow pattern and rate in and around fabric membrane structures is still relatively unexplored compared with that in more conventional structures. As with conventional buildings, understanding of airflow rate and pattern around these structures is of vital importance in order to assess appropriate comfort levels during the design process. It is the aim of this research to improve the understanding of airflow behaviour around and under conical membrane structures to assist in their design.

2. Application of CFD in Ventilation and Airflow around Buildings

Advanced building design requires information about airflow in and around buildings in order to assess thermal comfort and indoor air quality [BRO01]. The most common procedures for investigating airflow behaviour around buildings or in urban areas are wind tunnel modelling and field measurements. Computational fluid dynamics (CFD) offers an alternative to these traditional approaches, and facilitates a numerical rather than physical simulation of the flow variables [FLU04]. The airflow around bluff bodies and structures located in the atmospheric environment is complex and difficult to analyze and its pattern can differ considerably from one case to another compared with flow in a pipe or channel [MUR97]. These airflow patterns are important factors in predicting wind effects on structures and their immediate vicinity [LU99].

The application of CFD to problems that involve classical wind engineering and building aerodynamics is usually referred to as Computational Wind Engineering (CWE) [GRE99]. CWE as a branch of CFD has been developed rapidly to evaluate the interaction between wind and a building numerically. Stathopoulos [STA97] advocates the application of CWE to assess environmental wind conditions around building complexes. The CFD technique offers several advantages over traditional methods: there is no need to build a physical model, different (stable) wind conditions are easy to simulate and detailed results can be obtained where it may be difficult to take measurements from a scale model in the field.

3. The Choice of Form and Models Tested

For this study the conic shape was selected to explore the behaviour of structures such as those shown in Figure 1 as they are one of the simplest and most frequently used fabric structure roof forms. The 1:20 scale conical membrane model was used in different positions (straight, tilted, inverted) and with different heights. The conical membrane was set at 3 different heights without varying the diameter of the upper or lower rings, which were 3 cm, 7.5 cm and 17 cm high cone with closed and opened apex, at heights 20cm, 16cm and 12 cm from the base as shown in Figure 2. The cones were tested as straight and tilted in all cases. Also the same conditions were set testing a flat disc with the same diameter as the cones and at the same height from the base.

	Apex	Height of cone	Tilt
Test 1	Closed	17 cm	None
Test 2	Closed	7.5 cm	None
Test 3	Closed	3 cm	None
Test 4	None	Flat disc	None

Table 1. Cases reported in this paper

Figure 1. The use of conical membrane structures in different architectural projects (Photo courtesy TENSINET Library)

Results for test 1, 2, 3 and 4 shown in Table 1 are reported in this paper. A series of 2D and 3D models were built and tested using CFD. In parallel a series of wind tunnel tests were undertaken to visualize the airflow pattern and validate the results of the CFD modelling. The tests where designed to explore the effect the form of the roof has on the airflow, in order to inform the design of structures that enhance the natural ventilation and passive cooling of the enclosed space.
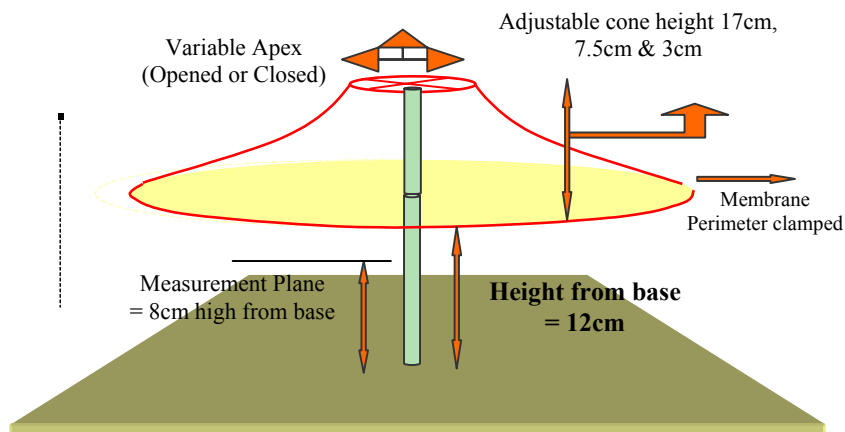

Figure 2. A diagrammatic representation of the tested models and table shows tests reported in this paper

4. Description of FLUENT

The CFD code FLUENT was used for the purpose of the current research investigation and a comprehensive description of the relevant theory can be found in both the FLUENT User's Guide [FLU93] and on the online FLUENT website "The Right Answer in CFD" [FLU04]. There is an extensive theory behind the numerical predictions of fluid flow, which is considered beyond the scope of this paper. However, an overview is presented in this section.

FLUENT ([FLU04], [FLU93]) is a state-of-the-art computer program for modelling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving flow problems with unstructured meshes that can be generated about complex geometries with relative ease. The program supports various mesh types such as 2D-triangular / quadrilateral, 3D-tetrahedral / hexahedral / pyramid / wedge, and mixed (hybrid) meshes.

Cells are created to represent, in geometrical terms, inlets, outlets, surfaces and obstructions in the flow field. The number of cells in the grid is one of the criteria that governs the accuracy of a CFD solution, and grid density is usually increased in areas where large gradients of fluid properties are expected, such as near wall boundaries and obstacles. In general, the larger the number of cells the better the solution accuracy [FLU04]. A basic structure of the FLUENT program is illustrated in Figure 3.
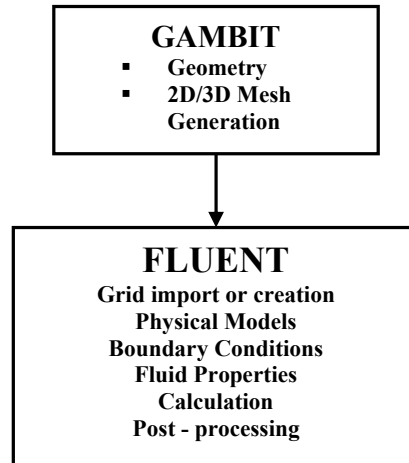
GAMBIT
- **Geometry**
- **2D/3D Mesh Generation**

FLUENT
Grid import or creation
Physical Models
Boundary Conditions
Fluid Properties
Calculation
Post - processing

Figure 3. FLUENT basic program structure [FLU93].

4.   2 D Modelling

To carry out CFD modelling in fluent first a 2D model was built in order to observe the behaviour of the airflow and its pattern to be compared to the wind tunnel visualization that was carried out. The results of the modelling were similar to those from the wind tunnel as shown in Figures 4 & 5, which raised greater interest in building models in 3D and testing them.
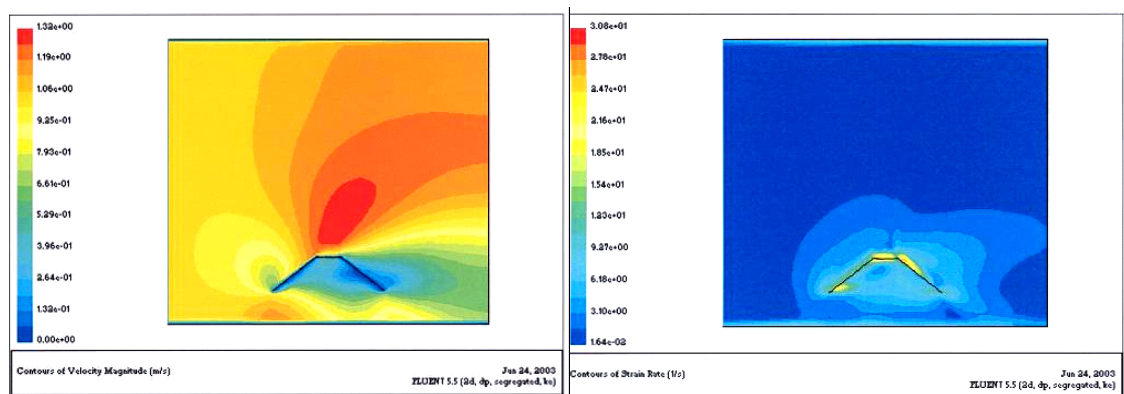
Figure 4. Contours of Velocity Magnitude and Strain rate of a 17cm high cone

Figure 5. Wind tunnel experiment showing the effect on airflow of a conical membrane with closed apex

As seen in Figures 4 & 5, for a conical membrane with a closed apex, the airflow tends to be deflected downwards into the occupied area then deflects back upwards again when leaving the area underneath the structure. The results were the same in all the tested cases of the closed apex cones, with the deflection of flow appearing to become more pronounced as the cone height was increased. This is not so pronounced in the case of the flat surface.

4. 3 D Modelling

Figure 6 represents the 3D model of the cone investigated in CFD (FLUENT), the grid used in the model is also presented. The approach to grid design in flow problems around bluff bodies is to maximise the grid density (i.e. minimize grid spacing) in the areas where the gradients of the flow variables are likely to be greatest. In the current investigations these regions are the regions around the cone itself, (i.e. under and over it) where flow separation and change of air velocity flow takes place. Figure 7 illustrates velocity magnitude contours for a vertical section in the mid axis of Test 1, which indicate a relatively stagnant zone within the cone and high velocities in the inhabited area. This behaviour follows that revealed in both the 2D modelling and wind tunnel tests.
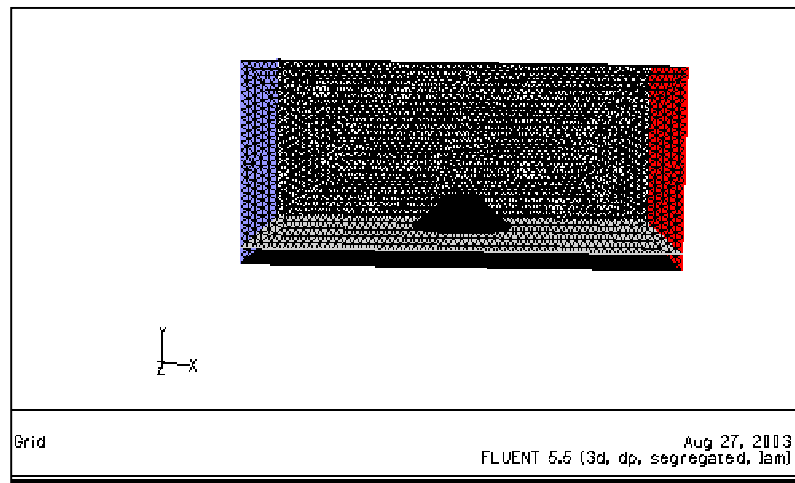

Figure 6. Depicts the 3D model of a 17cm high closed apex cone and meshing of the model in Fluent
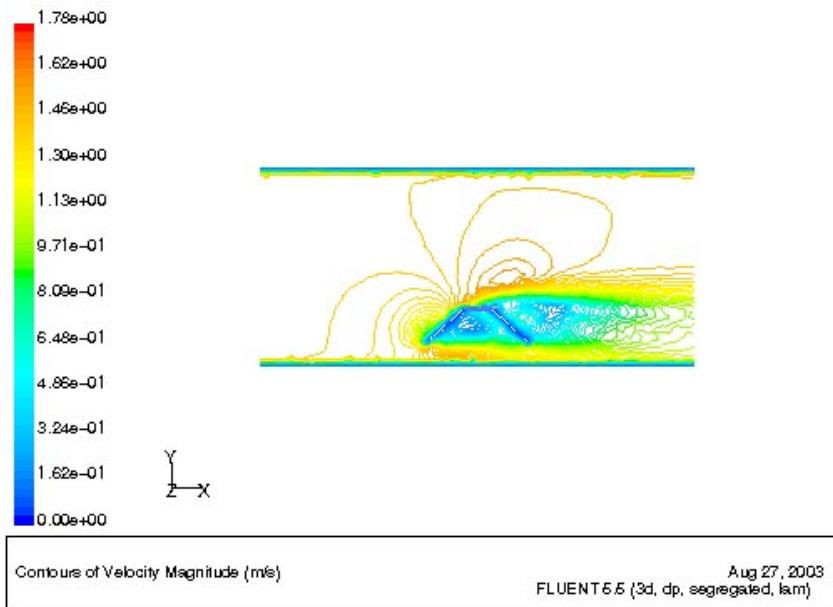
Figure 7. Contours of Velocity magnitude for a vertical section on the mid axis of a 17cm high closed apex cone at 12cm high from the base for a 3D model
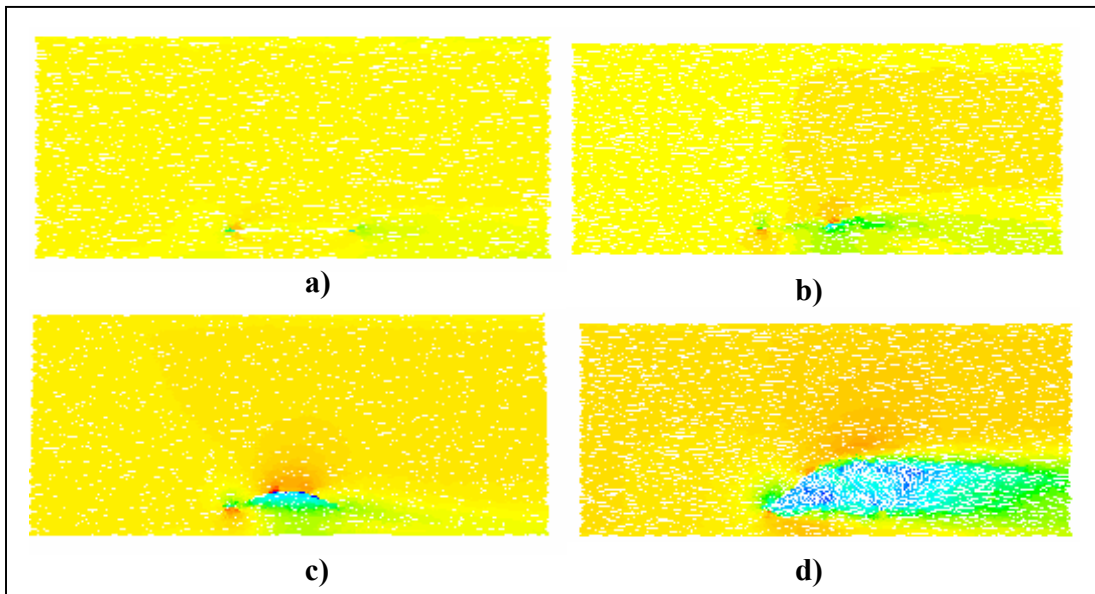

Figure 8. A vertical section in the mid axis of the closed apex cones at 12 cm high from the base for Velocity Vectors Coloured by velocity magnitude for 3D models

Figure 8 presents the variation in air velocity pattern around and under the conical structure in the 4 different cases referred to earlier in this paper. These are (a) the flat disc, (b) the 3cm high cone, (c) the 7.5 cm high cone and (c) the 17 cm high cone respectively, all at 12 cm above the base. As seen in Figure 8(a) when there is a flat disc, which has the same radius as the tested conical structure, the airflow remains steady and stable at almost all points then decreases slightly at the windward and leeward sides of the disc and at positions downwind of the obstacle. Figure 8(b) shows the air velocity under the 3cm high cone, which shows that a higher circulation happens than that shown in Figure 8(a) (flat disc). It is clear from Figures 8(b), 8(c) and 8(d) that airflow velocity in the inhabited area under the cones are higher when the conic membranes are present when compared with the flat disc case and also with the increase of the cone height.
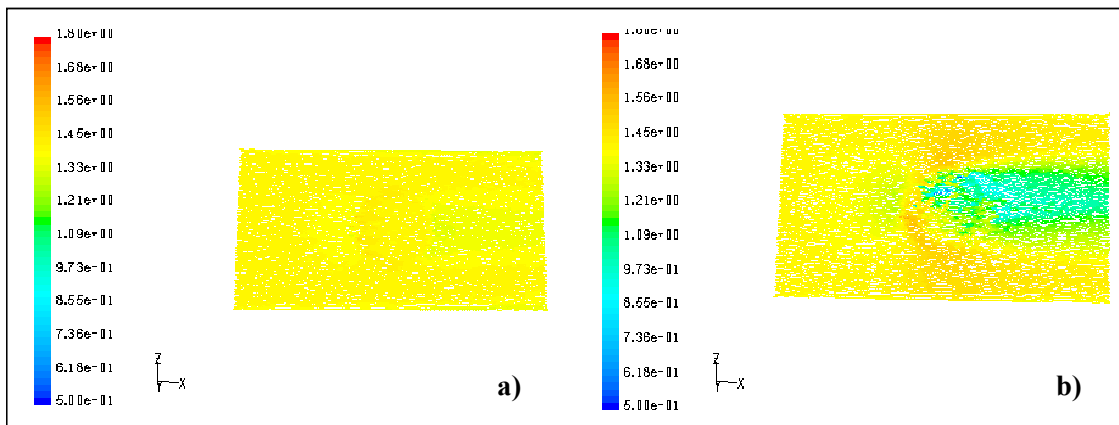

Figure 9. A horizontal section in plan at a height of 8 cm from base of (a) a flat disc and (b) closed apex 17cm high cone for Velocity Vectors Coloured by velocity magnitude

Figure 9 depicts a horizontal section in two of the cases tested which are the flat disc case and the 17cm high cone. It is clear from the figure that higher air velocities are achieved in the windward area with the presence of the 17cm high cone. Air velocity then tends to be lower at the leeward side of the cone when compared to the flat disc case. Air velocities tend to be almost the same at all points under the flat disc shown in Figure 9 (a).

4. DISCUSSION

Preliminary results of the investigation show that higher airflow velocities are achieved within the enclosure under certain conditions, for instance, with the increase in the height of the cone. This improved ventilation may enhance the comfort of occupants of the membrane enclosure, particularly in hot climates and reduce the demand for mechanical cooling systems (and consequently energy consumption). The results indicate the need for further research in this area, in order to fully realise the potential benefits offered by tensile membrane structures for modifying airflows in their vicinity and for their use as microclimate modifiers.

5. CONCLUSIONS

In this paper the reason for the CFD modelling has been discussed. Simple 2D and 3D modelling has shown that topology and orientation of a simple conical membrane structure may influence considerably the wind environment in its immediate vicinity. The results of CFD modelling were similar to data obtained from wind tunnel tests carried out on similar models and were briefly pointed to in this paper.

References

[ALL98] Francis Allard. (1998), Natural Ventilation in Buildings "A Design Handbook", 35-37 William road, London, James and James (Science Publishers).

[ASN68] T. C. Asngus and Introduction By: G. P. Crowden. (1968), Vol. 4: The Control of Indoor Climate, United Kingdom, Pergamon Press.

[BAN69] R. Banham (1969), The Architecture of the Well-Tempered Environment. London,: Architectural Press, The University of Chicago Press.

[BRO01] Charles R. Broderick, and Qingyan Chen., (2001), A Simple Interface to CFD Codes for Building Environment Simulations, Proceedings of the Seventh International IBPSA Conference, Building Simulation (Rio De Janeiro, Brazil, 13-15 Aug, 2001).

[FAN70] P. O Fanger (1970), Thermal Comfort Analysis and Applications in Environmental Engineering, New York, McGraw-Hill.

[FLU93] Fluent Inc. (1993), Fluent Users Guide. Lebanon, U.S.: Fluent Inc..

[FLU04] Fluent Incorporated Network Services (2004), "FLUENT; The Right Answers in CFD" [Web Page]. Accessed: 2 Feb 2004, Available at: http://www.fluent.com/.

[GRE99] N. E. Green, (1999), Investigations into the Impact of Traffic Pollution on Building Ventilation, University Of Nottingham.

[JON92] P. J. Jones and G. E. Whittle. (1992). Computational Fluid Dynamics for Building Air Flow Prediction - Current Status and Capabilities, Building and Environment, Vol. 27, (No. 3), pp. 321-338.

[LU99] W. Z. J. Lu., Isaac Y. F. Lun and Joseph C. Lam, (1999), CFD Investigation of Airflow around Objects with Different Shapes, Proceedings of the International Building Performance Simulation Association, IBPSA Conference).

[MUR97] Shuzo Murakami (1997), Current Status and Future Trends in Computational Wind Engineering, Journal of Wind Engineering and Industrial Aerodynamics, Vols. 67 & 68, pp. 3-34.

[HUT01] N. B., Hutcheon (2001), CBD (Canadian Building Digest), 102, Thermal Environment and Human Comfort, [Web-Page], Accessed: Nov 2001.Available at: http://www.nrc.ca/irc/cbd/cbd102e.html.

[STA97] T. Stathopoulos, (1997), Computational Wind Engineering: Past Achievements and Future Challenges, Journal of Wind Engineering and Industrial Aerodynamics, Vols. 67 & 68, pp. 509-532.