Numerical Analysis of Ventilation Flow Through a Three Dimensional Room Fitted with a Two-sided Windcatcher

A.R.Niktash[1] and B.P. Huynh[1]

[1]School of Electrical, Mechanical and Mechatronic Systems

University of Technology, Sydney, NSW 2007, Australia

Abstract

A windcatcher is a structure fitted on the roof of a building for natural ventilation; it extracts the inside stale air to the outside and supplies the outside fresh air into the building interior space working by pressure difference between outside and inside of the building and using ventilation principles of passive stacks and wind tower, respectively.

In this paper, air flow through a three-dimensional and real-sized room fitted with a two-sided windcatcher is investigated numerically, using a commercial computational fluid dynamics (CFD) software package.

The standard K-ε turbulence model is used. Flow pattern and flow velocity are considered in terms of some of the key factors on the performance of a typical windcatcher such as the windcatcher's location, the shape of the windcatcher's bottom, inlet velocity, and the length of the windcatcher's bottom. It is found that the windcatcher's shape at its bottom strongly affects flow pattern and flow velocity when inlet velocity is not too low. This leads to a way of developing the windcatcher's effectiveness in ventilating the living area (lower part) of a room.

Introduction

Renewable energy sources are the proper alternatives for fossil energy sources. Windcatcher is one of the green features for providing natural ventilation using wind power which has been employed over centuries in the hot arid parts of Iran and the other Persian Gulf countries to provide natural ventilation, passive cooling and thermal comfort [2,3,7].

The low cost of windcatcher system in comparison with mechanical ventilation system, being noiseless, durability, requiring no fossil energy, supplying clean air and using sustainable energy of wind power have led to use of the windcatcher as a passive and environmental friendly system.

In the modern design of windcatchers, the two ventilation principles of wind tower and passive stack are combined in one design around a stack that is divided into two halves or four quadrants/segments with the division running the full length of the stack. The cross sections of all windcatchers which have circular or squared shapes are divided internally into various segments to get one-sided, two-sided, three-sided, four-sided, hexahedral, and octahedral windcatchers[1].

The experimental studies of windcatcher systems for all different cases are obviously costly or even impossible. Using computational fluid dynamics (CFD) as a relatively new tool for the assessment of windcatcher systems' performance is very useful and reliable with reasonable accuracy [6,10].

In this paper, effects of a three dimensional two-sided windcatcher's location, the shape of the windcatcher at its bottom, inlet velocity, and the length of the windcatcher's bottom on flow pattern and flow velocity are considered. A numerical study for a two dimensional model has been done by the same authors in their previous work as well [8]. The previous studied model was simplified since the 3rd dimension has been ignored. In the present paper, the real sizes for all three dimensions are considered to reach more realistic results.

Modelling and Computation

A three dimensional room with the length of 5m, the width of 4m, and the height of 3m which has been fitted with a two-sided windcatcher is modelled using CFD-ACE+, a CFD software package from the ESI group.

The 3D models with different windcatcher's locations, different windcatcher's bottom shapes, different inlet velocities, and different windcatcher's bottom lengths are considered.

Figure 1 shows the room fitted with one of the windcatcher's models. The height of windcatcher has been assumed to be 2 m with the inlet and the outlet area of $80 \times 80 \; cm^2$. The length of the windcatcher at its bottom is 10 cm in this model. It is assumed that wind blows from right to left.
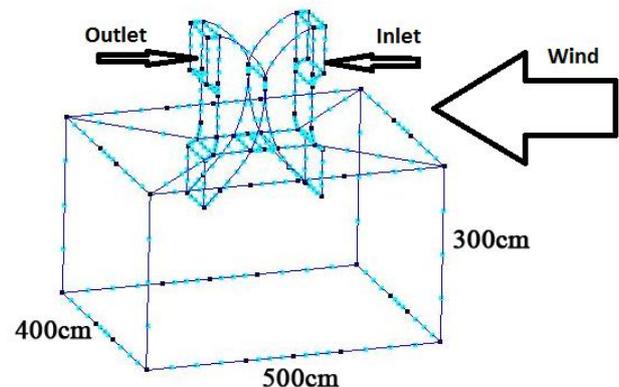

Figure 1. A modelled room fitted with a windcatcher

Unstructured triangle meshes have been used throughout the models to reach better accuracy of CFD simulation (Figure 2). The grid-independence study has been done for different grid numbers in all models to make sure that the grid pattern used is adequate. Consequently, the total number of grids in all models is around 186,000 and the maximum and the minimum grid areas are about 2×10^{-2} $\mathbf{m^2}$ and 1×10^{-4} $\mathbf{m^2}$, respectively.
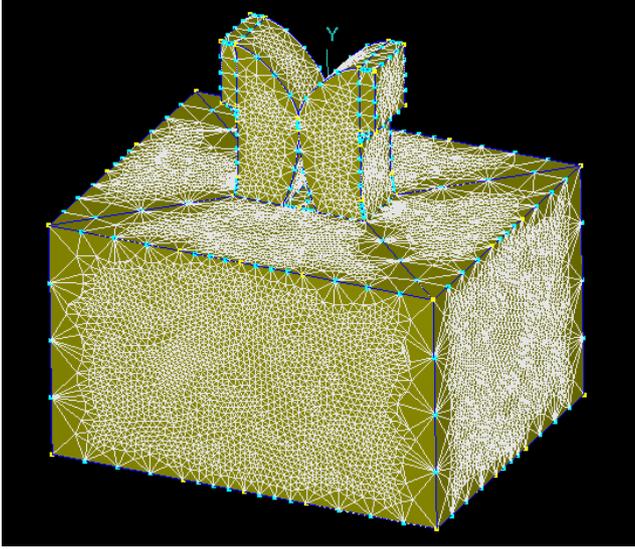

Figure 2. Unstructured triangle meshes in a model

In this work, Reynolds Averaged Navier-Stokes (RANS) simulation method is used and the standard two-equation K-$\mathcal{E}$ turbulence model is employed. The applied governing equations for the turbulent incompressible flow are listed below.

$$\frac{\partial U_j}{\partial x_j} = \mathbf{0} \tag{1}$$

$$U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[v\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + v_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}K\delta_{ij}\right] \tag{2}$$

In the above equations, v_t is the kinematic turbulent or eddy viscosity which is defined as:

$$v_t = {\mu_t}/{\rho} \tag{3}$$

μ_t is the local turbulent viscosity and is defined as follows:

$$\mu_t = \frac{C\mu \ \rho k^2}{\varepsilon} \tag{4}$$

The required additional differential transport equations (turbulent kinetic energy and energy dissipation rate) for the standard K-$\mathcal{E}$ model are as the following:

$$U_j \frac{\partial K}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_t}{\sigma_k}\right)\frac{\partial K}{\partial x_j}\right] + v_t\left[\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}\right] - \varepsilon \tag{5}$$

$$U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1\frac{\varepsilon}{k}v_t\left[\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}\right] - C_2\frac{\varepsilon^2}{k} \tag{6}$$

The equations contain five adjustable constants $\mathbf{C_1}$, C_2 , C_μ , σ_k, and σ_ε. These constants have been derived by comprehensive data fitting for a wide range of turbulent flows [4,5]:

$$\mathbf{C_1 = 1.44}, \mathbf{C_2 = 1.92} \quad , \quad \mathbf{C_\mu = 0.09}, \quad \sigma_k = \mathbf{1},$$
and $\sigma_\varepsilon = \mathbf{1.3}$.

In order to determine the inlet turbulence values, it is necessary to assume a value for the turbulence intensity. For internal flows, the turbulence intensity can be in the range of 1-5 % [9]. The turbulence kinetic energy (K) can be calculated as follows:

$$K = \frac{1}{2}\left(u'^2 + v'^2 + w'^2\right) \tag{7}$$

where u' is the turbulent fluctuation velocity and is equal to the inlet stream velocity multiplied by the turbulence intensity ($u' = U{\times}I$). By assuming u', v', and w' are equal to 2% of the average inlet velocity ($\overline{V}$), the inlet turbulent kinetic energy is calculated as:

$$K = \frac{3}{2}(0.02(\overline{V})\,)^2 \tag{8}$$

The dissipation rate can be determined from the following equation:

$$\varepsilon = \frac{C_\mu^{0.75} K^{1.5}}{\kappa L} \tag{9}$$

Where κ is von Kármán coefficient as 0.4 and L is reasonable length scale (here taken to be the windcatcher's inlet height).

Results and Discussion

Effects of Windcatcher's Location

Four two-canal windcatchers with different windcatcher's positions have been considered: centre (Figure 3-a), right-side (Figure3-b), left-side (Figure3-c), and front-side (back-side) (Figure3-d). Figure 3 shows the flow patterns, corresponding to these four windcatcher's positions with inlet velocity of 3 $\mathbf{m}/\mathbf{s}$.
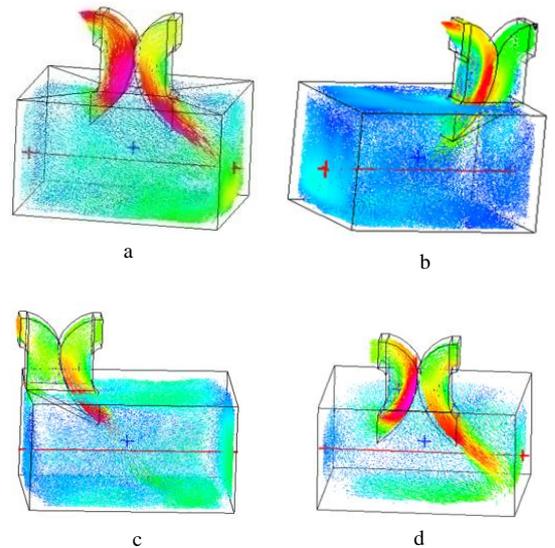

Figure 3. Flow pattern for different windcatcher's positions with inlet velocity of 3 m/s

According to these flow patterns, it is seen that the centred position windcatcher provides the room with the most uniform flow distribution which is the most desirable one.

Figures 4 shows velocity magnitude at level 1.2 m above the floor along the room for different windcatcher's positions, corresponding to Figure 3 above.

a. Centred position windcatcher

b. Right-sided position windcatcher

c. Left-sided position windcatcher

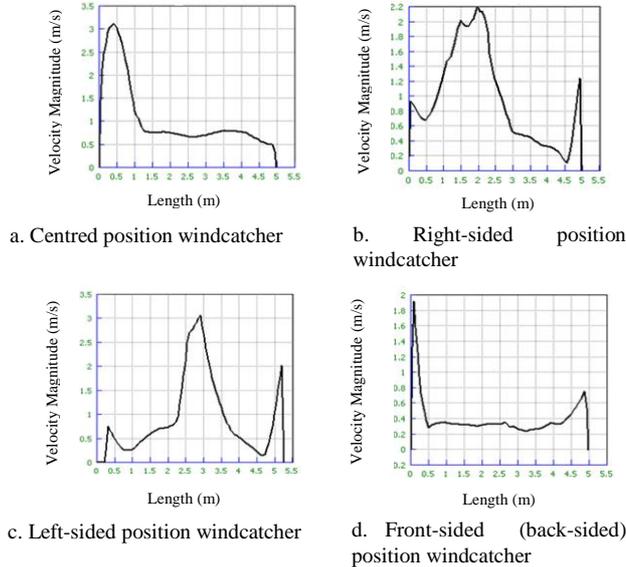d. Front-sided (back-sided) position windcatcher

Figure 4. Velocity magnitude along the room at level 1.2m above the floor for different windcatcher's positions

As it is seen from the above diagrams, the centred position windcatcher (Figure 4-a) provides the most uniform flow and results in the largest region of the room having velocity in the acceptable range for human comfort (0.2 - 1.50 m/s for indoor air speed in hot climates) [11] while in the other graphs, there is significant variation in velocity magnitude at the living area which causes non-uniformity in the fluid flow across the room.

Effects of Windcatcher's Bottom Shape

Effects of windcatcher's shape at its bottom on the flow pattern and flow velocity are considered, using three centred position windcatchers with different bottom shapes: two-rod, flat and two-canal.

Inlet velocity of $3 \, ^m/_s$ has been assumed in all of the models. The results are shown in figure 3-a and figure 5.
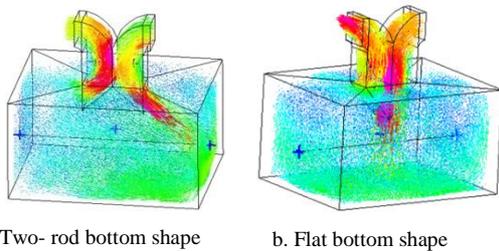

a.Two- rod bottom shape

b. Flat bottom shape

Figure 5. Flow patterns for centred position windcatchers with different bottom shapes

It is seen that flow pattern is strongly affected by the shape of the windcatcher's bottom.

Flat bottom shape (Figure 5-b) results in the most un-even flow distribution in the living area and thus is the least desirable one.

On the other hand, the two-canal bottom shape (Figure 3-a) gives the most desirable (uniform) flow distribution.

Figure 4-a and figure 6 show the velocity magnitude at level 1.2m above the floor along the room's length for different bottom shapes of centred position windcatchers.

The two-canal bottom shape (Figure 4-a) results in the most uniform velocity magnitude at the region and the flat bottom (Figure 6-b) provides the maximum instability in velocity magnitude along the room's length.

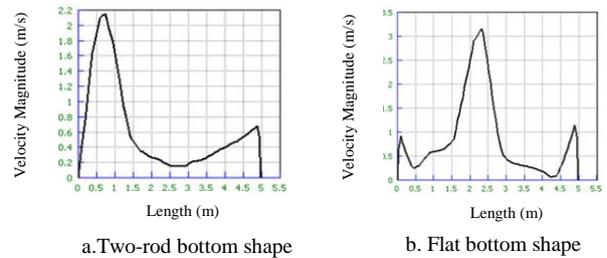a.Two-rod bottom shape

b. Flat bottom shape

Figure 6. Velocity magnitude along the room at level 1.2m above the floor for centred position windcatchers with different bottom shapes

Effects of Windcatcher's Inlet Velocity

The centred position windcatcher has been modelled with various inlet velocities of $0.5 \, ^m/_s$, $1 \, ^m/_s$, $3 \, ^m/_s$, $4.5 \, ^m/_s$ and $6 \, ^m/_s$.

Flow pattern does not vary significantly and is similar to figure 3-a for the various inlet velocities.

Figure 4-a and figure 7 show velocity magnitude at 1.2m above the floor along the room's length for different inlet velocities.

a. 0.5 m/s
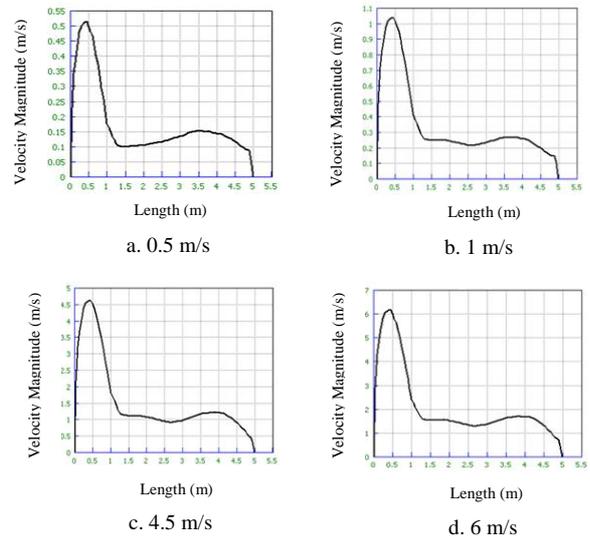
b. 1 m/s

c. 4.5 m/s

d. 6 m/s

Figure 7. Velocity magnitude along the room at level 1.2m above the floor for centred position two-canal windcatcher with different inlet velocities

It is seen that flow velocity is increasing proportionally to the windcatcher's inlet velocity increment.

Effects of Windcatcher's Bottom Length

Effects of the windcatcher's bottom length (distance between the ceiling and the top edge of windcatcher's bottom in the room) on flow pattern and velocity has been considered for the two-canal centred-positioned configuration. Flow patterns are shown in figure 3-a (bottom length 10 cm), figure 8-a (20 cm), and figure 8-b (40 cm); inlet velocity is 3 $\frac{m}{s}$ in all cases.
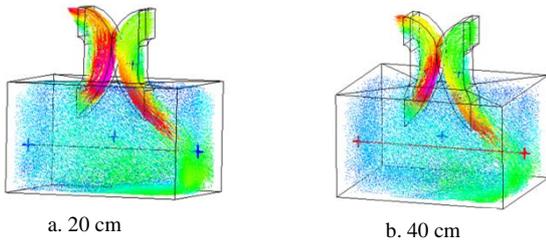

a. 20 cm   b. 40 cm

Figure 8. Flow patterns for different bottom lengths of two- canal centred position windcatchers

From the above figures, the bottom length of 10cm (Figure 3-a) seems to offer the most uniform flow distribution in the room while it is well above the floor without obstructing the access way through the room.

Figure 4-a and figure 9 show the velocity magnitude at 1.2m above the floor along the room's length for different bottom lengths.
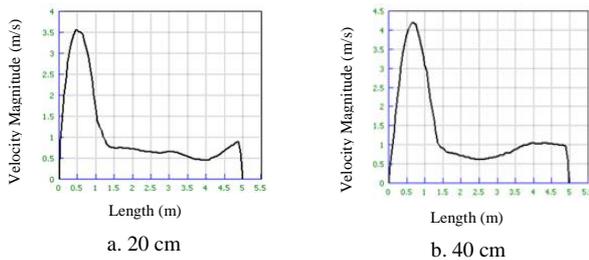

a. 20 cm   b. 40 cm

Figure 9. Velocity magnitude for different bottom lengths velocity of two- canal centred position windcatchers

From the above figure, it is seen that the length of windcatcher's bottom affects flow velocity significantly.
The bottom length of 10cm provides the most stability in velocity magnitude along the room's length which causes the most uniform flow velocity while for the bottom lengths of 20cm and 40cm, the variations in the velocity magnitude are noticeable.

Conclusions

Effects of factors such as windcatcher's location, windcatcher's bottom shape, inlet velocity, and windcatcher's bottom length on flow pattern and velocity have been considered through a three dimensional room fitted with a two-sided windcatcher by using

the standard $\text{K-}\varepsilon$ turbulence model.
It has been found that with the typical inlet velocity of 3 $\frac{m}{s}$ the centred position windcatcher provides the most uniform flow distribution and the most stable flow velocity along the room in comparison with the other windcatcher's positions.

The shape of windcatcher's bottom strongly affects flow pattern and flow velocity; the two-canal bottom shape produces more desirable (uniform) flow distribution than the other bottom shapes. It results in the most uniform velocity magnitude at the region as well.

Flow pattern does not vary significantly with inlet velocity while flow velocity increases proportionally with the windcatcher's inlet velocity.

Changing the length of windcatcher's bottom has significant effects on both flow pattern and velocity in the living area; the bottom length of 10cm seems to offer the most uniform flow distribution and the most stability in velocity magnitude along the room's length.

References

[1] Bahadori, M.N., Viability of Wind Towers in Achieving Summer Comfort in the Hot Arid Regions of the Middle-East, *Renewable Energy*, **5**, 1994, 879-89.

[2] Fathy, H., *Natural energy and vernacular architecture: principles and examples with reference to hot arid climates*, The university of Chicago press, 1986.

[3] Karakatsanis, C., Bahadori, MN. &Vickery, BJ., Evaluation of pressure coefficients and estimation of air flow rate in buildings employing wind tower, *Solar Energy*, **37**, 1986 363-374.

[4] Kivva, T., Huynh, B.P., Gaston, M. & Munn, D., A numerical study of ventilation flow through a 3-dimensional room with a fan, *Turbulence, Heat and Mass Transfer 6*, **10**, 2009, 1615-1625.

[5] Launder, B.E. & Spalding, D.B., The Numerical Computation of Turbulent Flows, *Comp. Meth. Appl. Mech. Eng.*, **3**, 1974, 269-289.

[6] Liu, L. & Mak, C.M., The assesment of the performance of a windcatcher system using computational fluid dynamics, *Building and Environment*, **42**, 2007, 1135-1141.

[7] Montazeri, H., Montazeri, F., Azizian, R. & Mostafavi, S., Two-sided windcatcher performance evaluation using experimental, numerical and analytical modeling, *J.Renewable Energy*, **35**, 2010, 1424-1435.

[8] Niktash, A.R. & Hyunh, B.P., Numerical Study of Ventilation Flow Through a Two Dimensional Room Fitted With a Windcatcher, in *CFD Applications for Optimization and Controls: IMECE 2011,* ASME, 2011.

[9] Tu, J., Yeoh, G.H., & Liu, C., *Computational Fluid Dynamics: A Practical Approach*, Butterworth-Heinemann, 2008.

[10] White, F.M., *Fluid Mechanics*, McGraw Hill, 2011.

[11] Auliciems, A & Szokolay, S.V., Thermal Comfort, *Passive and Low Energy Architecture International*, Department of Architecture of The University of Queensland, 2007.