19th Australasian Fluid Mechanics Conference
Melbourne, Australia
8-11 December 2014

CFD Simulation and Analysis of a Two-sided Windcatcher's Inlet\Outlet Geometric Shape Effect in Ventilation Flow Through a Three Dimensional Room

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 to provide ventilation for the interior space employing wind power; it exhausts 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, the effect of different geometric shapes of inlet/outlet for a two-sided windcatcher on the flow velocity, flow pattern , and flow rate which affect on the ventilation quality through a three-dimensional and typical room fitted with a two-sided windcatcher is observed numerically, applying a commercial computational fluid dynamics (CFD) software package.

The standard RANS K-ε CFD method is used in all simulations for three different geometric shapes namely square shape, rectangular shape and circular shape while they have the same area.

It is found that the geometric shape of windcatcher's inlet\outlet strongly affects flow pattern, flow rate and flow velocity specially in the living area of the room and there is a relationship between the length/width ratio of the rectangular shape and the ventilation quality.

Introduction

Applying renewable energy sources is the proper solution for preventing environmental pollution. Wind power is one of the clean energies employed in windcatcher systems for providing natural ventilation; windcatcher as a green feature has been used over centuries in the hot arid regions, particularly in Iran and the other Persian Gulf countries in one hand and north of Africa region such as Algeria, Egypt and other north African countries in another hand to provide natural ventilation, passive cooling and thermal comfort [1,2,3].

Ease of operation and maintenance, low cost , durability and being noiseless of windcatcher system in comparison with electro-mechanical ventilation systems, 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.

The experimental studies of windcatcher systems for all different cases are obviously expensive or even impossible. Using computational fluid dynamics (CFD) for the windcatcher performance assessment has become a creditable tool for flow analysis in the buildings [6,10]. According to the previous numerical studies and simulations which have been done by the same authors using the standard RANS K-ε and LES methods, effects of a 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 have been considered for two dimensional and three dimensional models [7 ,8].

In this paper, effect of three different shapes of a two-sided windcatcher's inlet/outlet including square shape, rectangular shape and circular shape but the same areas on flow velocity, flow pattern and flow rate of the windcatcher are considered.

Modelling and Computation

A three dimensional room with the size of $5\times4\times3$ m^3 has been fitted with a two-sided windcatcher, with different geometric shapes of inlet\outlet surrounded by large space, is simulated using CFD-ACE+, a CFD software package from the ESI group. Figure 1 shows the model with square shape of inlet/outlet without its surrounded space.

The height of top part of windcatcher (above the roof) from the roof's surface has been assumed to be 2 m and its height at the bottom (under the roof to the beginning of the windcatcher's opening) is assumed 10 cm in all models. It is assumed that wind blows from right to left and perpendicular to the windcatcher's inlet area.

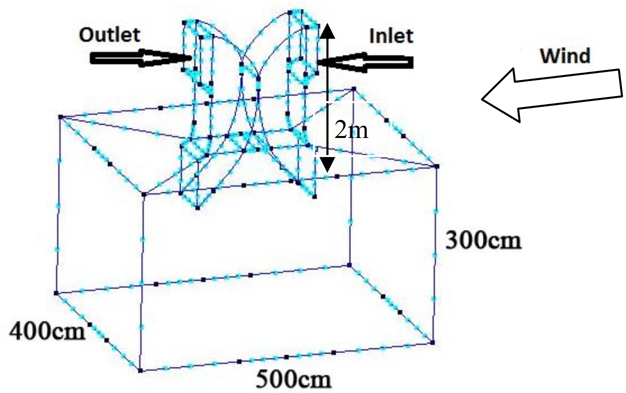

Figure 1. A modelled room fitted with a windcatcher for square shape of inlet/outlet

Figure 2 shows three different shapes of windcatcher's inlet\outlet. In figure 2, types A and B belong to circular and square shapes and types C and D allocate to rectangular shape in horizontal and vertical positions. The inlet/outlet projected area for all models is the same and 0.64 m^2.

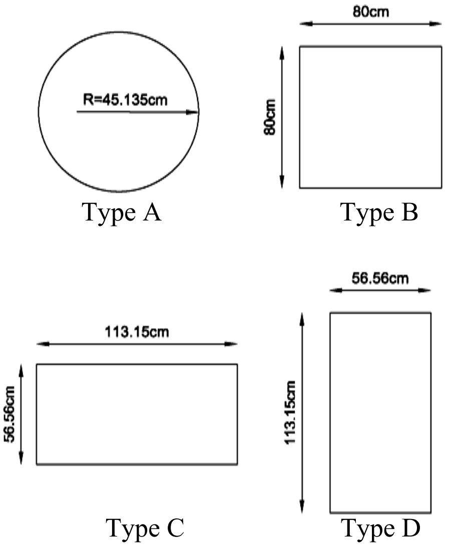

Figure 2. Different geometric shapes of windcatcher's inlet/outlet

Figure 3 shows the complete system for simulation including model with type A of inlet\outlet surrounded by large space for simulating the natural condition wherein wind blows from right to left with the speed of 3 m/s.

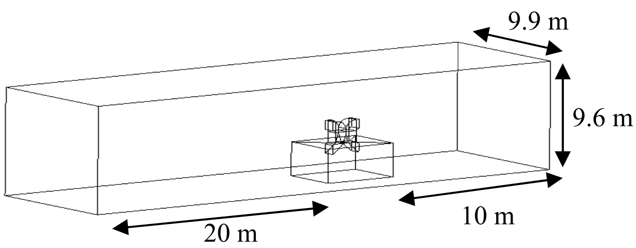

Figure 3. A complete system for simulation including a model and its large surrounded space

Unstructured triangle meshes have been used throughout the models and the surrounding space to reach better accuracy of CFD simulation (Figure 4).

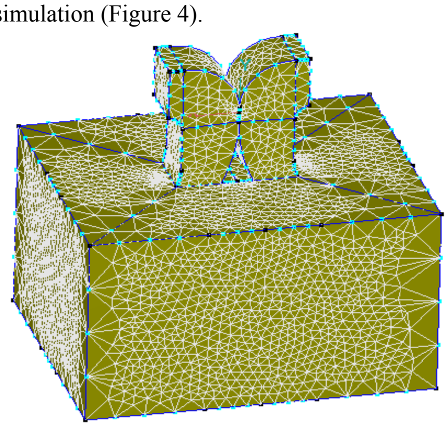

Figure 4. Unstructured triangle meshes in a room model (square inlet/outlet)

To save on computational efforts and reduce accumulated errors, mesh distribution is less dense in the expected near stagnant flow regions like the room corners.

On the other hand, dense mesh has been used in the living area which is far from the corners of the room (Figure 4).

A grid-independence study has been done for different grid numbers for all models to make sure that the grid pattern used is adequate. Consequently, the total number of grid cells in all models is around 135,000 and the maximum and the minimum grid areas are about 1.4 $\times$ $10^{-3}\,\mathrm{m}^2$ and 1.4 $\times 10^{-7}$ m^2, respectively.

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

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

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

The required additional equations for the standard K-Ɛ model are as the following:

$$U_j \frac{\partial K}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_t}{\sigma_k}\right)\frac{\partial K}{\partial x_j}\right] + v_t\left[\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}\right] - \varepsilon \tag{3}$$

$$U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1\frac{\varepsilon}{k}v_t\left[\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}\right] - C_2\frac{\varepsilon^2}{k} \tag{4}$$

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

$$v_t = {}^{\mu_t}/_{\rho} \tag{5}$$

μ_t is the local turbulent viscosity and is defined as follows [5]:

$$\mu_t = \frac{C\mu\ \rho k^2}{\varepsilon} \tag{6}$$

The equations contain five adjustable constants 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]; their values are as bellows:

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

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:

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

where u' is the turbulent fluctuation velocity and is equal to the inlet stream velocity multiplied by the turbulence intensity (u'= U×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:

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

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

The Model With Circular Windcatcher's Inlet/Outlet

In this model, the area of circular inlet\outlet is $0.64 cm^2$ (type A in Figure 2). The velocity magnitude at level 1.2 m above the floor along the room's central plane has been shown in Figure 5.

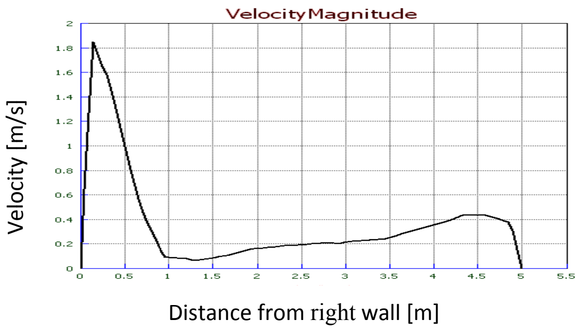

Figure 5. Velocity Magnitude at level 1.2 m for the model with circular windcatcher's inlet\outlet

According to the above graph, the velocity magnitude at this level is only stable in the range of 0.18~0.2 m/s at the distance of 2 m to 2.9 m from the right wall which is 18% of the total room's length. Figure 6 shows some traces of the flow path in this model.

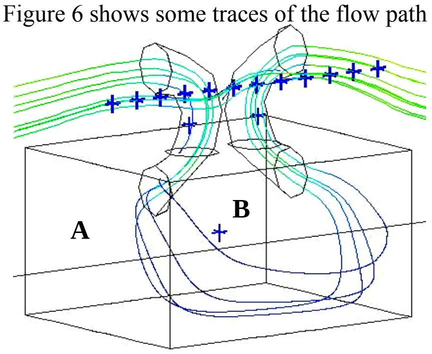

Figure 6. Some traces of flow path for the model with circular windcatcher's inlet\outlet

Based on the above figure , there is somehow full ventilation in part of the living area but there are some stagnation regions in the middle and left side of the room (indicated as "A" and "B").

The Model With Square Windcatcher's Inlet/Outlet

In this model (type B in Figure 2), the windcatcher's inlet\outlet size is 80×80 cm^2. Velocity magnitude at level 1.2 m above the floor across the room's central plane is shown in Figure 7.

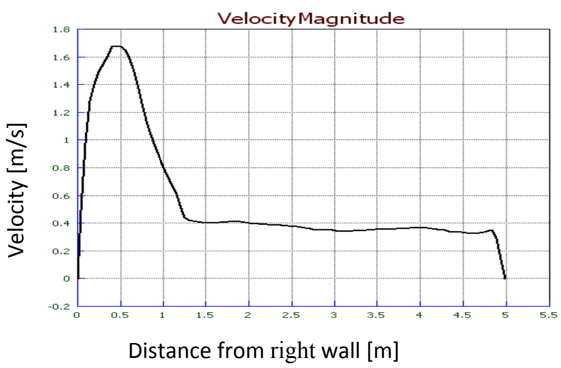

Figure 7. Velocity Magnitude at level 1.2 m for the model with square windcatcher's inlet\outlet

It is seen that the velocity magnitude at this level is stable in the range of 0.35~0.37 m/s at the distance of 2.1 m to 4.35 m from the right wall of the room which is 45% of the room's length.

The flow traces for this model is shown in Figure 8.

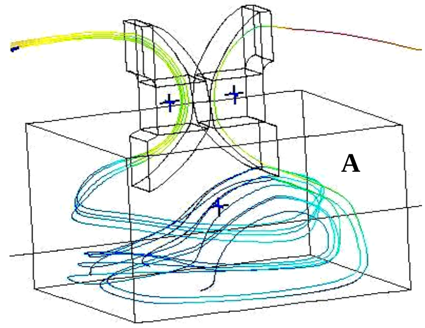

Figure 8. Some traces of flow path for the model with square windcatcher's inlet\outlet

As it is seen from the above flow path, there is full ventilation inside the room specially in the living area and there is a small stagnation region at the top corner of the room in the right side (indicated as "A").

The Model With horizontal Rectangular Windcatcher's Inlet/Outlet

The horizontal rectangle shape of windcatcher's inlet/outlet which has been shown as type C in Figure 2 has the same area of other types (0.64 m^2). The velocity magnitude for this model at level 1.2 m above the floor across the room's central plane has been shown in Figure 9.

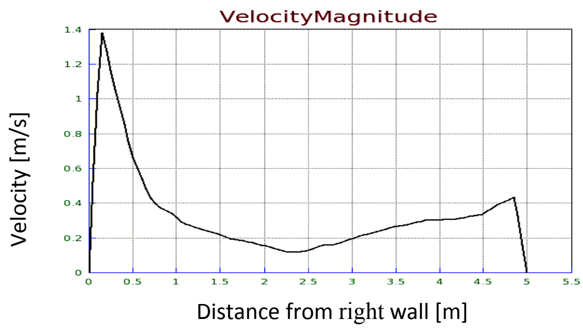

Figure 9. Velocity Magnitude at level 1.2 m for the model with horizontal rectangular windcatcher's inlet\outlet

It is seen that the velocity magnitude at this level is approximately stable in the range of 0.32~0.34 m/s at the distance of 3.8 m to 4.5 m from the right wall which is 14% of the total room's length. The model's flow traces is shown in Figure 10.

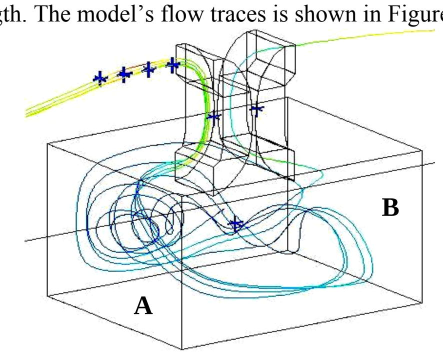

Figure 10. Some traces of flow path for the model with horizontal rectangular windcatcher's inlet\outlet

it is seen that there is almost full ventilation in the living area of the room while there some stagnation regions in two sides of the room (indicated with “A” and “B).

The Model With vertical Rectangular Windcatcher’s Inlet/Outlet

This model has been shown in Figure 2 as type D. It has the same dimension of type C but different orientation. Figure 11 shows velocity magnitude at level 1.2 m above the floor along the room’s central plane.

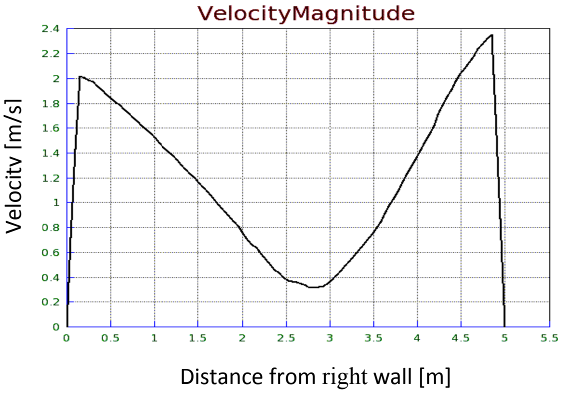

Figure 11. Velocity Magnitude at level 1.2 m for the model with vertical rectangular windcatcher's inlet\outlet

It is evident that the velocity magnitude at this level is completely unstable across the room. The flow traces for this model is shown in Figure 12.

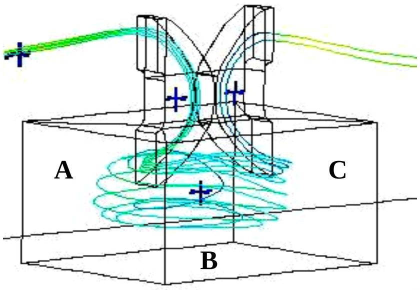

Figure 12. Some traces of flow path for the model with vertical rectangular windcatcher’s inlet\outlet

As it is seen from the above flow path, the circulation is not uniform and complete in the room and there are some stagnation regions in the living area without any circulation (indicated as “A” , “B” and “C”). Table 1 shows the flowrates for the studied models. It is seen that the maximum flowrate belongs to horizontal rectangular model (type C) and the minimum flowrate allocates to vertical rectangular model (type D).

Model Type	Flowrate (m^3/s)
A	0.804
B	1.119
C	1.179
D	0.667

Table 1. Flowrates for the models

Conclusions

The geometric shape of windcatcher’s inlet\outlet strongly affects flow pattern, flow velocity, and flow rate especially in the living area of the room.

Based on the velocity magnitude graphs at the living area (level 1.2 m above the floor) for the studied two-sided windcatcher with different geometric inlet/outlet shapes, the square windcatcher’s inlet/outlet model provides the stable velocity magnitude for the maximum length of the room (45%) and this model provides the near-maximum flowrate after the horizontal rectangular model while the most unstable velocity magnitude in the living area belongs to the vertical rectangular windcatcher’s inlet/outlet model which has the minimum flowrate as well. On the other hand, the best circulation in the studied models happens for square one while the worst is for the vertical rectangular one. Moreover, it is seen that length/width ratio for the rectangular windcatcher’s inlet/outlet models plays very important role in their ventilations quality; such as by increasing this ratio from 0.5 to 1 (square shape), stable velocity magnitude, full ventilation and flowrate will increase significantly while by increasing the ratio from 1 to 2 , increase in flowrate and instability in velocity magnitude happen.

Consequently, the square model satisfies two important human comfort factors for having proper indoor ventilation (stable velocity magnitude and full circulation across the room). It leads to a development in the performance of windcatcher systems.

References

[1] Bouchahm, Y., Bourbia, F. & Belhamri, A., Performance analysis and improvement of the use of wind tower in hot dry climate, *Renewable Energy*, **36**, 2011, 898-906.

[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] Niktash, A.R. & Huynh, B.P. , Numerical Analysis of Ventilation Flow Through a Three Dimensional Room Fitted with a Two-sided Windcatcher, AFMC, 2012.

[8] Niktash, A.R. & Huynh, B.P., Simulation and Analysis of Ventilation Flow Through a Room Caused by a Two-Sided Windcatcher Using a LES Method, ICME2014, IAEng, 2014.

[9] Tu, J., Yeoh, G.H., & Liu, C., *Computational Fluid Dynamics: A Practical Approach*, Butterworth-Heinemann, 2008, 263-265.

[10] White, F.M., Fluid Mechanics, McGraw Hill, 2011, 590-593.