International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:6, No:12, 2012

CFD Analysis of Natural Ventilation Behaviour in Four Sided Wind Catcher

M. Hossein Ghadiri, Mohd Farid Mohamed, and N. Lukman N. Ibrahim

Abstract**—**Wind catchers are traditional natural ventilation systems attached to buildings in order to ventilate the indoor air. The most common type of wind catcher is four sided one which is capable to catch wind in all directions. CFD simulation is the perfect way to evaluate the wind catcher performance. The accuracy of CFD results is the issue of concern, so sensitivity analyses is crucial to find out the effect of different settings of CFD on results. This paper presents a series of 3D steady RANS simulations for a generic isolated four-sided wind catcher attached to a room subjected to wind direction ranging from 0º to 180º with an interval of 45º. The CFD simulations are validated with detailed wind tunnel experiments. The influence of an extensive range of computational parameters is explored in this paper, including the resolution of the computational grid, the size of the computational domain and the turbulence model. This study found that CFD simulation is a reliable method for wind catcher study, but it is less accurate in prediction of models with non perpendicular wind directions.

Keywords**—**Wind catcher, CFD, natural ventilation, sensitivity study.

I. INTRODUCTION

REDUCTION of natural resources, global warming and rising fossil-fuel price is a real concern to human beings [1]. Buildings use more than 40% of the total world's energy consumption [2] while more than 60% of that is used for cooling, heating and ventilation systems [3]. Therefore such mechanical instruments shall be replaced with green architecture features such as wind catcher, solar chimney; light well as well as atria in order to consume less energy [3-5].Wind tower or wind catcher is one of these devices, neglected in modern buildings [6]. It is a traditional architectural device that is raised on the building's roof in order to replace stuffy air with fresh air [7]. They comes in variety of plan forms [8] and with different heights ranged between 5 m to 34 m [9]. The channel of these conventional wind catchers is commonly divided into two, four, six and eight parts to supply or extract air by means of ventilation.

Different methods are employed in order to investigate wind catcher's performance. One of them is CFD-based programs which have more advantages compare to others and offer a comprehensive report of the air flow. A CFD model is used by Elmualim and Awabi [10] and Elmualim [11] utilize CFD simulation to validate laboratory measurements of Cp (pressure coefficient) in windward and leeward quadrant of a square wind catcher at normal incidence, and good agreement (1% error) in windward and less successful result (77% error) in leeward side is achieved . Li and Mak [12] compare their CFD simulation results with wind tunnel measurements of Elmualim et al. [13] in a square wind catcher with length of 500mm and good agreement was achieved. A circular wind catcher performance is evaluated by Su Riffat [14] using CFD model demonstrating that this type of wind catcher is able to increase net flow rate by 4 times. Hughes et al. [15] used CFD to model a 1000mm Wind catcher in order to predict net flow rate. A comparison is made between wind tunnel experimental result of Elmualim [13] and Awabi [16], and the results is obtained with 20% error.

Liu Mak [5] modeled a wind catcher system with different numbers of louvers and louver's length to examine the performance of the device utilizing CFD. The performance of circular wind catcher with different number of openings are evaluated by Montazeri [17] using different methods including wind tunnel experiment, smoke visualization test and CFD simulation. Montazeri et al. [18] examine the effect of pressure coefficient on ventilation rate in a two-sided rectangular wind catcher by CFD validating with wind tunnel experiment.

Although in recent years, CFD application is widely use in analyzing air flow behaviour due to improvements in turbulence modeling [19], the accuracy of CFD is still doubtful. Likewise, detailed sensitivity analyses, indicating the effect of different setting of CFD on results, is crucial in order to present guidelines in this field. Best practice guidelines [20-24] have been reported for CFD simulation of wind flow around buildings in general. While these guidelines are not presented for wind catchers, detailed sensitivity analyses is essential. This paper presents CFD simulation of rectangular wind catcher attached to a vernacular house in hot dry region of Iran. A comparison is made between CFD results and wind tunnel experiments accomplished by Mahyari [25] by means of validation. The effect of mesh design, domain size and turbulence models on velocity coefficient value are assessed in this paper.

II. EXPLANATION OF THE EXPERIMENTS

Detailed wind measurements of wind-induced cross-ventilation for isolated four-sided wind catcher models were conducted by Mahyari [25]. The experimental investigation was accomplished in wind tunnel at the Sydney University. It

---

Maryam. Hossein Ghadiri, is with Universiti Kebangsaan Malaysia (UKM), (phone: +60172984805; e-mail: maryam200ghadiri@yahoo.com).

Mohd Farid Mohamed is with Universiti Kebangsaan Malaysia (UKM), (e-mail: farid0906@gmail.com).

N. Lukman N. Ibrahim is with Universiti Kebangsaan Malaysia (UKM), (email:n.lukman@gmail.com).

is a low speed open circuit boundary layer wind tunnel for environmental studies. It has a uniform rectangular cross sectional area equipped with a turn table in the middle of the test section. The effective dimension of the wind tunnel is 1.80m wide by 0.92m high by 8m long. Almost 5m long is devoted to the boundary layer growth section and 3m to the test section illustrated in Fig. 1. The net cross section area therefore is 1.6560 m2.

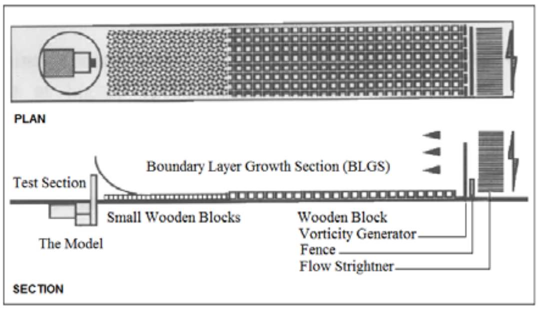

Fig. 1 Wind tunnel plan and cross section

A maximum wind tunnel blockage of 2.17% is produced by scaled model of wind catcher (1:20) based on the dimension of the wind tunnel.

A series of measurements took place in the wind catcher's tower. To determine velocity and velocity coefficient in each individual shaft, four probes of anemometers were used in the centre of each shaft vertically, one meter above the roof plane, and one at the reference height 0.4m above the terrain (8m in real scale). All measurements were conducted at speed of 5 m/s.

A. Velocity Coefficient Measurement

The comparison between Mahyari's wind tunnel experiment and CFD result of this study requires both values be in the same mode. The velocity coefficient which is a dimensionless coefficient is defined as the ratio of the value of velocity at the point of interest to the value at the reference height as in (1).

$$C_{v_i} = V_i / V_{ref} \qquad (1)$$

where V_i is the mean or peak wind speed at the point of interest.

V_{ref} is the mean or peak wind speed at the reference height.

B. The Wind Catcher Model

Four-sided wind catcher's plan is divided into four separated channels by several vertical blades in order to ventilate indoor air. Rectangular wind catcher with plus blade form is chosen among Mahyari's investigated samples. As shown in Fig. 2, width, length, tower height and shelf height of the wind catcher is 1m, 1.5m, 2m and 8m respectively. The wind catcher is connected to a typical house of Yazd consisted of a 4×5 meter "Iwan" (ground floor) and the same size basement room, located on the short size of an enclosed 6×10 meter courtyard. The height of the rooms was 3m, and the height of the courtyard was 4m.

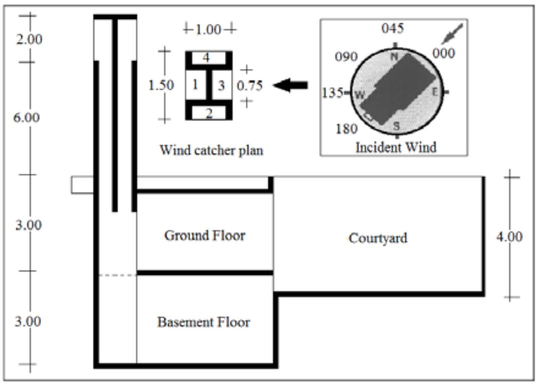

Fig. 2 Side and top view of four-sided wind catcher model for simulations with indication of wind incident angle. (All dimensions are in meter)

III. CFD Simulations; Reference Case

Reference case is illustrated in Fig. 3 and CFD settings as well as the results of the case are presented in this part.

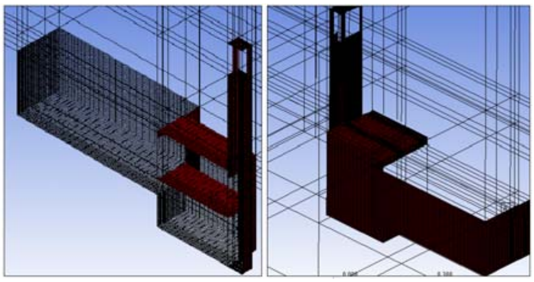

Fig. 3 Side views of four-sided wind catcher model

A. Computational Domain and Grid

The constructed geometry around the model in order to develop fluid is called computational domain [26]. Best practice guidelines of Frank [22] and Tominaga [24], is used to define the domain dimension except for the upstream length, which was set at 400mm in reduced scale (the value of wind catcher height). Therefore the dimension of width, length and the height of the domain are set at 1.8, 5.2 and 0.92 m^3 in reduced scale which corresponds to 36, 104 and 18.4 m^3 in full scale respectively. The computational grid is fully structured and has 2,074,642 hexahedral cells for wind incident angle of 0º, 90º and 180º illustrated in Fig. 4 and tetrahedral mesh design is used for the cases with wind direction of 45ºand 135º. The grid resolution resulted from a grid-sensitivity analysis that will be outlined in Section A.

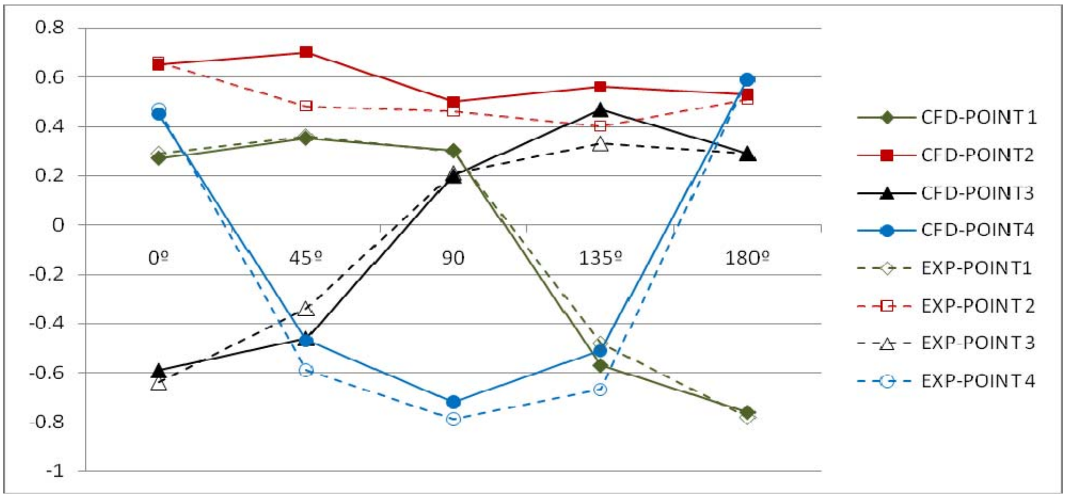

Fig. 5 Velocity coefficient through the windward and leeward sides for different wind incident angles

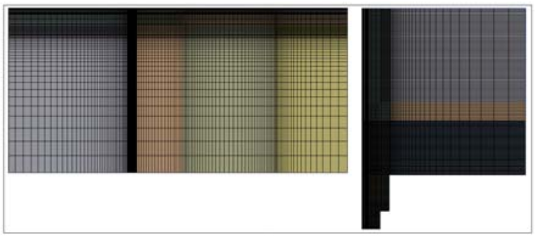

Fig. 4 Side and top view of mesh design of wind catcher model with hexahedral mesh

B. Boundary Conditions

To develop a theoretical velocity profile the power law equation presented in (2) with the recommended values suitable for this study is applied according to wind speed in wind tunnel experiment condition.

$$V_h / V_H = (h / H)^{\beta} \tag{2}$$

where V_h and V_H are the mean wind velocity at heights h and H, respectively, and β is the power law index, β =0.28, which is recommended value for suburban area.

Zero static pressure is applied at the outlet plane and symmetry conditions, i.e. zero normal velocity and zero normal gradients of all variables, at the top and lateral sides of the domain.

C. Solver Settings

The simulations are carried out with the commercial CFD code Ansys CFX-14. The 3D steady RANS equations are solved with k-ε turbulence model. Convergence criteria recommended by Tominaga [24] is used which is leveling off the scaled residuals to 10^{-5}. Turbulence intensity value is set at 5% and second order discretization scheme is used.

D. Results and Comparison with Wind Tunnel Experiments

The results of the reference case at different wind directions were compared with the wind tunnel experiments conducted by Mahyari [25]. Fig. 5 and Table I compare the CFD-calculated and measured velocity coefficient through point 1, 2, 3 and 4 in various prevailing wind directions (α). Generally in the comparison to the velocity coefficient, a satisfactory agreement between the CFD results and measurements was achieved. The readings matched well in windward and leeward sides when wind below perpendicular to the wind catcher facade at $\alpha=0^\circ$, 90º and 180º. There is a little variation in CFD result compare to experimental measurements when the wind angle is 45º and 135º. It is tends to over-prediction at point 2 and point 4 and under-prediction at point 1 and 3.

TABLE I
MEASURED VELOCITY COEFFICIENT THROUGH WIND CATCHER SYSTEM FOR DIFFERENT WIND INCIDENT

Case		Point 1	Point 2	Point 3	Point 4
0º	CFD	0.27	0.65	-0.59	0.45
	EXP.	**0.29**	**0.66**	**-0.64**	**0.47**
45º	CFD	0.35	0.70	-0.46	-0.47
	EXP.	**0.36**	**0.48**	**-0.34**	**-0.59**
90	CFD	0.30	0.50	0.20	-0.72
	EXP.	**0.30**	**0.46**	**0.21**	**-0.79**
135º	CFD	-0.57	0.56	0.47	-0.51
	EXP.	**-0.48**	**0.40**	**0.33**	**-0.67**
180º	CFD	-0.76	0.53	0.29	0.59
	EXP.	**-0.78**	**0.51**	**0.29**	**0.59**

IV. CFD SIMULATIONS; SENSITIVITY ANALYSIS

A sensitivity analysis was carried out with different CFD settings and comparing to the reference case and later assessing the effect of these parameters on the simulation results. The parameters tested are the size of the computational domain (Section A), the resolution of the computational grid (Section B) and the turbulence model (Section C). Table II provides a summary of the computational parameters for the sensitivity analysis with indication for the reference case.

TABLE II
OVERVIEW OF COMPUTATIONAL PARAMETERS FOR SENSITIVITY ANALYSIS

	domain size	grid resolution	Turbulence model
Ref. case	**D=1H**	**Grid B (2,074,642 cells)**	**k- ε**
	D=2H	Grid A (4,673,912 cells)	RNG k- ε
	D=3H	Grid C (684,721 cells)	SST
			k- ω
			SSG-RSM

A. Impact of Computational Grid Resolution

In addition to the reference grid (grid B) with 2,074,642 cells, one finer and one coarser grid were constructed, consisting of 4,673,912 (Grid A) and 684,721 cells (grid C). Upstream dimension was fixed at 400mm and k-ε turbulence model was set to solve all cases. The results of all three grids are shown in Table III and Fig. 6. The analysis showed that the reference grid that was used for the reference case was an appropriate grid. The result in the velocity coefficient through wind catcher channels were almost same when grid A and B is applied and less accurate result was achieved when coarser mesh was set. Grid B was chosen due to less memory effort for meshing process.

TABLE III
MEASURED VELOCITY COEFFICIENT THROUGH WIND CATCHER SYSTEM FOR DIFFERENT MESH DESIGNS

Case	Point 1	Point 2	Point 3	Point 4	No. of cells
Exp.	0.29	0.66	-0.64	0.47	-
Grid A	0.27	0.65	-0.61	0.46	4,673,912
Grid B	**0.27**	**0.65**	**-0.59**	**0.45**	**2,074,642**
Grid C	0.19	0.52	-0.47	0.38	684,721

B. Impact of Size of Computational Domain

Width and length of the computational domain was the same as the wind tunnel dimension in order to have same blockage ratio in both cases which is 1.8m and 0.92m respectively. The downstream length of the domain is set at 10H, (4000mm) with H (400mm in reduced scale) the wind catcher height. The upstream length of the domain (D) consisted of H, 2H and 3H. Turbulence model for all cases is k-ε with grid B and wind direction of 0º. Through Table VI, the difference between experimental and numerical value of velocity coefficient is decrease in windward side (point 3), point 2 and 4 and increased in leeward side (point 1) by increasing the upstream length. Therefore the lowest rate of error is for upstream length of H which is 6.5%, 1.5%, 7.5% and 4.5% through point 1 to 4 respectively.

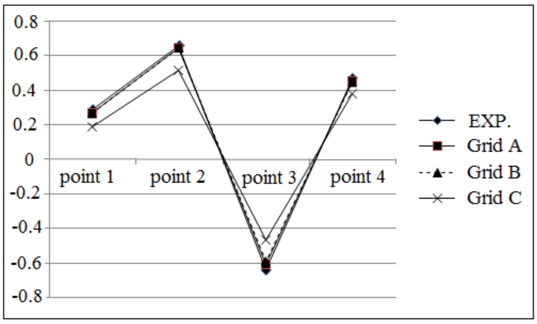

Fig. 6 Velocity coefficient through the windward and leeward sides of the model with different grid mesh designs

TABLE VI
MEASURED VELOCITY COEFFICIENT THROUGH WIND CATCHER SYSTEM FOR DIFFERENT UPSTREAM DIMENSION

Case	Point 1	Point 2	Point 3	Point 4
Exp.	0.29	0.66	-0.64	0.47
1H	**0.27**	**0.65**	**-0.59**	**0.45**
2H	0.28	0.64	-0.53	0.44
3H	0.29	0.63	-0.48	0.41

C. Impact of Turbulence Model

CFD simulations are arranged with various turbulence models:

- Standard k-ε model (k-ε)
- Renormalization Group k-ε model (RNG k-ε)
- Shear-stress transport k-ε model (SST k-ε)
- SSG Reynolds Stress Model (RSM)
- k-ω model

The effects of the turbulence models on the velocity coefficient through point1, 2, 3 and 4 are reported in Table V. All cases have medium mesh design (grid B) with upstream length of H. As shown in Table V; SSG-RSM turbulence model is not suitable for such complicated models and solution didn't converged and the result of the case with standard k- ε and RNG k- ε turbulence models has good agreement with experimental result. Convergence was obtained after 18 hours in model with RNG k- ε turbulence model which is really time-consuming, so k- ε turbulence model was chosen taking just 2 hours to converge.

TABLE V
MEASURED VELOCITY COEFFICIENT THROUGH WIND CATCHER SYSTEM FOR VARIOUS TURBULENCE MODELS

Case	Point 1	Point 2	Point 3	Point 4
Exp.	0.29	0.66	-0.64	0.47
k- ε	0.27	0.65	-0.59	0.45
RNG k- ε	0.25	0.63	-0.61	0.51
SST	0.13	0.77	-0.60	065
SSG-RSM		Not converged		
k- ω	0.24	0.65	-0.58	0.45

V. SUMMARY AND CONCLUSION

CFD simulation is the perfect way to evaluate the wind catcher performance. The accuracy of CFD results is the issue of concern, so sensitivity analyses is crucial to find out the effect of different settings of CFD on results. In this paper the CFD simulations were validated with detailed wind tunnel experiments. The influence of an extensive range of computational parameters was explored in this paper, including the resolution of the computational grid, the size of the computational domain and the turbulence model. This study found that satisfactory results were achieved in the case with computational grid of 2,074,642 cells solving by k-ε turbulence model. In the comparison to the velocity coefficient in the cases with different wind incident angles, a satisfactory agreement between the CFD results and measurements was achieved in wind angles of 0º, 90º and 180º and there was a little variation in wind angles of 45º and 135º. Therefore, it could be concluded that CFD is less accurate in prediction of models with non perpendicular wind directions.

REFERENCES

[1] Mahlia, T., et al., "*A review on fuel economy standard for motor vehicles with the implementation possibilities in Malaysia*". Renewable and Sustainable Energy Reviews, 2010. 14(9): p. 3092-3099.

[2] Masoso, O. and L. Grobler, "*The dark side of occupants' behaviour on building energy use*". Energy and buildings, 2010. 42(2): p. 173-177.

[3] Chan, H.Y., S.B. Riffat, and J. Zhu, "*Review of passive solar heating and cooling technologies*". Renewable and Sustainable Energy Reviews, 2010. 14(2): p. 781-789.

[4] Khan, N., Y. Su, and S.B. Riffat, "*A review on wind driven ventilation techniques*". Energy and Buildings, 2008. 40(8): p. 1586-1604.

[5] Liu, S., C. Mak, and J.L. Niu, "*Numerical evaluation of louver configuration and ventilation strategies for the windcatcher system*". Building and Environment, 2011.

[6] Saadatian, O., et al., "*Review of windcatcher technologies*". Renewable and Sustainable Energy Reviews, 2012. 16(3): p. 1477-1495.

[7] Bahadori, M., "*Passive cooling systems in Iranian architecture*". Sci. Am.;(United States), 1978. 238(2).

[8] Nouanegue, H., L. Alandji, and E. Bilgen, "*Numerical study of solar-wind tower systems for ventilation of dwellings*". Renewable Energy, 2008. 33(3): p. 434-443.

[9] Montazeri, H. and R. Azizian, "*Experimental study on natural ventilation performance of one-sided wind catcher.* Building and Environment, 2008. 43(12): p. 2193-2202.

[10] Elmualim, A.A. and H.B. Awbi, "*Wind tunnel and CFD investigation of the performance of wind catcher ventilation systems*". International Journal of Ventilation, 2002. 1(1): p. 53-64.

[11] Elmualim, A.A., "*Dynamic modelling of a wind catcher/tower turret for natural ventilation*". Building Services Engineering Research and Technology, 2006. 27(3): p. 165-182.

[12] Li, L. and C. Mak, "*The assessment of the performance of a windcatcher system using computational fluid dynamics*". Building and Environment, 2007. 42(3): p. 1135-1141.

[13] Elmualim, A., et al. "*Evaluating the performance of a windcatcher system using wind tunnel testing*". 2001.

[14] Su, Y., et al., "*Experimental and CFD study of ventilation flow rate of a Monodraught™ windcatcher*". Energy and buildings, 2008. 40(6): p. 1110-1116.

[15] Hughes, B.R. and S. Abdul Ghani, "*A numerical investigation into the effect of windvent dampers on operating conditions*". Building and Environment, 2009. 44(2): p. 237-248.

[16] Awbi, H. and A. Elmualim. "*Full scale model Windcatcher performance evaluation using a wind tunnel*". 2002.

[17] Montazeri, H., "*Experimental and numerical study on natural ventilation performance of various multi-opening wind catchers*". Building and Environment, 2011. 46(2): p. 370-378.

[18] Montazeri, H., et al., "*Two-sided wind catcher performance evaluation using experimental, numerical and analytical modeling*". Renewable Energy, 2010. 35(7): p. 1424-1435.

[19] Chen, Q.Y., "*Using computational tools to factor wind into architectural environment design*". Energy and buildings, 2004. 36(12): p. 1197-1209.

[20] Richards, P. and R. Hoxey, "*Appropriate boundary conditions for computational wind engineering models using the k- turbulence model*". Journal of Wind Engineering and Industrial Aerodynamics, 1993. 46: p. 145-153.

[21] Casey, M. and T. Wintergerste, "*Best Practices Guidelines: Ercoftac Special Interest Group on Quality and Trust in Industrial CFD*". 2000: Ercoftac.

[22] Franke, J. "*Recommendations on the use of CFD in wind engineering*". 2004.

[23] Blocken, B., T. Stathopoulos, and J. Carmeliet, "*CFD simulation of the atmospheric boundary layer*"*: wall function problems.* Atmospheric environment, 2007. 41(2): p. 238-252.

[24] Tominaga, Y., et al., "*AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings*". Journal of Wind Engineering and Industrial Aerodynamics, 2008. 96(10): p. 1749-1761.

[25] Mahyari, A., "*The wind catcher: a passive cooling device for hot arid climate*". 1996.

[26] Straw, M.P., "*Computation and measurement of wind induced ventilation*". 2000, University of Nottingham.