AN OPTIMIZATION OF WIND CATCHER GEOMETRY IN A PASSIVE DOWNDRAUGHT COOLING TOWER USING CFD

Sarjito[1]

[1]Mechanical Engineering Department, Faculty of Engineering, Universitas Muhammadiyah Surakarta
Jl. A. Yani Tromol Pos 1 Pabelan Kartasura 57102 Telp 0271 717417
Email: sarjito_ums@yahoo.com

Abstrak

The aims of the research work described in this paper is a part were to use computational fluid dynamics (CFD) to investigate the factors affecting the performance of a single-stage downdraught evaporative cooling device for low-energy cooling of buildings developed from a novel prototype device described by Pearlmutter et al. (1996; 2008); and to model and explore the performance of the device when integrated within a hypothetical building. This involved carrying out simulations: to select the most effective wind catcher geometry. Two types of wind catcher using curved deflector and closed cowl design were studied: In total five alternative arrangements were investigated. Arrangements 1 and 2 were bi-directional wind catchers. Arrangement 1 was modelled without a baffle and arrangement 2 was modelled with an extended baffle. Arrangements 3, 4 and 5 were uni-directional closed cowls. Arrangement 3 was modelled without a baffle, arrangement 4 was modelled with a short baffle and arrangement 5 was modelled with an extended baffle and an increased inner radius of 1 metre which had the effect of raising the mid-plane height of the cowl inlet by 1 metre. Initially, for comparison in all studies, the inlet wind speed was set at 10 m/s at a reference height of 11.5 metres which corresponded to the mid plane height of the wind catcher and wind cowl entry ducts for arrangements 1 to 4. All simulations were carried out using ANSYS CFX, versions 13.0, and the performances of the device were focused in selecting optimum air flow induced into the devices. The CFD simulations were carried out to define the optimum geometry of a wind catcher. Based on these simulation results, it was concluded that a uni-directional closed-cowl design was the most effective arrangement.

Key words: *CFD; Comfortable living; Energy and building, Wind catcher*

Introduction

Strategies for the thermal management of living, working and storage spaces in hot dry climates which minimize the use of air-conditioning systems have grown rapidly nowadays. Such strategies are usually based upon harnessing effects such as evaporation in conjunction with the stack or chimney effect to induce buoyancy-driven upward or downward flows, and it is well known that substantial levels of cooled downward flow or downdraught can be induced in a column of air by introducing a finely-atomized water mist at its upper end.

The performance of downdraught evaporative cooling devices is limited by the prevailing ambient wet-bulb temperature. As the cooled airflow temperature approaches the wet-bulb value, further reductions in temperature become impossible, and overall performance improvements can be achieved only by increasing the airflow, by optimizing the device geometry and by using fans, and by optimizing the evaporation process to minimize both water consumption and the need to recycle excess liquid water.

Wind catcher and wind cowl designs

Wind catchers and wind cowls have existed for many hundreds of years in hot dry middle-eastern countries. They exist in uni-directional and multi-directional forms as shown in Figure 1. Recent developments in low-energy building design have revived interest in these devices. The thesis by Elzaidabi (2008) and the review papers by Khan et al. (2008) and by Hughes et al. (2012) provide up-to-date information on a wide range of modern wind catcher and wind cowl designs.

In a recent comparative study, Bahadori et al. (2008) described experiments and one-dimensional numerical calculations on two new designs of wind tower with evaporative cooling and on a conventional wind tower of identical dimensions without evaporative cooling which was used as a control. In one tower, the arrangements for evaporative cooling consisted of a vertical array of cloth curtains wetted by water sprays. In the other tower evaporative cooling was provided by wetted pads at the tower inlet. In both towers with evaporative cooling, the outlet air was found to have a much lower temperature and higher relative humidity than the ambient air. Under ambient conditions of 29 to 37°C and 10 to 13% relative humidity, outlet air temperatures between 20 to 24°C were achieved with relative humidity values of 63 to 80% approximately. The arrangement with the vertical array of

cloth curtains was found to perform better in high wind speeds than the arrangement with wetted pads. At low wind speeds the arrangement with wetted pads gave the better performance.
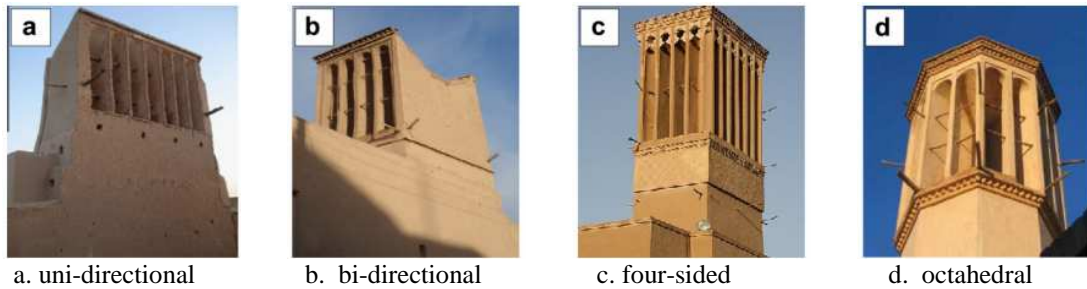
a. uni-directional   b. bi-directional   c. four-sided   d. octahedral

Figure 1 Traditional wind towers in Yazd in Iran
(Hughes et al. (2012), with permission from Elsevier, license no. 2906050180807)

A detailed comparative experimental study of the effectiveness of a number of traditional wind catcher designs was carried out at one third scales by Pearlmutter et al. (1996). The alternative designs which had a flow area of 1 m^2 are shown in Figure 2.
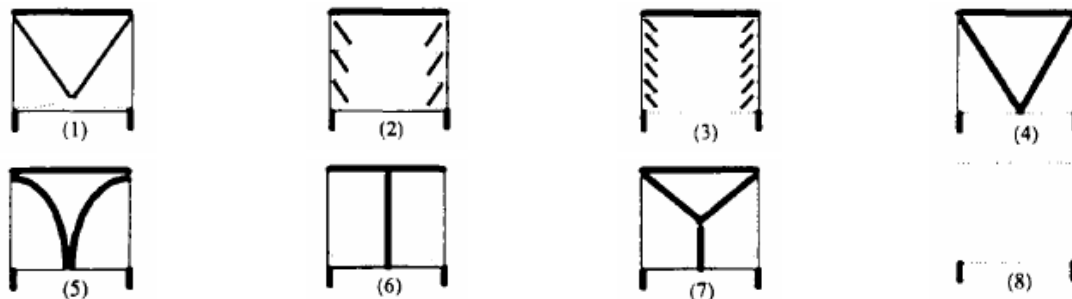

Figure 2 Alternative wind catcher designs (Pearlmutter et al. (1996),
with permission from Elsevier, license no. 2901170325105)

The three designs configurations 1-3 in Figure 2 utilized inwardly swinging louvres, the configurations 4 and 5 used flat and curved fixed deflectors, the configuration 6 employed a single central swinging louvre, and the configuration 7 used a single central swinging louvre combined with fixed deflectors. Configuration 8, with no wind catcher, was used as a control. The configuration 5 with the curved fixed deflector was found to have minimal flow resistance and to provide the greatest overall wind capture.

In some cases, modern designs of wind cowls have the ability to separate the fresh incoming air from the warm stale outgoing air and to rotate or weather-vane so that the inlet duct faces the incoming wind at all times and the outlet duct discharges into a low pressure area. An example of such a device is shown in Figure 3.

Figure 3 Wind cowl (Source: www.zedfactory.com)

CFD simulations of wind-driven ventilation and evaporative cooling in buildings

Although much experimental investigation and one-dimensional numerical modelling of evaporative cooling has been carried out in recent years, studies which investigate wind-driven passive evaporative cooling in buildings using CFD methods are relatively uncommon in the literature.

Li and Mak (2007) used CFD to investigate the performance a louvred wind vent for wind speeds in the range 0.5 to 6 m/s using FLUENT. The outlet boundaries at the base of the wind catcher were modelled as pressure openings and the ventilated space below the wind catcher was not modelled. The study did not include water spray

modelling. An unstructured mesh using approximately 50,000 elements was used and the standard two-equation k-epsilon turbulence model was adopted. Although the mesh was relatively coarse, the reported errors between the simulated air flow rate entering the ventilated space and published experimental wind tunnel results lay in the range 2% to 22%. The authors concluded that the CFD technique could be used to study other wind catcher systems.

Hughes and Mak (2011) simulated a standard commercially-available wind vent, using FLUENT. The flow domain consisted of an external space or macro-climate to simulate the ambient wind condition of 4.5 m/s and a micro-climate to simulate an occupied space which was a standard classroom at Sheffield Hallam University. The study did not include water spray modelling. An adaptive mesh of approximately 1,200,000 elements with the k-epsilon turbulence model was used to obtain the solution. Velocity predictions at 9 sampling points were compared with experimental measurements and an error range of 3% to 8% was reported.

De Melo and Guedes (2007) describe experimental measurements and a simplified two-dimensional CFD simulation of an existing building which had been modified to incorporate a passive downdraught evaporative cooling shower system. The building was situated in Moura, a hot-dry area of Portugal, in which the daytime ambient air temperature in August can reach 42°C and the relative humidity is less than 20%. The work provides useful experimental data on the performance of the shower system with internal temperatures between 22°C and 26.6°C and relative humidity values from 62 to 95.5% being recorded. The CFD simulation which used the CFD software AMBIENS, is not described in detail, with no information on boundary conditions, or on mesh quality or turbulence modelling being provided.

In a simultaneous paper, Cook et al. (2000) describe a CFD simulation of passive evaporative cooling in a hypothetical office building in the centre of Seville, Spain using CFX-4.2. A sectional elevation of the hypothetical building, which was designed by Ford & Associates for use in performance simulations, is shown in Figure 4.
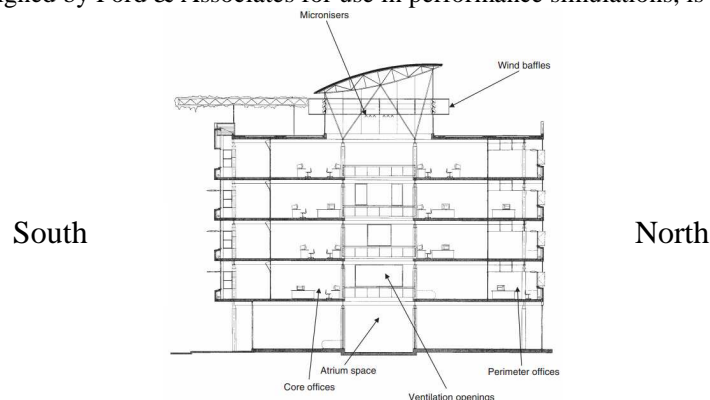

Figure 4 Sectional elevation of hypothetical building design by Ford & Associates, (Robinson et al. (2004), with permission from Sage publications, no license required)

A full description of the expected air flow paths within the building is provided in the paper together with a detailed description of the theoretical background to the droplet thermal, momentum and mass transfer model implemented within CFX-4.2.

The CFD simulations reported in the paper were carried out to investigate the air flows, temperatures and relative humidity produced within the building atrium, core offices and perimeter offices with the passive evaporative cooling system operating under conditions of no wind, and with a wind speed of 4 m/s from the South and from the North. For all the simulations, appropriate ambient conditions for Seville of 35°C and 34% relative humidity were used, but no account was taken of ambient wind shear, and it is not clear from the paper if wind flow around the building in the East-West direction was considered. In addition, from the images supplied in the paper, the upstream and the downstream flow domain dimensions appear a little small and the flow domain height seems restricted. No information on the precise flow domain dimensions is given to enable these observations to be checked.

An initial series of simulations were carried out to establish a suitable rate of spray water mass flow rate, and it was found that a water injection rate of 51 litre/hr or 0.014 kg/s would achieve comfortable conditions in the occupied spaces. The spray water injection was modelled using particle tracking with each atomiser represented by using 5 trajectories. No information on the type of atomiser or on the droplet size distribution used in the simulations is given in the paper and there is no discussion of flow domain sizing or mesh quality. All solutions were carried out using the standard k-epsilon turbulence model which was selected by the authors for its robustness, accuracy and speed compared to other models.

The results from the simulations are presented as velocity vector plots and temperature and relative humidity contour plots in the absence of wind, and with a wind speed of 4 m/s from the South and from the North. In the

simulation with no wind, the authors report air temperatures of about 29°C in the lower spaces and observe that the higher spaces are overheated. For the simulations with wind effects included, the authors observe that the downward flow within the atrium space exhibited significant asymmetry due to the cross flow at the upper inlets, with the result that the temperature and relative humidity distributions within the core offices and perimeter offices also showed local asymmetry. The temperature distributions within the building obtained from the simulations with wind speeds of 4 m/s from the South and from the North are shown in Figure 5, (the original images are black-and-white only).
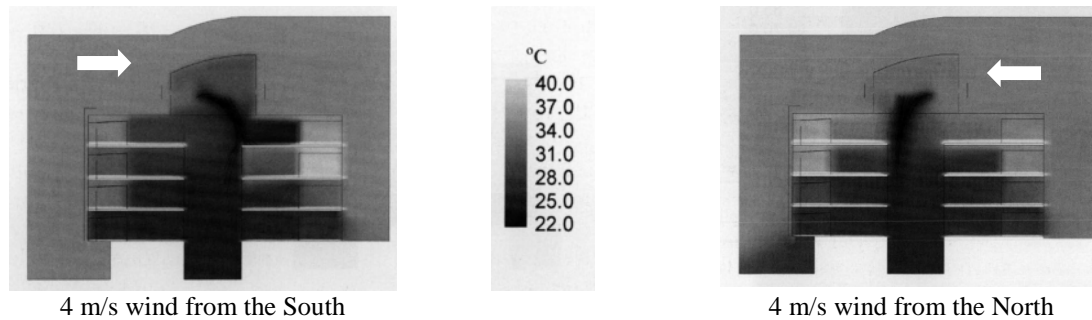

| 4 m/s wind from the South | 4 m/s wind from the North |

Figure 5 Temperature distributions with passive evaporative spray cooling, (Cook et al. (2000)

The local asymmetric effect within the office spaces can be seen clearly from these plots, however, the precise temperature and relative humidity levels achieved are not stated clearly by the authors. It appears from the plots shown above that temperatures below 25°C were achieved over most of the lower floor and also in the core offices on the second floor. Limited information on the relative humidity levels within the building is provided in the paper, with just one image for the simulation with the wind speed from the South being included. This image indicates high relative humidity levels, in excess of 85%, in the regions of the lower floor and the core offices on the second floor with temperatures below 25°C, and suggests that the relative humidity levels produced in the simulation with the wind from the North were also very high. The authors observe that the openings between the atrium and the occupied spaces would require carefully sizing and vertical balancing to achieve similar temperatures on all floors. They also conclude that the CFD simulations demonstrated that particle transport modelling was a suitable mean of representing the injection and evaporation of water into an air flow.

Objectives and Methodology

Objectives

This paper contains a description of CFD studies carried out with the objective of defining the optimum geometry of a wind catcher. As the wind catcher is the first component in the evaporative cooling tower system, it was considered to be important to optimize its geometry and performance at an early stage before detailed studies of the downstream components were carried out.

Methodology

Two basic types of wind catcher have been used in recent years. A range of bi-directional wind catchers with two openings were studied by Pearlmutter et al. (1996), and single-entry closed-cowl wind catchers were considered by Chance, (2009). In the work by Pearlmutter et al. (1996), the type 5 wind catchers shown in Figure 2 which featured curved deflectors was found to be the most effective. Further performance advantages are possible if uni-directional single entry cowl designs can be used.

In the present work two types of wind catcher were studied: one using curved deflectors and the other using a closed-cowl design. In total five alternative arrangements were investigated. These are shown in Figure 6

Arrangements 1 and 2 were bi-directional wind catchers. Arrangement 1 was modelled without a baffle and arrangement 2 was modelled with an extended baffle. Arrangements 3, 4 and 5 were uni-directional closed cowls. Arrangement 3 was modelled without a baffle, arrangement 4 was modelled with a short baffle and arrangement 5 was modelled with an extended baffle and an increased inner radius of 1 metre which had the effect of raising the mid-plane height of the cowl inlet by 1 metre. Initially, for comparison in all studies, the inlet wind speed was set at 10 m/s at a reference height of 11.5 metres which corresponded to the mid plane height of the wind catcher and wind cowl entry ducts for arrangements 1 to 4.

The flow domain dimensions (x, y, z) were set to (15, 25, 15) metres and the principal vertical axis of the model was located 7.5 metres from the domain inlet. A three-dimensional image of the flow domain for arrangement 5 is shown in Figure 8(a). The same flow domain dimensions were used for all the arrangements. The computational domain was split in two parts with the upper part as a free stream volume of height 19 metres and the

lower part as an enclosed volume of height 6 metres to represent the space being cooled. In all cases a transition duct of length 3 metres was incorporated to change the cross-sectional shape from 3 metres square to a 3 metres diameter circular duct of length 1 metre. The sheet thickness used for the wind catcher and cowl material was 0.02 metres and the ceiling thickness was specified as 0.1 metres.

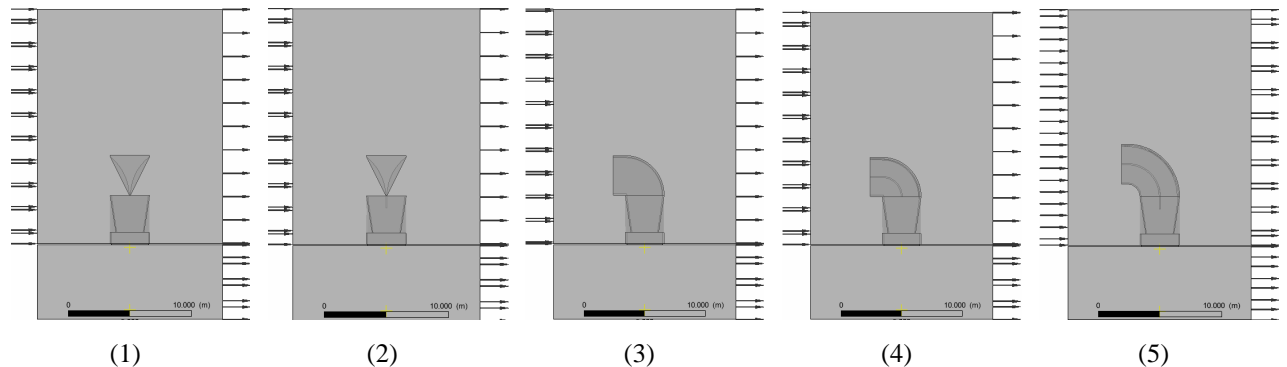(1)　　　　　　　　(2)　　　　　　　　(3)　　　　　　　　(4)　　　　　　　　(5)

Figure 6- Type of wind capture investigated

An unstructured mesh was set up within the computational domain using minimum and maximum face spacings of 0.05 metres and 0.5 metres on the ceiling, duct, wind catcher and wind cowl surfaces, and minimum and maximum face spacings of 0.065 metres and 0.5 metres on the remaining surfaces. The boundary layers on the model surfaces were approximated using a near wall inflation with five layers of elements with a growth ratio of 1.2 in a total thickness of 0.02 metres. The spatial growth rate of the volume mesh was controlled using an expansion factor of 1.2. These settings created approximately 754,000 elements, and resulted in y+ values less than 100. This quality of boundary layer modelling was considered to be acceptable in these initial simulations.

An inlet boundary condition was imposed on the Z-plus face in the upper part of the flow domain above the ceiling and an outlet boundary condition with zero static pressure relative to the atmospheric pressure datum was imposed on the Z-minus face both above and below the ceiling. The X-plus, X-minus and Y-plus faces in the upper part of the flow domain were set as free slip adiabatic wall boundaries and all others solid boundaries were set as no slip adiabatic walls.

A no-slip adiabatic wall boundary condition was imposed on all solid surfaces and an inlet velocity profile $V(y)$ representing the effects of wind shear was set on the domain upstream face in accordance with the following equation (Smith Et al. 2002):

$$\frac{V(y)}{V_{ref}} = \left(\frac{y}{H_{ref}} \right)^{\alpha} \tag{1}$$

In this equation, $V(y)$ is the wind velocity (m/s) at height y (metres), V_{ref} is the reference wind velocity at a reference height H_{ref}, and the exponent α gives a measure of the surface roughness of the local terrain. For the present work a value of 0.14, corresponding to grassy level ground was used for α, and steady-state solutions were obtained initially for a constant wind speed V_{ref} of 10 m/s at the reference height of 11.5 metres. The simulations were carried out using auto timescale control, at a standard temperature of 25°C with a residual target of 1E-4 using the k-epsilon turbulence model with the inlet turbulence level set to 5%. Initially, the simulation were carried out for 300 iterations and then continued until 400 iterations. As no changes in the results were observed then considered to be convergence.

Results and Discussions

Vector plots of velocity on the YZ symmetry plane for the five alternative arrangements are shown in Figures 7. A number of interesting flow features can be seen from the above (Figures 7.a to 7.e). In the arrangement with the wind catcher without a baffle, (Figure 7.a), a significant flow spillage is evident from the downstream opening. This flow spillage is eliminated in the wind catcher with the extended baffle, (Figure 7.b). There is no flow spillage in the wind cowl arrangements.

In the wind cowl without a baffle, (Figure 7.c), a significant region of flow separation can be seen on the inner side of the bend. This flow separation is reduced but not entirely eliminated in the wind cowl with the short baffle, (Figure 7.d). A separate region of separated flow can be seen on the concave face of the baffle. In the wind

cowl with the increase inner radius and extended baffle, (Figure 7.e), both regions of flow separation are greatly reduced, as is the asymmetry in the flow entering the lower space.

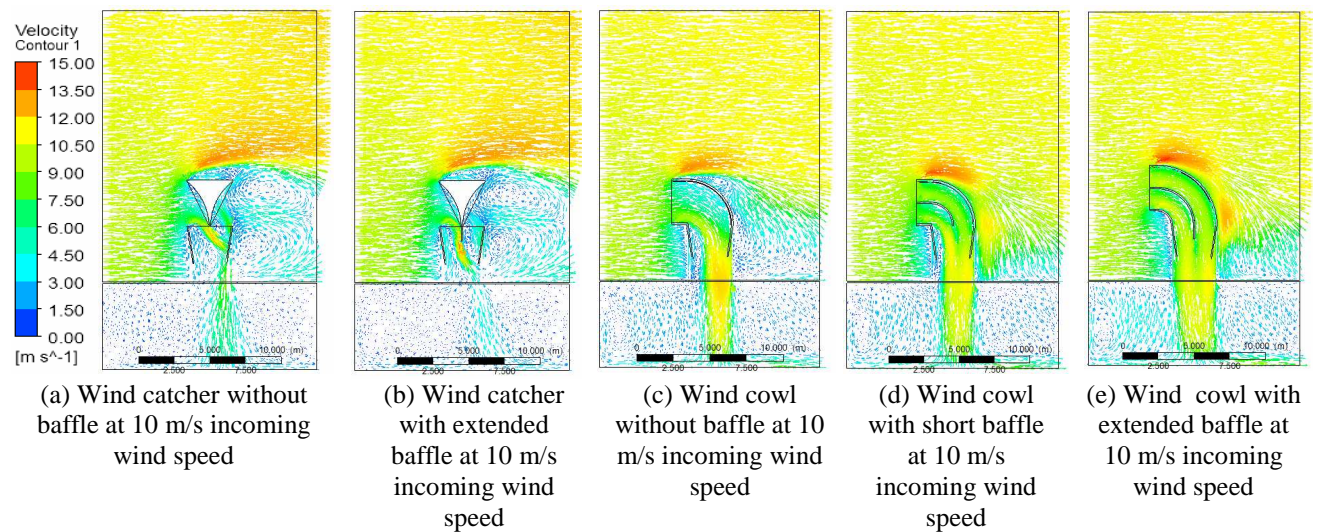| (a) Wind catcher without baffle at 10 m/s incoming wind speed | (b) Wind catcher with extended baffle at 10 m/s incoming wind speed | (c) Wind cowl without baffle at 10 m/s incoming wind speed | (d) Wind cowl with short baffle at 10 m/s incoming wind speed | (e) Wind cowl with extended baffle at 10 m/s incoming wind speed |

Figure 7-Vector plot of velocity on for five types of wind catcher investigated

By employing the CFX function calculator the mass flows captured by each arrangement were determined. The relative performance of each arrangement was assessed by comparing the mass flow captured with the potential mass capture rate based on the free-stream flow entry area and mid-height velocity of each arrangement. The values obtained are shown in Table 1.

Table 1- Comparison of mass capture performance

Type of wind capture arrangement	Entry area (m^2)	Mid- height velocity (m/s)	Airmass flow rate (kg/s)	Mass capture "efficiency" (%)
a. Wind catcher without baffle	9	10	17.3	16.2
b. Wind catcher with extended baffle	9	10	23.5	22.0
c. Wind cowl without baffle	9	10	63.2	59.2
d. Wind cowl with short baffle	8.94	10	72.4	68.3
e. Wind cowl with extended baffle	8.94	10.12	85.2	79.4

In examining the data in Table 1, it must be recognized that in any real building installation there would be significant flow resistance generated in the lower space which would reduce the above mass flows significantly. Also, it must be noted that all simulations were carried out with the wind in the z-minus or optimum direction as indicated in (Figure 8.a). Clearly, if the wind direction were to be altered so that the wind entered at an angle, this would reduce the mass flows also. As all arrangements could be made to rotate or weather-vane to face the prevailing wind direction by including a horizontal slewing-ring or bearing, this effect could be avoided. For the purposes of comparison of the relative performance of the arrangements, valid conclusions can be drawn from the data shown above. It is clearly evident from the tabulated results that the wind cowl with the extended baffle was the best arrangement.

Investigation of local wind flows

Having established that the wind cowl with the extended baffle produced the best performance, further studies were carried out to investigate the effect of local flow disturbances which could be caused by proximity to the leading edge of a building and to establish the performance over a range of wind speeds from 0.5 to 5 m/s. The flow domain used for the previous investigations was modified by extending the upstream distance by 7.5 metres as shown in (Figure 8.b) to simulate the effect of placing the wind cowl close to the leading edge of a building. The image in (Figure 8.a) shows the original flow domain which can be considered as representing a wind cowl placed at a distance from the leading edge of a building.

In carrying out the simulations the effect of wind shear was modelled in the same way as in the previous study using a reference height of 11.5 metres to define the incoming wind speed profile. By employing the CFX function calculator the mass flows captured for each wind speed were determined. The values obtained are shown in Table 2.
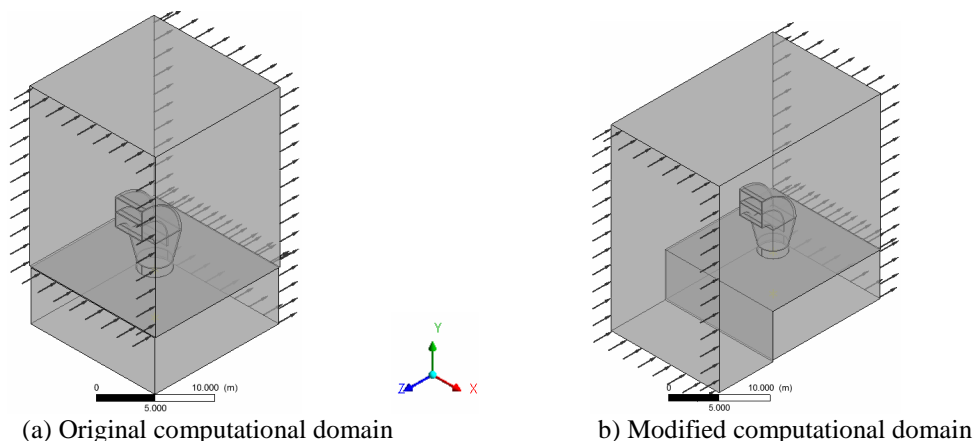
(a) Original computational domain   b) Modified computational domain

Figure 8-Flow domain for local flow investigations

Table 2- Mass capture performance near building leading edge

Incoming wind speed (m/s)	Mass flow with wind cowl distant from building leading edge (kg/s)	Mass flow with wind cowl close to building leading edge (kg/s)
0.5	0 (stalled)	0 (stalled)
1	0 (stalled)	0 (stalled)
1.5	0.1	12.1
2	8.3	20.9
2.5	14.2	29.9
3	20.3	37.5
4	31.1	53.2
5	40.3	67.1

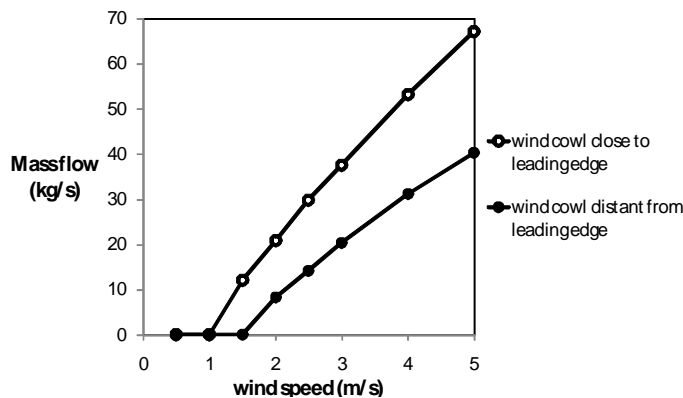The variation in the captured mass flow with wind speed is shown in Figure 9.

Figure 9-Variation in captured mass flow with wind speed

The flow-enhancing effect of the building leading edge is clearly evident in Figure 9. However, it must be recognized that, in reality, the leading edge flow would be un-steady, and the flow enhancement effects would be sensitive both to the distance of the wind cowl from the leading edge of the building and to the height of the wind cowl. They would therefore be unpredictable.

Conclusions

CFD simulations were carried out to define the optimum geometry of a wind catcher. Based on the results of these simulations it was concluded that a uni-directional closed-cowl design was the most effective arrangement.

Acknowledgements

References

ANSYS 13.0, CFX-Pre User's Guide, (2010).

Bahadori, M. N. et al. (2008) 'Experimental investigation of new design of wind tower', Renewable Energy, (33), pp. 2273-2281.

BedZed (2009) Available at http://www.ZedFactory.com, Accessed: 05 May 2012

Chance, T. (2009) 'Toward sustainable residential communities; the Beddington zero energy development (BedZed) and beyond', Environment and Urbanization, (21), pp. 527-544.

Cook, M. J. et al. (2000) 'Passive downdraught evaporative cooling', Indoor Built Environment, pp. 325-334.

De Melo, A. C. M. and Guedes, M. C. (2007) 'Dynamical and thermal modelling of PDEC: using traditional chimney and new dwelling as case studies in Portugal', Instituto Superior Técnico, Universidade Técnica de Lisboa, Lisboa, Portugal.

Elzaidabi, A. A. M. (2008) 'Low Energy, Wind Catcher Assisted Indirect - Evaporative Cooling System for Building Applications', Published PhD thesis, UK, University of Nottingham.

Hughes, B. R. and Mak, C. M. (2011) 'A study of wind and buoyancy driven flows through commercial wind towers', Energy and Buildings, (43), pp. 1784-1791.

Hughes, B. R. et al. (2012) 'The development of commercial wind towers for natural ventilation: A review', Applied Energy, (92), pp. 606-627.

Khan, N. et al. (2008) 'A review on wind driven ventilation techniques', Energy and Buildings, (40), pp. 1586–1604.

Li, L. and Mak, C. M. (2007) 'The assessment of the performance of a wind catcher system using computational fluid dynamics', Building and Environment, (42), pp. 1135-1141.

Pearlmutter D.et al., Refining the use of evaporation in an experimental downdraft cool tower", Energy and Buildings, 23, (1996) 91-197.

Pearlmutter D. Et al., A multi-stage down-draft evaporative cool tower for semi-enclosed spaces: Experiments with a water spraying system, Solar Energy, 82, (2008) 430–440.

Smith, K. et al. (2002) 'Evaluation of Wind Shear Patterns at Midwest Wind Energy Facilities', National Renewable Energy Laboratory, NREL/CP-500-32492, Association (AWEA) WINDPOWER Conference Portland, p. 3.