CHEMICAL ENGINEERING TRANSACTIONS
VOL. 34, 2013

A publication of
AIDIC
The Italian Association of Chemical Engineering
Online at: www.aidic.it/cet

Guest Editors: Neven Duić, Petar Sabev Varbanov

ISBN 978-88-95608-25-9; ISSN 1974-9791 DOI: 10.3303/CET1334008

Numerical Investigation of the Integration of Heat Transfer Devices into Wind Towers

John Kaiser Calautit[a], Ben Richard Hughes[a], Saud Ghani*[,b]

[a]School of Civil Engineering, University of Leeds, Leeds, LS2 9JT, United Kingodm
[b]Department of Industrial and Mechanical Engineering, Qatar University, Doha, Qatar
cnjksc@leeds.ac.uk

The purpose of this study is to incorporate heat transfer devices inside the passive terminal of a wind tower unit, highlighting the potential to achieve minimal restriction in the external air flow stream while ensuring maximum contact time, thus optimising the cooling duty of the device. Computational Fluid Dynamics (CFD) was used to develop a numerical model of a wind tower system and simulate the air flow pattern around and through the device to the test room. Results have indicated that the average internal airflow rate was reduced following the integration of the vertical and horizontal heat transfer device configuration, reductions of 4.11 % and 8.21 % was obtained respectively. Furthermore, the proposed cooling system was capable of reducing the air temperatures by up to 15 K. The technology presented here is subject to IP protection under the QNRF funding guidelines.

1. Introduction

In hot and arid regions such as the Middle East, there is a large dependence on electricity to run mechanical systems for providing ventilation and thermal comfort. During the summer months more than 50 % of the electrical demand comes from HVAC loads (Lombard et al., 2008). As stated by (WBCSD, 2011), commercial and residential buildings accounts for almost 40 % of the world energy usage. The resulting carbon emissions are substantially more than those in the transportation sector. This represents a significant opportunity for reducing the buildings energy consumption and greenhouse gas emissions. Natural ventilative methods such as wind towers are increasingly being employed in new buildings to reduce the energy consumption and carbon foot print.

Wind towers have been in existence in various forms for centuries as a non-mechanical means of providing indoor ventilation, energy prices and climate change agendas have refocused engineers and researchers on the low carbon credentials of modern equivalents. Conventional and modern wind towers architecture can be integrated into the designs of new buildings, to provide thermal comfort without the use of electrical energy (Hughes et al., 2011). A wind tower system is divided by partitions to create different shafts. One of the shafts functions as inlet to supply the wind and the other shafts works as outlet to extract the warm and stale air out of the living space as shown in Figure 1 (Jones and Kirby, 2008). Experimental and numerical studies (Hughes and Cheuk-Ming, 2011) have shown that wind driven force is the primary driving force for the wind tower device, providing 76 % more indoor ventilation than buoyancy driven forces.

The cooling capabilities of wind towers which depend mainly on the structure design itself are inadequate. A recent study by (Calautit et al., 2012) investigated the ventilation and thermal performance of a row house model integrated with a traditional and modern four-sided wind tower. Both systems were capable of supplying the required internal air supply rates however the reductions in internal temperature were insignificant (1-2 K). Therefore it is essential to cool the air in order to reduce the building heat load and improve the thermal comfort of its occupants during the summer months (Kalantar, 2009). Traditional wind tower systems were integrated with evaporative cooling devices to increase its thermal performance. The induced warm air is passed through cooling tubes or moist surfaces, which allows for the cooling of the air stream before entering the living spaces (Bouchahm, 2011). However, the addition of these cooling devices may reduce the air flow rate inside the channel and reduce the overall efficiency of the wind tower. Another disadvantage of this configuration is the requirement of taller towers to have sufficient contact time between the air flow and cooling surfaces (Calautit et al., 2013). (Hughes et al., 2012) reviewed different types of wind tower designs and cooling methods which have been studied using numerical CFD analysis. Furthermore, several works have also used analytical methods (Jones and Kirby, 2009), wind tunnel (Montazeri et al., 2010) and far-field testing (Hughes and Ghani, 2009) to validate the results. The good correlation between different methods of analysis suggests that the CFD techniques in use were suitable for this type of device and such have been used for the purpose of this research.

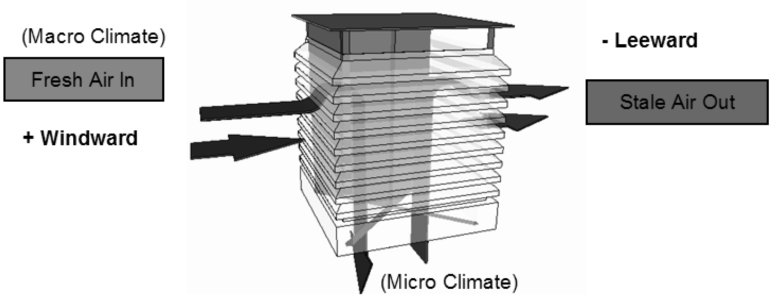

Figure 1: A flow diagram representing ventilation through a multi-direction wind tower

A wind tower system incorporating heat transfer devices was designed to meet to lower the internal temperatures in hot and humid condtions. Heat transfer devices were installed inside the passive terminal of the wind tower unit, highlighting the potential to achieve minimal restriction in the external air flow stream while ensuring maximum contact time, thus optimsing the cooling duty of the device. A standard roof-mounted wind tower was used as a benchmark for the comparison of the two different heat transfer device orientations.

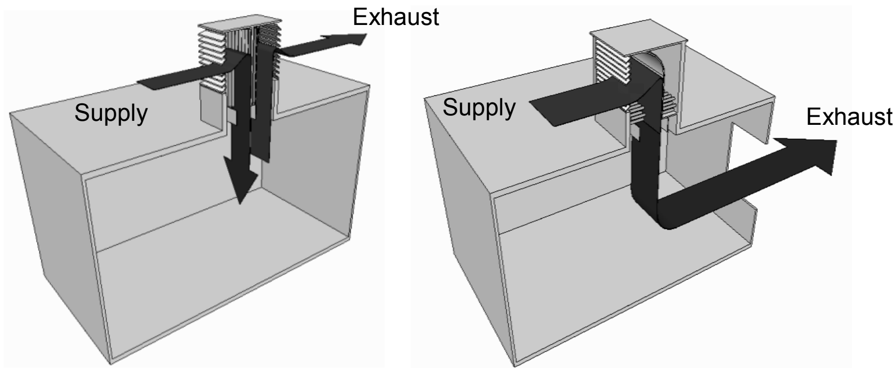

Figure 2: Wind tower systems incorporating (a) vertical HTD arrangement (b) horizontal HTD arrangement

2. CFD Setup

A four-sided square wind tower with an internal diameter of 1 m and height of 1.5 m is used for the numerical analysis. The vertical HTD arrangement is identical to the standard wind tower model, the heat transfer devices are closely arranged in an x-shaped vertical pattern positioned next to the cross-dividers inside the passive terminal. A one-sided circular wind tower is used for the analysis of the horizontal heat pipe configuration, while retaining the physical parameter and standard components of the benchmark model. A total of 70 copper HTDs with an internal diameter and length of 0.015 m and 0.9 m were used in the simulation. Figure 2 displays the proposed configurations.

The proposed wind tower system with the heat transfer devices is incorporated to a test room (micro climate) with the height, width, and length of 3, 6, and 6 m (Building Bulletin, 2012). An enclosure was created to represent the external wind environment (macro climate). The enclosure with a height, width, and length of 8, 26, and 46 m creates a direct interface through the geometry. The enclosure (the flow domain) was set at a distance from the geometry to avoid reversed flow in the region (Mehta, 1991). A flow domain representation of the physical geometry of the wind tower design under investigation and location of set boundary conditions are shown in Figure 3.

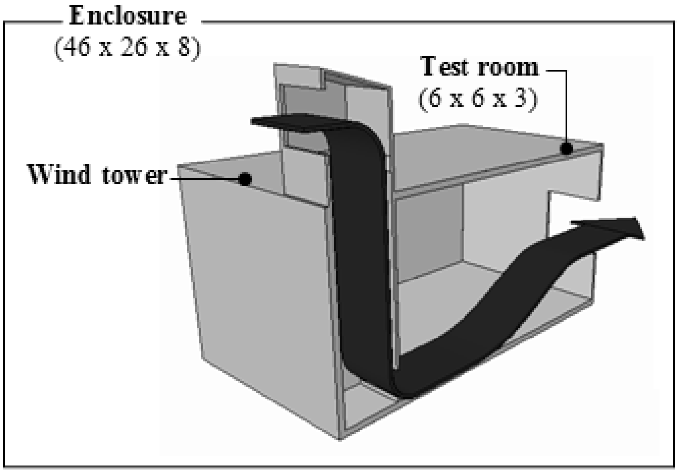

Figure 3: Schematic of a wind tower system mounted on the test room

2.1 Grid Generation

The quality of the mesh has important implication on the convergence and the level of accuracy of the achieved results (Chung, 2002). The size of the mesh element was extended smoothly to resolve the sections with high gradient mesh and to improve the accuracy of the results of the temperature fields. A non-uniform mesh was applied to the volumes of the computational model. The mesh arrangement consisted of 344,643 tetrahedral non-uniform mesh elements with 70,435 nodes, as shown in Figure 4.

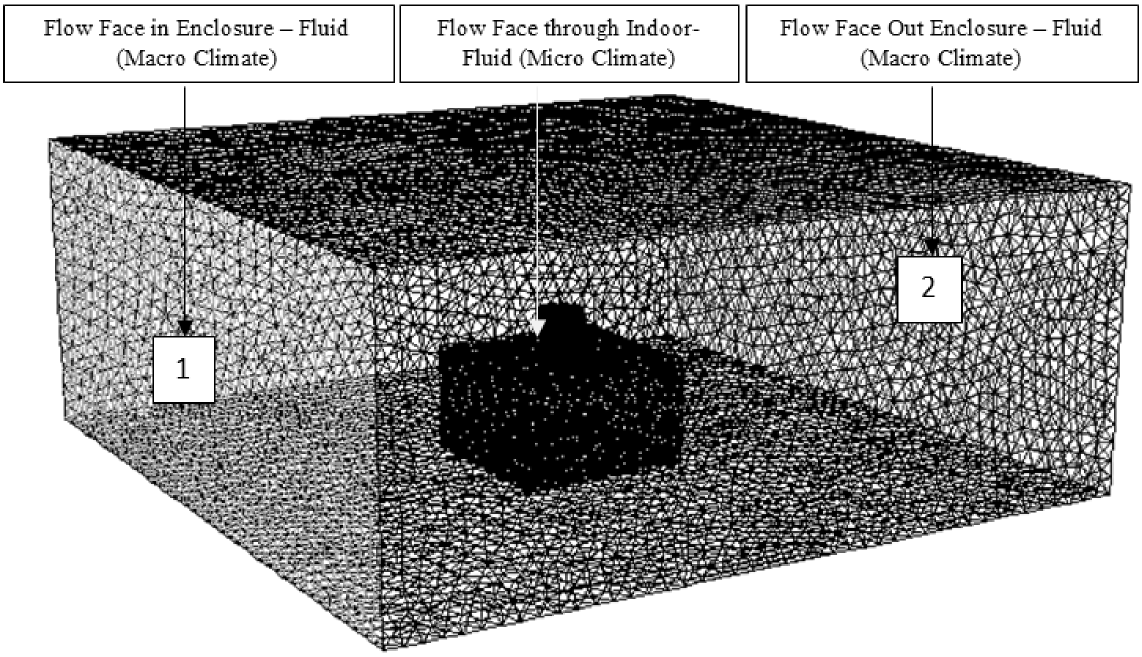

Figure 4: Flow domain representation of the physical geometry of the wind tower design under investigation and location of set boundary conditions

2.2 Boundary Conditions

The basic assumptions for the CFD simulation include a three-dimensional, fully turbulent, and incompressible flow. The CFD code uses the Finite Volume Method (FVM) approach and employs the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) velocity–pressure coupling algorithm with the second order upwind discretization as recommended in the literature (Saffari and Hosseinnia, 2009). The governing equations are the Navier-Stokes and energy equation which will not be repeated here but are available in detail in (Fluent, 2006). The turbulent nature of the flow was modelled by the standard *k*-epsilon as suggested by (Elmualim, 2006). This technique is well established in the field of research on airflow and temperature distribution in structures (Park et al., 2001). The geometry (micro climate) was modelled as an open structure with openings on the windward and leeward side which allows the incoming air to pass through it, in order to simulate and analyse the air flow pattern inside the structure. The enclosure consists of a velocity inlet (operating velocity) at the left hand side of the enclosure, and a pressure outlet (atmospheric pressure) on the opposing boundary wall of the enclosure as shown in Figure 4. The boundary conditions for the CFD model are shown in Table 1.

Table 1: CFD Model Boundary Conditions

Inlet velocity (m/s)	0.1 - 4 m/s
Inlet temperature	310 K
Pressure outlet	Atmospheric
Time	Steady State
Traditional evaporative cooling wind tower	
Injected water temperature (K)	293
Injected water mass flow rate (kg/s)	0.05
Heat transfer devices integrated wind tower	
HTD wall temperature (K)	293

2.3 Grid Adaptation

Grid adaptation was used to validate the programming and computational operation of the computational model. The numerical grid was refined and locally enriched using the hp-method grid adaptation technique (Hughes and Ghani, 2009). This procedure of evaluation requires the use of different mesh sizes or higher order approximations by the use of a posterior error estimates. The grid was evaluated and refined until the posterior estimate error becomes insignificant between the number of nodes and elements, computational iterations and the posterior error indicator. Figure 5 shows that at 1,400,000 elements the percentage error between the grid refinements was at its lowest in the last two steps.

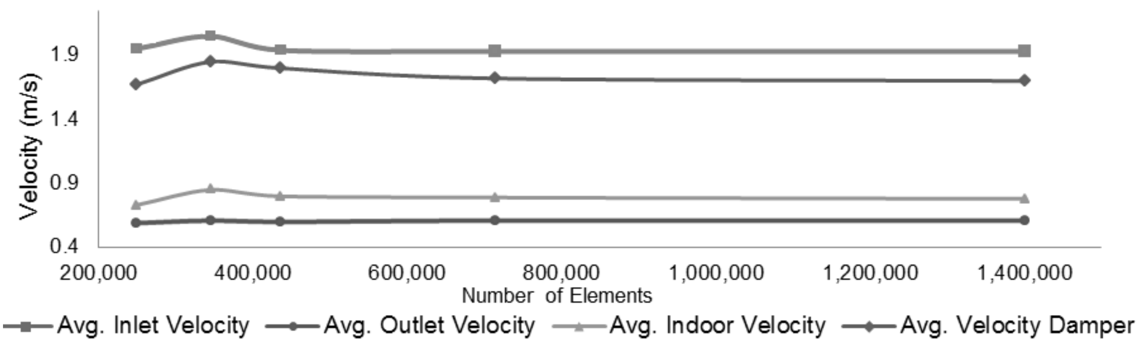

Figure 5: Comparison of solutions from various grid adaptation methods. The variables are the average velocity at the inlet and outlet of the wind towers and interior of the test room

3. Results and Discussion

Figure 6a display the velocity contour of the cross-sectional plane in the test room model. From the illustration it is observed that the air flow entering the opening of the uni-directional wind tower is directed down to the enclosed space through the floor diffuser. The airflow is accelerated as it shears across the walls and floor of the structure, reaching a maximum velocity of 3 m/s. The air stream is circulated inside the structure and exits the opening located on the other end. Figure 6b depicts the simulated temperature distribution inside the test room with a uni-directional wind tower incorporating the evaporative cooling method. Temperature reduction of 12.6 K is obtained from the CFD analysis.

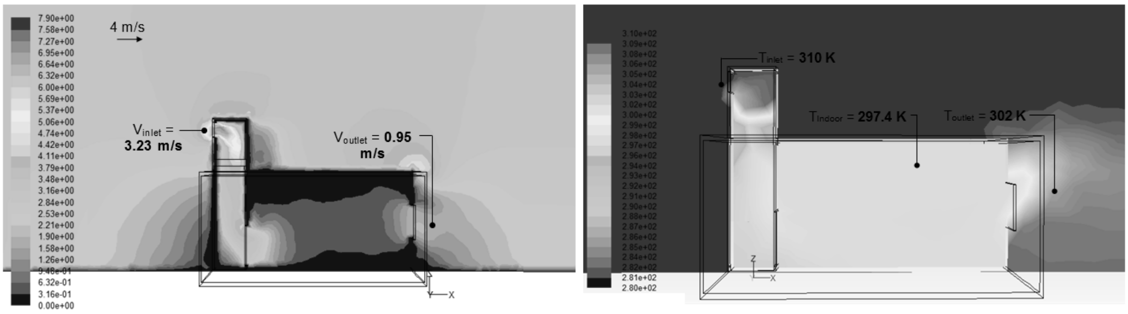

Figure 6: Velocity contour lines of a cross sectional plane in the test room with (a) Temperature distributions within the test room with the evaporative cooling tower (b)

Figure 7 shows the flow comparisons between the air supply rates as recommended by government regulations (Building Regulations, 2000) for a room of 20 occupants based on air supply per occupant, and the air supply rates as calculated through the use of the CFD modelling of benchmark four-sided wind tower, vertical heat pipe configuration, and horizontal heat pipe configuration. Each of the wind tower systems did not meet this recommendation for an external wind velocity of 1m/s; however, for each of the following external wind velocities, each wind tower system surpasses the recommendation.

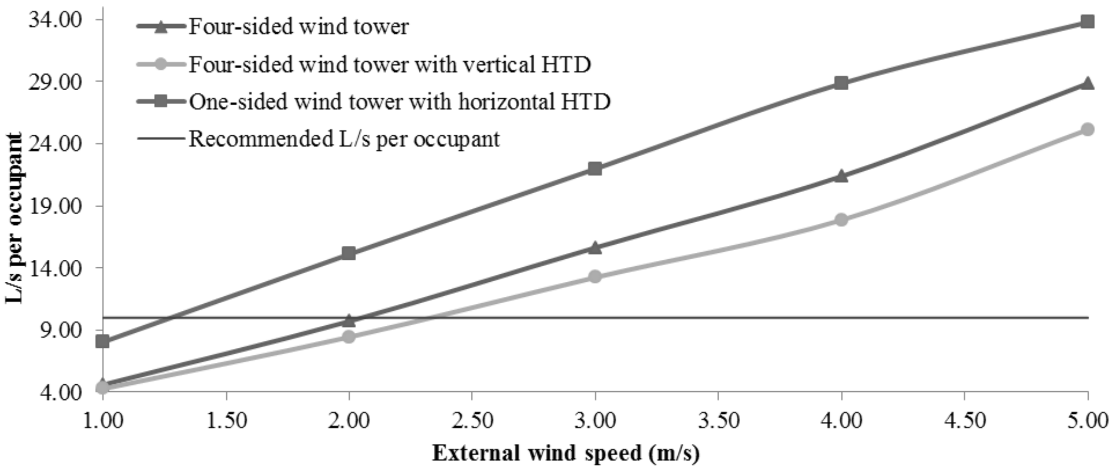

Figure 7: Comparison of the air supply rates of the wind tower configurations at various external speeds

Figure 8 shows the contour of static temperatures inside the test room with a wind tower incorporating the vertical and horizontal HTD arrangement. Air temperature reduction is observed inside the microclimate, average temperature of 296.2 K and 295.8 K are obtained inside the models, which is 11-12 % lower compared to the wind tower employing traditional evaporative cooling devices (Figure 6b).

4. Conclusions

A wind tower system involving heat transfer devices was designed to lower the internal temperatures. A numerical analysis was carried out using CFD software to simulate and analyse the air flow pattern, pressure coefficient and temperature distribution around and through the wind tower to the test room. The work highlighted the effect of evaporative cooling and heat transfer devices on the thermal performance of the passive ventilation device. The proposed cooling system was capable of reducing the air temperatures by up to 15 K, depending on the configuration and operating conditions. Results have indicated that the internal airflow rate was slightly reduced following the integration of the vertical and heat pipe configuration, reductions of 4.11 % and 8.21 % were obtained from the achieved computational model.

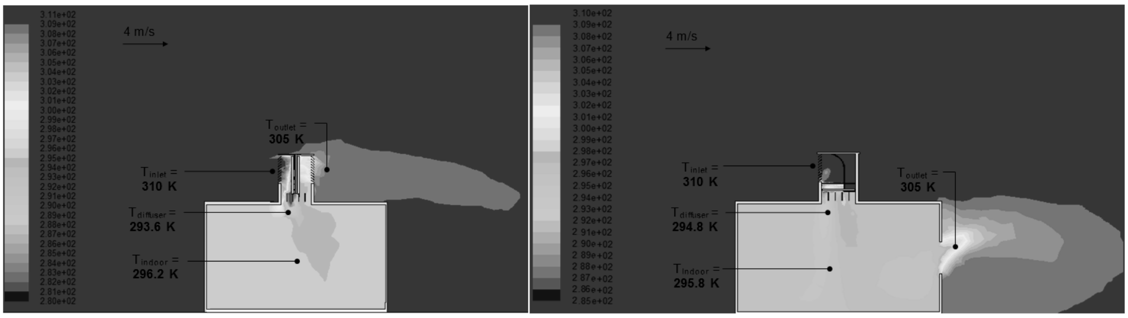

Figure 8: Temperature distributions inside the test room with a commercial wind tower incorporating the vertical HTD arrangement (a) and horizontal HTD arrangement (b).

5. Acknowledgement

This paper was made possible by a NPRP grant from the Qatar National Research Fund (A member of the Qatar Foundation). The statements made are solely the responsibility of the authors. NPRP 09-138-2-059. The technology presented is subject to IP protection under the QNRF funding guidelines.

References

ANSYS, 2006, Fluent User Guide <www.ansys.com>, Accessed 01.12.2011.

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

Building Regulations, 2000. Approved Document F1A: Means of Ventilation 2010, NBS: London, UK.

Building Bulletin, 2011. Ventilation and indoor air quality in schools Market <www.education.gov.uk>, Accessed 12.09.2011.

Calautit J.K., Hughes B.R., Ghani S.A., 2012. A numerical investigation into the feasibility of integrating green building technologies into row houses in the Middle East, Architecture Science Review, 1-18.

Calautit J.K., Chaudhry H.N., Hughes B.R., Ghani S.A., 2013. Comparison between evaporative cooling and a heat pipe assisted thermal loop for a commercial wind tower in hot and dry climatic conditions, Applied Energy, 101, 740-755.

Chung T.J., 2002. Computational fluid dynamics, Cambridge University Press, Second Edition, Cambridge, UK, 15-21.

Elmualim A., 2006. Effect of damper and heat source on wind catcher natural ventilation performance, Energy and Buildings, 38, 939-948.

Hughes B.R., Calautit J.K., Ghani S.A., 2012. The Development of Commercial Wind Towers for Natural Ventilation: a review, Applied Energy, 92, 606-627.

Hughes B.R., Cheuk-Ming M., 2011. A Study of Wind and Buoyancy Driven Flows Through Commercial Wind Towers, Building and Environment, 43, 1784-1791.

Hughes B.R. and Ghani S.A., 2010. A numerical investigation into the effect of Windvent louvre external angle. Building and Environment, 45, 1025-1036.

Jones B., Kirby R., 2009. Quantifying the performance of a top–down natural ventilation Windcatcher™, Building and Environment, 44, 1925-1934.

Kalantar V., 2009. Numerical simulation of cooling performance of wind tower (Baud-Geer) in hot and arid region, Renewable Energy, 34, 246-254.

Lombard L., Ortiz J., Pout C., 2008. A review on buildings energy consumption information, Energy and Buildings, 40, 394-398.

Mehta U., 1991. Some aspects of uncertainty in computational fluid dynamics results, Journal of Fluids Engineering, 113, 539-543.

Montazeri J., Montazeri F., Azizian R., Mostafavi S., 2010. Two-sided wind catcher performance evaluation using experimental, numerical and analytical modelling, Renewable Energy, 35,1424-35.

Saffari H., Hosseinnia S., 2009. Two-phase Euler-Lagrange CFD simulation of evaporative cooling in a Wind Tower, Energy and Buildings, 41, 991-1000.

Park H., Fazio P., Holland D., 2001. The effect of location of a convective heat source on displacement ventilation: CFD study. Building and Environment, 36, 883-889.

WBCSD, 2011. Enery Efficiency in Buildings: Transforming the Market <www.wbcsd.org>, Accessed 10.09.2011.