Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 75 (2015) 1692 – 1697

Energy Procedia

The 7th International Conference on Applied Energy – ICAE2015

CFD and Wind Tunnel Study of the Performance of a Multi-Directional Wind Tower with Heat Transfer Devices

John Kaiser Calautit[a]*, Ben Richard Hughes[a], Dominic O'Connor[a], Sally Salome Shahzad[b]

[a] Department of Mechanical Engineering, University of Sheffield, Sheffield, S10 2TN, United Kingdom
[b] School of Civil Engineering, University of Leeds, Leeds, LS2 9JT, United Kingdom

Abstract

The aim of this work was to investigate the performance of a multi-directional wind tower integrated with heat transfer devices (HTD) using Computational Fluid Dynamics (CFD) and wind tunnel analysis. An experimental scale model was created using 3D printing. The scale model was tested in a closed-loop wind tunnel to validate the CFD data. Numerical results of the supply airflow were compared with experimental data. Good agreement was observed between both methods of analysis. Smoke visualisation test was conducted to analyse the air flow pattern in the test room attached underneath it. Results have indicated that the achieved indoor air speed was reduced by up to 17 % following the integration of the cylindrical HTD. The effect of varying the number of HTD on the system's thermal performance were investigated. The work highlighted the potential of integrating HTD into wind towers in reducing the air temperature. The technology presented here is subject to a UK patent application (PCT/GB2014/052263).

Peer-review under responsibility of Applied Energy Innovation Institute

Keywords: CFD; heat transfer device; hot climates; passive cooling; wind tower/catcher;

1. Introduction

Buildings today account for 30 - 40% of the world's primary energy consumption and are responsible for about one-third of global carbon emissions. Heating Ventilation and Air Conditioning (HVAC) systems consume more than 60% of the total energy use of buildings [1]. Extensive efforts have been focused on an environmentally friendly approach to building design, revealing the on-going interest of the scientific community on the topic. Passive cooling technologies such as wind towers are increasingly being employed in buildings for increasing the fresh air rates and reducing energy consumption [1]. A wind tower is a wind driven ventilation device, which captures air at high elevations and directs the air

* Corresponding author. Tel.: +44-754-415-8981
E-mail address: J.K.Calautit@leeds.ac.uk.

Peer-review under responsibility of Applied Energy Innovation Institute
doi:10.1016/j.egypro.2015.07.425

into the interior of the building. The internal volume of the device is divided into quadrants, which allow fresh air to enter as well as stale air to escape, irrespective of the wind direction [1].

In hot regions such as the Middle East, there is a huge dependency on electricity to run mechanical ventilation systems. In these areas, using wind towers is a well-known technique. However, the cooling capabilities of wind towers, which depend only on the structural design, are limited. Therefore it is essential to cool the air in order to reduce the building heat load and to improve the thermal comfort of occupants during summer months [2]. Traditionally, wind towers have been integrated with evaporative cooling to reduce the air temperature. The induced hot air is sprayed with water or passed through cooling pads, evaporating the water in the process. Thus, the air becomes heavier and sinks to the bottom of the channel (Fig 1a). However, the addition of these cooling devices may reduce the airflow rate inside the channel. Another disadvantage of this configuration is the requirement of taller towers to have sufficient contact time. Furthermore, evaporative coolers use a substantial amount of water to run [2].

In this study, Heat Transfer Devices (HTD) were integrated into the passive terminal of a commercial wind tower to reduce the temperature of supply air. As shown in Fig 1b, the hot outdoor air (1) enters the wind tower through the louvers. The louvers are used to deflect the impact of weather and direct sunshine from entering the device. The airflow is driven downwards and passed through a series of HTD (2), which absorbs heat from the air and transfers it into a parallel cooling system (3). This is similar to the evaporative cooling system, but unlike the traditional method, the water is recirculated inside the cold sink and source (4). Adjustable dampers are mounted at the bottom of the unit to control the delivery rate (5). The cooled air is supplied to the room beneath the channel via ceiling diffusers (6).

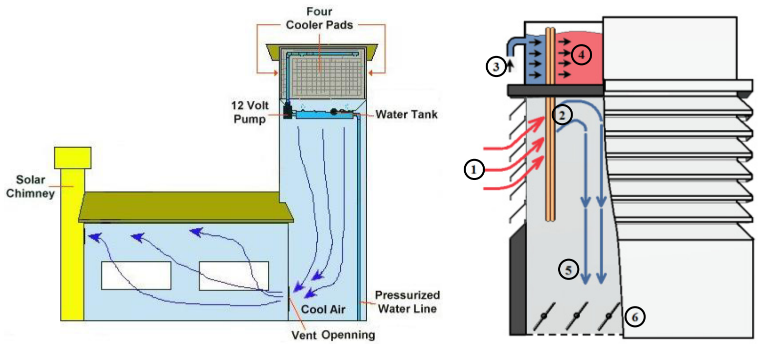


Fig. 1. (a) wind tower with evaporative cooling; (b) wind tower integrated with heat transfer devices (HTD)

A number of studies have assessed the natural ventilation performance of wind towers using Computational Fluid Dynamics (CFD) and wind tunnel analysis. CFD played a major role in development of wind towers due to the low computation resources required for the design and simulation of prototypes [1, 3]. The optimal design of various components of wind towers has been conducted including the damper and louver. Complimentary wind tunnel experiment of a smaller scale model validates the CFD simulation to improve the reliability [4, 5].

Several research projects investigated the integration of cooling techniques into wind towers for hot climates. Hughes *et al.* [1] highlighted the different cooling techniques integrated with wind tower systems to improve their thermal performance. Kalantar [6] evaluated the performance of a wind tower with evaporative cooling in the hot region of Yazd using CFD. Using the same CFD method, Calautit *et al.* [2] compared the thermal performance of an evaporative cooling and HTD assisted cooling for traditional wind towers. The study concluded that height of the wind tower was not a factor for the HTD integrated design, making it viable for commercial wind towers. The aim of this study was to investigate the performance of a commercial wind tower integrated with Heat Transfer Devices (HTD) using CFD and wind tunnel analysis. An experimental scale model was created using 3D printing. The scale model was tested in a closed-loop wind tunnel to validate the CFD model. Smoke visualisation experiment was conducted to analyse the pattern of the airflow in a room with the device.

2. Computational Fluid Dynamics (CFD) Setup

The basic assumptions for the steady-state CFD simulation included a three-dimensional, fully turbulent, and incompressible flow. The turbulent nature of the flow was modelled by the standard k–epsilon model. This technique is well established in the field of natural ventilation and wind tower research [1]. The CFD code used the Finite Volume Method (FVM) approach and employed the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) velocity-pressure coupling algorithm with the second order upwind discretisation. The governing equations are available in [7].

2.1. Physical domain and mesh generation

The CFD analysis was carried out using the ANSYS FLUENT software. The flow domain representation of the geometry of the wind tower and location of set boundary conditions are shown in Fig 2. A 5 x 5 x 10 m^3 enclosure was created to simulate the velocity of the outdoor wind. Furthermore, the model of the 1 x 1 m^2 wind tower was integrated to a test room located beneath it. The test room with an internal volume of 3 x 3 x 5 m^3 represented a small classroom of 15 occupants [8]. The diameter of the cylindrical HTD was 0.002 m and the horizontal spacing between the HTD was 0.05 m. The thermal performance of three HTD configurations were compared; configuration 1 (1 row), configuration 2 (2 rows) and configuration 3 (3 rows). The performance of the cold sink (parallel cooling system) was not investigated in this study, therefore it was modeled as a solid block in the numerical analysis.

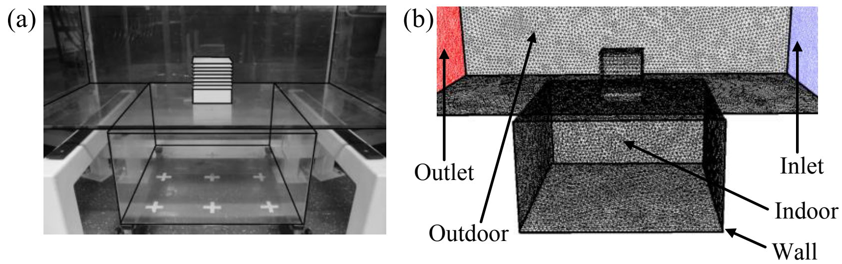


Fig. 2. (a) Closed-loop wind tunnel test set-up; (b) view of the computational mesh of the wind tower and test room model.

A non-uniform mesh was applied to the volumes of the computational model. The computational mesh of the wind tower and test room model is shown in Fig 2b. The mesh was refined around critical areas of interests (HTD, louvers, etc.) in the simulation [8]. Several meshes were generated to investigate the solution independency from the mesh. The numerical mesh was refined using the hp-grid adaptation method [9]. The mesh was refined (mesh sizes ranging from 3.8 to 7 million elements) until the posterior estimate error became insignificant between the number of elements and the posterior error indicator (supply velocity). The discretisation error was found to be the lowest at over 7 million elements.

2.2. Boundary conditions

Fig 2b shows the computational domain of the macro- and micro-climate volumes. A wall boundary condition with a set roughness height and roughness constant [5] was used to create a boundary between each volume. Boundary conditions for the numerical modeling of the flow were chosen to be the same as the conditions in the wind tunnel experiment [5]. CFD analysis was performed at various outdoor wind speeds (0 - 5 m/s). The pressure outlet was set to 0 Pa (atmospheric). For the analysis of the wind tower with HTD, the outdoor air temperature was set to 318 K to simulate a hot outdoor environment [2]. In order to cool the induced air, the HTD wall temperature was set to 293 K.

3. Wind Tunnel Experimental Set-up and Measurement Procedure

The experimental investigation was conducted in a closed-loop wind tunnel in the School of Civil Engineering of the University of Leeds [9, 10]. The size of the test section was 0.5 x 0.5 m^2 and 1m length (Fig 3a). A 1:10 scale model of the wind tower with HTD was used in the experimental study. The creation of an accurate scaled model was essential for the study, therefore, the wind tower was constructed using 3D printing. The scale of the model of the wind tower was selected to maintain, as close as possible, equality of model and prototype ratios of overall dimensions to the important meteorological lengths of the simulated wind. Cylindrical rods with an outer diameter of 0.002 m were used to model the HTD (Fig 3b). The scale model produced a maximum wind tunnel blockage of 4.8% [11, 12]. The model of the wind tower was connected to a 0.5 x 0.5 x 0.3 m^3 test room, mounted underneath the test section.

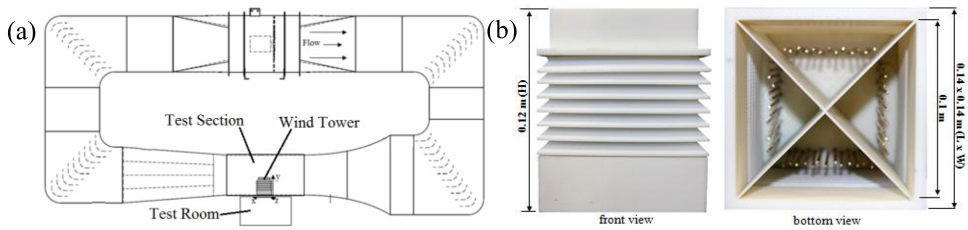


Fig. 3. (a) Closed-loop wind tunnel; (b) 3D printed model of a wind tower with cylindrical HTD (configuration 2).

In this study, the induced airflow into the test room was measured using a hot-wire anemometer positioned below the channels of the wind tower. The cross-sectional area of the supply and exhaust channels of the wind tower was divided into several portions (16 points). The hot wire probe (Testo 425) gave velocity measurements with uncertainty of ±1.0 % rdg. at speeds lower than 8 m/s. In order to recognise the flow pattern in the test room, smoke visualisation test was also carried out.

4. Results and Discussion

Fig 4a shows the velocity contours of the cross sectional plane in the test room model. As observed, the airflow passed around the wind tower, parts of it entered the channel and portion of it exited through the pressure outlet on the left wall. The flow entering the wind tower speeded up as it hit the cross-divider and streamed downward towards the test room. The air then spread sideways and upwards in all direction and part of it escaped through the wind tower outlet. At an outdoor wind speed of 3 m/s, the average velocity in the wind tower diffuser was 1.42 m/s while the average velocity in the room was measured 0.53 m/s. Fig 4b illustrates the temperature distribution inside the test room. The average temperature inside the microclimate was 311.8 K when the outdoor air temperature was at 318 K. A greater temperature reduction was obtained at the immediate downstream of the Heat Transfer Devices with a supply temperature value of 311.5 K, a reduction of 6.5 K. The results displayed that the HTD was able to reduce the temperature of the induced air stream while ensuring that the space was adequately ventilated.

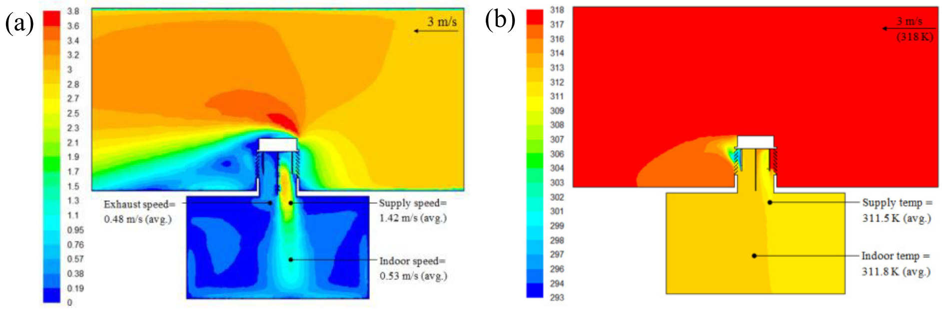


Fig. 4. (a) Velocity contour of a cross-sectional plane inside the test room; (b) temperature distribution inside the test room model

Fig 5a compares the supply airflow speed of a multi-directional wind tower with and without HTD. The airflow speed was reduced by 4 - 17 % following the addition of HTD. Fig 5b shows the effect of various outdoor wind speeds on the thermal performance of the HTD integrated wind tower. As observed, by increasing the number of HTD (from 1 row to 3 rows), the cooling performance of the system increased. Furthermore, significant reduction in temperature was observed at lower wind speeds.

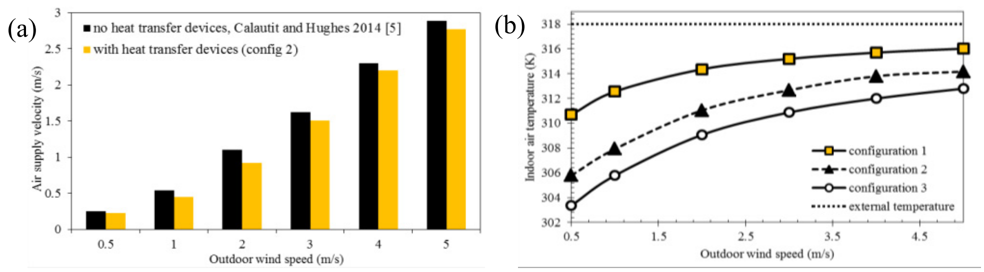


Fig. 5. (a) Effect of the integration of HTD on the air supply velocity (b) Thermal performance of different HTD configurations

Fig 6 shows a comparison between the experimental and CFD results of the velocity measurements. This comparison showed a low difference range and the trends were to be in a good agreement. Average error across the points was 7.5 %. Fig 7 displays a comparison between CFD and the experimental flow pattern inside the test room model. A similar flow pattern was observed; the airflow entering the wind tower was directed towards the floor of the room and spread outwards in all directions. As the airflow hit the bottom surface the air slowed down and flowed through the side walls, and parts of the air escaped through the exhaust quadrant.

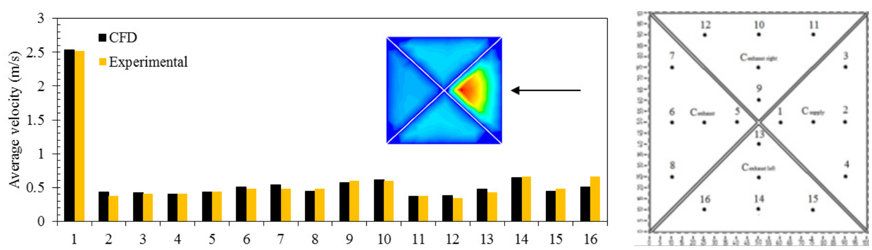


Fig. 6. Comparison between the experimental and CFD results of the airflow velocity measurement

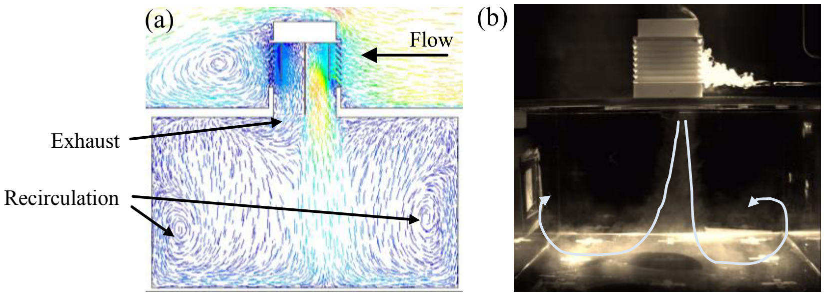


Fig. 7. (a) CFD flow visualisation; (b) Smoke testing in the wind tunnel

5. Conclusion

A numerical and experimental investigation was carried out to investigate the performance of a wind tower with Heat Transfer Devices (HTD). The numerical model was validated against the wind tunnel data and good correlation was achieved between both methods, as the error was below 10 %. The results indicated that the supply rate was reduced by 4 – 17 % following the integration of the HTD. The effect of different numbers of HTD on the thermal performance was investigated. The work highlighted the potential of integrating HTD into wind towers to reduce the temperature of the air induced into the ventilated space. Full scale field testing is required for further analysis and validation of the results.

References

[1] Hughes BR, Calautit JK, Ghani SA, The Development of Commercial Wind Towers for Natural Ventilation: a review. *Applied Energy* 2012;**92**:606-27.

[2] Calautit JK, Hughes BR, Ghani SA. A Numerical Investigation into the Feasibility of Integrating Green Building Technologies into Row Houses in the Middle East. *Architectural Science Review* 2013;**56**:279-296.

[3] Calautit JK, O'Connor D, Hughes BR. Determining the optimum spacing and arrangement for commercial wind towers for ventilation performance. *Building and Enviornment* 2014;**82**:274-287.

[4] Calautit JK, Hughes BR. Measurement and prediction of the indoor airflow in a room ventilated with a commercial wind tower. *Energy and Buildings* 2014;**84**:367-377.

[5] Calautit JK, Hughes BR. Wind tunnel and CFD study of the natural ventilation performance of a commercial multi-directional wind tower. *Building and Environment* 2014;**80**:71-83.

[6] Kalantar V. Numerical simulation of cooling performance of wind tower (Baud-Geer) in hot and arid region. *Renewable Energy* 2009;**34**:246–54.

[7] Calautit JK, Hughes BR. Measurement and prediction of the indoor airflow in a room ventilated with a commercial wind tower. *Energy and Buildings* 2014;**84**:367-77.

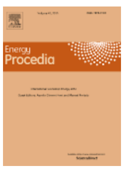
[8] Calautit JK, O'Connor D, Sofotasiou P, Hughes BR. CFD Simulation and Optimisation of a Low Energy Ventilation and Cooling System. *Computation* 2015;**3**:128-149.

[9] Calautit JK, Chaudhry HN, Hughes BR, Sim LF. A validated design methodology for a closed-loop subsonic wind tunnel. *Journal of Wind Engineering and Industrial Aerodynamics* 2014;**125**:180-94.

[10] Calautit JK, Hughes BR, Ghani SA. Numerical investigation of the integration of heat transfer devices into wind towers. *Chemical Engineering Transactions* 2013;**34**:43-48.

[11] Calautit JK, Hughes BR. Integration and application of passive cooling within a wind tower for hot climates, *HVAC&R Research* 2014;**20**:722-730.

[12] O'Connor D, Calautit JK, Hughes BR. A Study of Passive Ventilation Integrated with Heat Recovery, *Energy and Buildings* 2014;**82**:799-811.



Biography

Dr Calautit is a Research Associate in the University of Sheffield. He received his Mechanical Engineering degree from Heriot-Watt University and his PhD from the University of Leeds. His PhD study focused on numerical and experimental techniques to design and investigate the performance of a wind towers with passive cooling.