Energy and Buildings 84 (2014) 367–377

Contents lists available at ScienceDirect

Energy and Buildings

journal homepage: www.elsevier.com/locate/enbuild

Measurement and prediction of the indoor airflow in a room ventilated with a commercial wind tower

John Kaiser Calautit*, Ben Richard Hughes

School of Civil Engineering, University of Leeds, Leeds LS2 9JT, United Kingdom

ARTICLE INFO

Article history:
Received 26 May 2014
Received in revised form 20 July 2014
Accepted 15 August 2014
Available online 23 August 2014

Keywords:
Air change effectiveness
Air change rate
CFD
Indoor air quality
Mean age of air
Wind tower/catcher
Wind tunnel

ABSTRACT

This paper introduces a method for a full assessment of the internal air movement characteristics in a room ventilated with a commercial wind tower using scaled wind tunnel and computational fluid dynamics (CFD) analysis. An accurate geometrical representation of the experimental situation was recreated in the CFD simulation. The experimental investigation was carried out using a closed-loop low speed wind tunnel and the indoor air flow distribution was measured and compared with the CFD analysis. Good correlation between the numerical and experimental results was observed (10% average error). Flow visualisation was also conducted to further analyse the airflow structure within the room. The work assessed several ventilation parameters to describe the indoor airflow characteristics such as the mean age of air (MAA), air change rate, and air change effectiveness (ACE). The MAA was calculated using the steady-state method, by solving an additional partial differential equation describing the transport of the scalar "age of air". The numerical analysis of the MAA allowed to detect less or insufficiently ventilated areas. The CFD code could be useful for the calculation of the MAA and ACE, allowing reduction in time and cost in the evaluation of indoor air quality in buildings ventilated with wind tower or similar device.

1. Introduction

Buildings account for 40% of the world energy usage and are responsible for almost 40–50% of the global carbon emissions. Heating, ventilation and air-conditioning (HVAC) systems consumes more than 60% of total building energy utilisation [1,2]. The need to increase the energy efficiency of buildings, while providing comfortable and healthy indoor environment, has become a key challenge in the building sector. Great effort is being made to develop sustainable technologies that are capable of providing good indoor environmental quality (IAQ) and exploiting renewable energy resources [3,4].

A wind tower natural ventilation system is one of the green technologies that has the potential to tackle these challenges. In such a ventilation method, airflow through living spaces is achieved by using the natural driving forces of external wind and the buoyancy effect caused by air temperature differences between indoor and outdoor air. The wind tower technology has been around for many centuries in the Middle East and has been applied commercially in the UK over the last few years [5]. A wind tower can provide a significantly higher airflow rate than in an equivalent area ventilated by an open window [6]. Wind towers can also provide the benefit of night-time cooling without posing a security risk and daytime ventilation without relying upon opening windows [7].

For a commercial multi-directional wind tower, the channel is typically divided into four quadrants with the cross-divider running the full height of the channel. As shown in Fig. 1, the outdoor fresh air flow is directed into the indoor environment via the windward openings, while the stale air is extracted through other external openings side due to negative wind pressure. As the direction of wind changes so do the function of each of the quadrant in the wind tower. This allows the wind tower to capture and supply fresh air irrespective of the wind direction [4].

Due to the increasing emphasis on the development and use of wind tower devices, there is constant scope for accurately analysing their performance. A number of studies have assessed the performance of modern or commercial wind towers using experimental, numerical and analytical analysis. Most of the research work have primarily focused on the geometry of wind towers (dimension, shape, control damper, diffuser and louvre) and achieved ventilation rates. Elmualim and Awbi [8] compared the natural ventilation performance of modern wind towers with square and circular cross-sections using experimental and numerical analysis. The experimental set-up consisted of a full scale wind tower

* Corresponding author. Tel.: +0044 75544158981; fax: +0044 75544158981.
E-mail address: J.K.Calautit@leeds.ac.uk (J.K. Calautit).

http://dx.doi.org/10.1016/j.enbuild.2014.08.015
0378-7788/

Nomenclature

u	velocity magnitude (m/s)
X, Y, Z	Cartesian co-ordinates (m)
Re	Reynolds number
ρ	air density (kg/m^3)
μ	kinematic viscosity (m^2/s)
Q	volume flow rate (m^3/s)
g	gravitational acceleration (m/s^2)
A	cross-sectional area (m^2)
ΔP	total pressure loss (Pa)
P	pressure (Pa)
P_o	total pressure (Pa)
P_s	static pressure (Pa)
L	length (m)
W	width (m)
H	height (m)
t	time (s)

connected to a model test room. Due to the size of the wind towers (1.5 m × 0.5 m × 0.5 m and 1.5 m × 0.55 m diameter), the experimental investigation was carried out in an open test section (2 m × 2 m) wind tunnel.

The work concluded that the sharp edges of the square wind tower created a larger region of flow separation and high pressure difference across the device openings, making it more effective than the circular shape wind tower. Furthermore, the CFD code predicted a reasonable air flow rate compared with the wind tunnel result, despite the limitations of the test setup. Later, Elmualim [8] studied the effect of volume control dampers and diffusers on the ventilation rates of a commercial wind tower using the same method in the previous work [6]. The achieved results showed that the damper and diffuser reduced the air flow by approximately 20% at 3 m/s external wind velocity and 50% at 1 m/s.

Hughes and Ghani [9] analysed the effect of varying the wind tower louvre angle on the ventilation pefromance. Eight CFD models were generated with the louvre angle increased by 5° increments, for a range of 10–45°. Performing the same analysis using physical models would have required more time and effort for the design adjustments. Hence, CFD was used as the primary tool for the investigation. Results showed that a louvre angle of 35° provided the optimum ventilation performance.

Liu et al. [10] performed a similar study but evaluated the effect of varying the number and length of louvres on the ventilation rates instead. The results indicated that the flow rate of air induced into the wind tower increased with the number of louvre layers and the highest ventilation rate was achieved when the louvre length equated with the reference length. Like other earlier CFD-based studies on commercial wind towers, the numerical results were also validated against the experimental data of [6]. A good correlation between both methods was observed, although the numerical calculation domain in their CFD analysis did not accurately represent the experimental situation. An outdoor far field wind was considered instead of an open section wind tunnel.

Calautit et al. [11] used CFD to compare the performance of a row house model integrated with a traditional and commercial wind towers. The study used grid adaptation to verify the programming and computational operation of the computational model. Su et al. [5] measured the net flow rate of a commercial wind tower using an experimental set-up which included an inlet cone flow meter, balancing chamber and variable-speed blower fan. The measured data was compared with CFD results, and a good agreement between the two methods was achieved. Furthermore, CFD modelling of the wind tower was carried out to simulate the conditions similar to the situation of an outdoor far field wind. The calculated extract flow rate of the wind tower in a far field wind was approximately double that for the situation using a blower fan.

A few studies [6,12] have also assessed the performance of a commercial wind tower using analytical modelling. Jones and Kirby [12] proposed a semi-empirical approach in which a comprehensive analytical model was coupled with the data from the controlled experiments of [6] to quantify the flow rates. The semi-empirical model performed well against the CFD model of [9]. The latest research [13–16] on wind towers were mainly focused on traditional systems.

1.1. Indoor air distribution characteristics assessment

Fig. 2 shows the full assessment methodology flow chart for the evaluation of the indoor air distribution characteristics in buildings integrated with a commercial wind tower or similar ventilation device. The method uses indices of air change rate, mean age of air, and air change effectiveness. The wind tower geometry and the indoor domain were modelled using a commercially available CAD modelling software and then imported to ANSYS DesignModeller to generate a computational model for the CFD analysis. The same CAD model was used for the rapid prototyping of the wind tunnel model. The work combined the approach of rapid prototyping with wind tunnel testing in order to perform efficient wind tower experimentation in terms of cost-effectiveness and time management. Additionally, the creation of an accurate scaled wind tunnel prototype was essential for the experimental study. The wind tower geometry featured a variety of unconventional and complex parts such as the external louvers and cross-dividers. A model of this

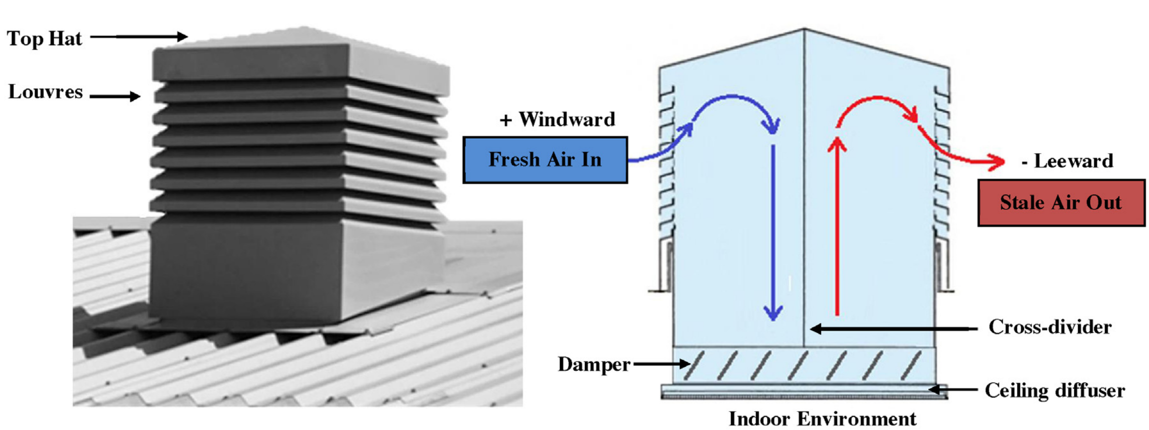

Fig. 1. Schematic of a commercial multi-directional wind tower system.