Building and Environment bae_3656_gr14_3c.tif - xxx (2014) 1–13

Contents lists available at ScienceDirect

Building and Environment

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

Evaporative cooling by water spray systems: CFD simulation, experimental validation and sensitivity analysis

H. Montazeri [a,*], B. Blocken [a,b], J.L.M. Hensen [a]

[a] *Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands*

[b] *Building Physics Section, Department of Civil Engineering, Leuven University, Kasteelpark Arenberg 40 – Bus 2447, 3001 Heverlee, Belgium*

A R T I C L E I N F O

Article history:
Received 28 January 2014
Received in revised form
21 March 2014
Accepted 23 March 2014

Keywords:
Computational Fluid Dynamics (CFD)
Sensitivity study
Validation
Urban physics
Evaporation
Hollow-cone spray

A B S T R A C T

Evaporative cooling by water spray is increasingly used as an efficient and environmentally-friendly approach to enhance thermal comfort in built environments. The complex two-phase flow in a water spray system is influenced by many factors such as continuous phase velocity, temperature and relative humidity patterns, droplet characteristics and continuous phase–droplet and droplet–droplet interactions. Computational Fluid Dynamics (CFD) can be a valuable tool for assessing the potential and performance of evaporative cooling by water spray systems in outdoor and indoor urban environments. This paper presents a systematic evaluation of the Lagrangian–Eulerian approach for evaporative cooling provided by the use of a water spray system with a hollow-cone nozzle configuration. The evaluation is based on grid-sensitivity analysis and validated using wind-tunnel measurements. This paper also presents a sensitivity analysis focused on the impact of the turbulence model for the continuous phase, the drag coefficient model, the number of particle streams for the discrete phase and the nozzle spray angle. The results show that CFD simulation of evaporation by the Lagrangian–Eulerian (3D steady RANS) approach, in spite of its limitations, can accurately predict the evaporation process, with local deviations from the wind-tunnel measurements within 10% for dry bulb temperature, 5% for wet bulb temperature and 7% for the specific enthalpy. The average deviations for all three variables are less than 3% in absolute values. The results of this paper are intended to support future CFD studies of evaporative cooling by water spray systems in outdoor and indoor urban environments.

1. Introduction

As a result of climate change more buildings will be exposed to milder winters and hotter summers [1,2]. Research indicates that a major European heat wave, such as that of 2003, will occur more frequently in the future [3] and it could become a common event by 2040 [4]. Increased heat waves and heat stress are likely to cause increased illness and death as occurred in the hot summers of 2003 and 2006 [5]. These problems are aggravated by the urban heat island effect (UHI) [6,7]. The term urban heat island is used for urban areas which exhibit higher temperatures than their rural surroundings. Therefore, adaptation strategies such as evaporative cooling need to be evaluated and implemented to reduce heat stress in the outdoor and indoor urban environment.

Several research organizations and consortia have initiated projects regarding climate change adaptation in cities as the Intergovernmental Panel on Climate Change (IPCC) has expressed the importance of adaptation measures [8]. Climate Proof Cities (CPC) is one of these research consortia investigating the climate vulnerability of urban areas and the development of climate change adaptation measures [9]. The consortium consists of universities, research institutes, policy makers and city officials to perform both basic and applied science, the latter being an integrated and thorough analysis for several locations in the Netherlands.

Evaporative cooling by water spray is increasingly used as an efficient and environmentally-friendly approach to enhance thermal comfort in urban environments (outdoors and indoors) (e.g. Refs. [10,11]). In a water spray system, a cloud of very fine water droplets is produced using atomization nozzles. It enhances mixing and increases the contact surface area between the air stream and the water droplets resulting in a higher rate of evaporation yielding greater cooling of the ambient air.

* Corresponding author. Tel.: +31 (0) 40 247 4374; fax: +31 (0) 40 243 8595.
E-mail address: h.montazeri@tue.nl (H. Montazeri).

http://dx.doi.org/10.1016/j.buildenv.2014.03.022
0360-1323/

For assessing the potential and performance of evaporative cooling by water spray systems in outdoor and indoor environments, different methods can be used: (i) full-scale measurements, (ii) wind-tunnel measurements, and (iii) numerical simulation with Computational Fluid Dynamics (CFD). Full-scale measurements offer the advantage that the real situation is studied and the full complexity of the problem is taken into account. However, full-scale measurements are usually only performed in a limited number of points in space. In addition, there is no or limited control over the boundary conditions. Reduced-scale wind-tunnel measurements allow a strong degree of control over the boundary conditions, however at the expense of – sometimes incompatible – similarity requirements. Furthermore, wind-tunnel measurements are usually also only performed in a limited set of points in space. CFD on the other hand provides whole-flow field data, i.e. data on the relevant parameters in all points of the computational domain [12–15]. Unlike wind-tunnel testing, CFD does not suffer from potentially incompatible similarity requirements because simulations can be conducted at full-scale. CFD simulations easily allow parametric studies to evaluate alternative design configurations, especially when the different configurations are all a priori embedded within the same computational domain and grid (see e.g. Ref. [14]). However, the accuracy and reliability of CFD are of concern, and verification and validation studies are imperative (e.g. Refs. [13,16–19]). CFD is increasingly used to study a wide range of atmospheric and environmental processes (e.g. Refs. [13,20–22]). Examples include pedestrian wind comfort and wind safety around buildings [23–26], natural ventilation of buildings [12,14,27–35], air pollutant dispersion [36–39], wind-driven rain [40] and convective heat transfer [41,42]. CFD has also been used on several occasions in the past to evaluate the performance of spray systems for different applications (e.g. Refs. [43–48]). In the vast majority of these studies the Lagrangian–Eulerian (LE) approach has been used in which the continuous phase (air in this study) is represented in an Eulerian reference frame while the discrete phase (water droplets in this study) is represented in a Lagrangian reference frame. The numerical implementation of this approach was introduced and applied by O'Rourke [49,50] and Dukowicz [51] for internal combustions engine applications. However, it has been developed and used for many other applications including evaporating spray systems. A comprehensive review of the LE method including its advantages over the Eulerian–Eulerian (EE) method, modelling issues and numerical implementation is provided by Subramaniam [52].

To the best of our knowledge, a detailed evaluation of the LE approach for predicting evaporative cooling has not yet been performed. This paper presents a systematic evaluation of the LE approach for predicting evaporative cooling provided by a water spray system with a hollow-cone nozzle configuration. The evaluation is based on grid-sensitivity analysis and on validation with wind-tunnel measurements by Sureshkumar et al. [53]. This paper also presents a sensitivity analysis focused on the impact of the turbulence model for the continuous phase and the number of particle streams for the discrete phase. In addition, the important impact of nozzle spray angle is demonstrated.

The results of this paper are intended to support future CFD studies of evaporative cooling by water spray systems in outside and inside urban environments.

2. Wind-tunnel experiments

In the experiments by Sureshkumar et al. [53] the evaporative cooling performance of a hollow-cone nozzle spray system was investigated. The experiments were performed in an open-circuit wind-tunnel with a uniform mean wind speed. The test section of the wind-tunnel was 1.9 m long with a cross section of 0.585 m × 0.585 m (Fig. 1(a)). The dry bulb temperature (DBT) and wet bulb temperature (WBT) variations of the air stream between the inlet plane of the test section, where the spray nozzle was installed, and its outlet plane were measured for different air flow conditions and spray characteristics.

The inlet air DBT and WBT were measured by using two thermocouples placed upstream of the nozzle. Electric heaters were employed upstream of the tunnel blower to reduce the impacts of the background air temperature fluctuations. These fluctuations were limited within ±0.3 °C during each set of experiments. The outlet air DBT and WBT were measured using 18 thermocouples installed at the tunnel outlet (Fig. 1(b)). A thermal probe installed upstream of the spray nozzle was used to measure the air stream velocity. The maximum experimental uncertainty for the mean velocity was estimated to be less than ±0.05 m/s for air velocity up to 2 m/s and ±0.2 m/s for air velocity between 2 and 4 m/s.

A drift eliminator with z-shaped plates was placed close to the tunnel outlet to collect the remaining water droplets in the air flow to avoid wetting of the thermocouples. The sump water was collected in a separate tank to avoid mixing of supply and sump water in order to keep the water inlet temperature constant during each set of experiments. The inlet and outlet water temperatures were measured using two thermocouples upstream of the nozzle and downstream of the drift eliminator, respectively. Water pressure was also measured by a pressure gauge upstream of the nozzle.

In order to evaluate the impact of nozzle characteristics on cooling performance of the spray system, four identical nozzles but with different discharge openings of 3, 4, 5 and 5.5 mm were used. Each nozzle was installed in the middle of the test section (Fig. 1(a)) and designed in a way that the exiting water forms a hollow-cone sheet disintegrating into droplets. The droplet diameter distribution was determined using an image-analysing technique. The uncertainty of this technique for the mean droplet size was estimated to be ±22%. The half-cone angle was measured in still air and reported as a function of nozzle diameter, water pressure and background wind speed. As the exact value of the half-cone angle was not reported by Sureshkumar et al. [53], in Section 4.3.3 the influence of this parameter will be investigated. No correlations between droplet size and velocity were given by Sureshkumar et al. [53].

The experiments were conducted in two periods of time to resemble different ambient (i.e. in the wind-tunnel) conditions. The first set of experiments, which is used in the present study, were carried out in April–June representing a hot and dry climate condition in which DBT and humidity ratio (RH) ranged between 35 and 45 °C, and 10 and 35%, respectively. The second set was carried out in July–September representing hot–humid conditions with ambient DBT between 25 and 40 °C and RH between 30% and 90%. The inlet water temperature varied between 33 and 36 °C for the two ambient conditions. For each ambient condition, experiments were conducted for 36 cases; four different nozzle discharge diameters (i.e. 3, 4, 5 and 5.5 mm), three inlet nozzle gauge pressures (1, 2 and 3 bar) and three background wind speeds (1, 2 and 3 m/s). The three cases with a nozzle discharge diameter of 4 mm and a gauge pressure of 3 bar were taken since droplet size distribution data were also available for these cases.

3. CFD simulation

In this study the commercial software ANSYS/Fluent 12.1 [54] is used in which the Lagrangian–Eulerian approach is implemented to simulate multi-phase flows in sprays and atomizers.