Energy and Buildings 76 (2014) 565–571

Contents lists available at ScienceDirect

Energy and Buildings

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

Effects of different inlet vent positions on the uniformity of humidity inside a building chamber

Liang Pu[a,b,*], Yanzhong Li[a], Fu Xiao[b], Zhenjun Ma[c], Di Qi[a], Suping Shen[a]

[a] Institute of Refrigeration and Cryogenic Engineering, Xi'an Jiaotong University, Xi'an 710049, China
[b] Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong SAR, China
[c] Sustainable Buildings Research Centre (SBRC), Faculty of Engineering, University of Wollongong, Australia

ARTICLE INFO

Article history:
Received 10 May 2013
Received in revised form 16 January 2014
Accepted 11 March 2014
Available online 25 March 2014

Keywords:
Humidity distribution
Building room
Experimental study
CFD simulation

ABSTRACT

Inlet conditions, including inlet size and location as well as inlet temperature and humidity, have significant effects on the humidity distribution in building room, testing chamber and built environment. This paper presented the investigation of the changes of inlet air temperature, humidity and inlet vent positions on the humidity distribution in the environmental chamber through experimental approach and simulation method. The average relative humidity (RH) and humidity uniformity inside the chamber with different inlet vent positions were investigated. The results show that non-uniformity of the humidity distribution decreases with the increase of the inlet RH value under low inlet temperature conditions while it increases with the increase of the inlet RH value under high inlet temperature conditions. The humidity uniformity is not very good under the lower inlet vent positions. Comparison studies on the humidity uniformity between experimental tests and numerical simulations show that the simulation results agree well with the experimental test results.

1. Introduction

Environmental testing chamber has been widely used to assess the performances of various electronics, electricity equipment and new materials and simulate built environment for building room. The temperature field and humidity distribution are two important performance indexes often used to evaluate the performance of environmental testing chambers.

A number of experimental studies have been conducted to study the air flow field, temperature distribution and uniformity, and the thermal comfort problems inside the finite room, like car rooms of different transports and other built environment. Zhang et al. [1] experimentally studied the air flow and temperature fields inside a passenger compartment for thermal comfort improvements and energy savings. Fredriksson et al. [2] established a mock of an office room and conducted the experimental tests of the airflow field and pattern in the room. The results showed that the air flow from the chilled beam has similar behavior to a two-dimensional plume but exhibits strong oscillation both in the sideways and along the chilled beam. Ding et al. [3] studied the flow fields, temperature field and uniformity inside a chamber of two kinds of refrigerators by both computational fluid dynamics (CFD) simulation and experimental method. Zingano [4] carried out the experimental studies and discussed the importance of the humidity to the thermal comfort temperatures. An indirect method for determining the midpoint of the thermal comfort temperature through analysing preferred bath water temperatures was proposed. Because contaminant transport has a strong effect on disease spreading in the aircraft cabin environment and further influences the comfort and health of cabin members. Li et al. [5] experimentally investigated the complex contaminant distribution fields of aircraft cabin. The experimental measurement was conducted in a part of the economy-class cabin without heating manikins and occupied with heating manikins, respectively.

Humidity distribution is another important parameter that should be considered in order to study the influence of the temperature and humidity uniformities and energy consumption in actual indoor and built environment projects. The CFD method and numerical simulation method have been applied to study the fluid movement, fluid fields, temperature fields and pollution control. Yang et al. [6] investigated the effects of wind-break wall configurations on the thermal-flow performance of air-cooled condensers in a power plant. The results showed that the thermal-flow performance can be improved through the extensions of the inner and

* Corresponding author at: Institute of Refrigeration and Cryogenic Engineering, Xi'an Jiaotong University, Xi'an 710049, China.
E-mail addresses: puliang@mail.xjtu.edu.cn (L. Pu), yzli-epe@mail.xjtu.edu.cn (Y. Li).

http://dx.doi.org/10.1016/j.enbuild.2014.03.022
0378-7788/

Nomenclature

φ	relative humidity at the measuring point (%)
$\bar{\varphi}$	average relative humidity (%)
n	number of the measuring points
y	non-uniformity of relative humidity

outer walkways and elevation of the wind-break wall. In order to investigate a special roofing system, Manzan and Saro [7] carried out numerical simulations of flow field, temperature field and distribution of water vapor within the duct for thermal performance evaluation purposes. To choose the best project, several different inlet and exhaust locations and different design of the vents were investigated by Khan et al. [8]. The room concentration patterns were calculated through CFD simulations for various inlet and exhaust positions and other conditions. In order to supply fresh and humidified air to each person on an air plane, Zitek et al. [9] presented a novel design method and system to simulate personalized and humidified fresh air for the airliner passengers. In this system, each seat in the cabin had a separate air supply system and exhaust system. Yang et al. [10] established the mathematical model of a hyperbolic natural draft wet-cooling tower and carried out the simulation tests to optimize the cooling tower and evaluate its operational efficiency. Bojic et al. [11] used a CFD method to simulate and predict the temperature and flow fields inside the recessed spaces that differ in heights and condenser-unit locations. The results showed that a powerful rising hot air stream and one vortex pair were formed in the space near to its corner wall. A further study of accurate thermal boundary conditions on the simulation accuracy of the air field inside an indoor room was carried out by Chow and Holdo [12]. The variation of radiative absorptivity on the vertical temperature profile was studied to determine the contribution of thermal re-distribution by surface heat radiation.

Moreover, there are some studies using both experimental method and CFD simulation to compare the flow and temperature fields inside the finite room. Antonio et al. [13] measured the temperature values in a commercial household refrigerator with a couple of thermocouples located at different points, and then compared the measurement results with that from two different simulation methods: Fluent code and Artificial Neural Network method. Xie et al. [14] proposed a two-dimensional mathematical model for a minitype constructional cold store and developed a related simulation program with SIMPLE algorithm and crisscross girding technique. Rohdin et al. [15] also compared the measurement results with simulation results of the flow field through different turbulence models. The performance of CFD for industrial ventilation system simulation and two different supply principles in a contaminant-intensive process with temperature and density stratification were evaluated. Liu et al. [16] studied a kind of horizontal airflow pattern and the airflow field in an operating room and assessed the effectiveness and performance of horizontal unidirectional airflow to control infectious airborne particles by onsite test and CFD simulation as well.

Some researchers have investigated the heat and moisture transfer between the air and water and the relations between the temperature and humidity under certain conditions. For instance, Liu et al. [17] proposed a new method to investigate the heat and moisture transfer between the free water surface and surrounding air through experimental tests and CFD simulation. Evaporative cooling of air by water sprays is considered as an energy efficient and environmentally friendly technology for providing comfortable environment in hot and arid areas. Sureshkumar et al. [18,19] discussed heat and mass transfer processes between a water spray and ambient air under different conditions by experimental and simulation studies. However, there are few studies focusing on the main factors that influence the humidity uniformity inside the test chamber or built environment.

In this study, a series of experimental tests under different inlet vent position conditions, inlet temperatures and inlet relative humidity (RH) are performed to investigate the characteristics of temperature and humidity distributions inside the chamber. The influence of the inlet temperature, humidity and inlet positions on the temperature field and humidity distribution in the chamber is analyzed in detail. The variation of the average RH values under different inlet vent positions and humidity uniformities of RH inside the chamber are also studied. Comparison studies between the experimental results and CFD simulation results are carried out as well.

2. Description of the experimental system

The experimental system used in this study mainly consists of an air-handling system, a chamber and a data acquisition system. The dimensions of the testing chamber and the effective working zone are 1.0 m (L) × 1.0 m (W) × 1.5 m (H) and 0.6 m × 0.6 m × 1.1 m, respectively. The structure of the chamber was made of Polyurethane set with stainless steel. There is no absorption of the water droplets happened on the internal surface. The treated air can be supplied from the upper, middle and lower inlet vents (0.3 m × 0.1 m) into the chamber independently, and exhausts to the air handling system that was located at a lower outlet vent (0.5 m × 0.1 m).

In this experimental system, ambient air is firstly processed by the air-handling unit and then supplied to the testing chamber through different inlet vents. There are a total of 15 sets of humidity sensors instrumented in the experimental facility, including the inlet vent and outlet vent. The other 13 sets of sensors, which are numbered from 1 to 13 (see Fig. 1), were installed in the working zone [20,21]. According to their physical locations, these measurement points can be divided into three layers, i.e. upper layer, middle layer and lower layer distributed in the three horizontal sections inside the chamber. The time-dependent temperature and humidity values were measured in the experimental tests. The air velocity is measured by a hot-wire anemometer at the inlet of the chamber. All measurements are transferred into the data acquisition system. The accuracies of the temperature and humidity sensors were 0.5 °C and 1.5%, respectively.

A total of 12 experimental tests were carried out under different conditions of three inlet vents, with a variety of inlet temperature and humidity, respectively. At each inlet vent position (i.e. lower, middle and upper), 12 groups of experimental tests with three different inlet temperature and four humidity conditions were carried out. During the experimental tests, the data was collected when the temperature and humidity distributions within the chamber reached to a steady state. The test conditions are summarized as below.

- The pressure was maintained at the atmosphere pressure;
- The average ambient temperature during the test period was in the range of 26–30 °C;
- The thermal balance tests and calculations were performed firstly and the suitable thermal boundary conditions were then provided to the latter CFD simulations.

3. CFD algorithms and simulation method

Ultrasonic humidifier was used as humidifying way during experimental study, and simulation study of distribution of water mist produced from ultrasonic humidifier and humidity field were