Building and Environment 77 (2014) 29–39

Contents lists available at ScienceDirect

Building and Environment

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

Experimental investigation on transient natural ventilation driven by thermal buoyancy

Xiufeng Yang [a,b], Ke Zhong [a], Hui Zhu [a], Yanming Kang [a,*]

[a] School of Environmental Science and Engineering, Donghua University, Shanghai 201620, China
[b] School of Energy and Power Engineering, Yangzhou University, Yangzhou 225127, China

ARTICLE INFO

Article history:
Received 8 December 2013
Received in revised form
1 March 2014
Accepted 16 March 2014
Available online 22 March 2014

Keywords:
Full-scale experiment
Transient natural ventilation
Thermal buoyancy
Initial temperature difference
Vent characteristic

ABSTRACT

Full-scale experiments were carried out to explore the effects of the initial temperature difference between the interior and exterior and the vent characteristics on the transient development of natural ventilation driven by thermal buoyancy. Air temperature and tracer gas concentration in a test chamber were measured and the experimental results were compared with the theoretical predictions. It was found that the initial temperature difference has a large influence on the thermal stratification, the concentration distribution of tracer gas, the flow rate and the removal rate of tracer gas. The time taken to reach steady-state ventilation is shorter for a larger vent if the initial indoor temperature is greater than or equal to the outdoor temperature. For a pre-cooled room, the time taken for the ventilation to transform the airflow direction is shorter for a larger vent. Increasing the vent area would yield a greater flow rate and thus improve the efficiency of gas removal. Experimental results also show that the vent shape has little impact on the flow rate.

1. Introduction

In spaces containing heat sources, the thermal buoyancy derived from the air density differences can drive a ventilation flow through the room, which is an energy efficient means of delivering fresh air to the occupants [1,2]. The ventilation flow would reach the steady state after a period of development, the timescale of which may be comparable to the time of occupancy [3–5].

In contrast to steady-state natural ventilation, theoretical studies on unsteady natural ventilation are relatively limited due to the complexity of turbulent plume dynamics and the time-dependent flow patterns. Most of previous work reported in the literature assumed two-layer or multi-layer thermal stratification within the space and proposed some simplified mathematical models for transient buoyancy-driven ventilation. Fitzgerald and Woods [4] examined a naturally-ventilated room with a distributed heat source and accounted for the effect of the penetrative entrainment across two layers. They illustrated that three different ventilation modes may develop depending on the initial temperature of the room relative to the final equilibrium temperature and the exterior temperature. In a follow-up paper [5], they investigated the transient ventilation of a pre-heated room with a point heat source at the base of the space. It was noted that four different transient flow patterns would develop for different cases. Kaye and Hunt [6] and Yang et al. [7] presented different models for the transient buoyancy flow driven by the localized source of heat when the interior is initially at the same temperature as the exterior. The main difference between their models is that the buoyant layer was regarded as composed of a middle layer and a near-ceiling layer in Ref. [7], rather than being well-mixed in Ref. [6]. Bower et al. [8] developed quantitative models to examine the transient evolution of an emptying filling box when the source buoyancy flux is changed instantaneously. They found that two qualitatively different transient flows can occur, depending on whether the source heat flux increases or decreases. Hunt and Coffey [9] investigated the transient flows when the interior is either initially stably stratified in two homogeneous layers or is filled entirely with a dense fluid. They found that four distinct patterns of flow are possible, which is determined by three controlling geometrical parameters. Sandbach and Lane-Serff [10] proposed a modified filling-box model to calculate the temporal stratification. They modeled the upper layer as a series of layers of different buoyancy and integrated their equations numerically.

Computational fluid dynamics (CFD) has also been applied to model the buoyancy-driven natural ventilation in buildings. Jiang and Chen [11] studied single-sided natural ventilation driven by buoyancy numerically and experimentally for a room with an open

* Corresponding author. Tel.: +86 21 67792554; fax: +86 21 67792522.
E-mail address: ymkang@dhu.edu.cn (Y. Kang).

http://dx.doi.org/10.1016/j.buildenv.2014.03.013
0360-1323/

door or an open window. The experimental data were used to validate two CFD models: Reynolds averaged Navier–Stokes equation modeling and large eddy simulation. Kaye et al. [12] modeled a time-dependent buoyancy-driven natural ventilation to benchmark CFD models against previously published experimental results and theoretical models in Ref. [6].

Indoor concentration of gaseous contaminants would vary with time during the buoyancy-driven natural ventilation if the contaminant concentration outdoors differs from that of the interior. Hunt and Kaye [13] extended the transient model of [6] to consider a contaminant flushing problem in a naturally-ventilated space containing isolated heat sources. It was noted that the contaminant concentration will decay exponentially with time and the decay rate is a function of the room geometry and the strength, number and geometry of the heat sources. The contaminant flushing work of [13] was then extended to consider the case of uniformly distributed heat gains in Ref. [14]. It was demonstrated that for large vent areas a uniform heat distribution provides the fastest flushing. However, for smaller vent areas, localized heat sources produce the fastest flushing. Bolster and Linden [15] were concerned with the transport of passive contaminants in displacement ventilation systems. They did not model the buoyant layer as a well-mixed region, which was in contrast to [13] and [14]. Based on the work of [13] and [14], Zhong et al. [16] focused on the effects of the incoming air concentration and indoor pollutant source on the pollutant concentration in a displacement natural ventilation room. Previous studies mentioned above examined the decay of contaminant concentration indoors after buoyancy-driven natural ventilation reaches the steady state. Unlike above work, Yang et al. [17] investigated the pollutant flushing of transient natural ventilation produced by a localized heat source at the floor of a room in which the initial temperature is equal to the exterior temperature.

Simplified mathematical models mentioned above examined the simple case of a well-insulated room with vents at floor and ceiling levels. Both vents were assumed to have a negligible height and the influence of the vent shape on the transient ventilation was not included. Furthermore, the initial temperature in a naturally-ventilated room may be equal to, higher than or lower than the external ambient temperature. The last situation is very generic and may correspond to the transient ventilation of a cool and initially closed space. However, few theoretical studies on this type of ventilation have been carried out so far. It is, therefore, necessary to evaluate and understand transient natural ventilation in buildings by means of an experimental method.

In this study, full-scale experiments are conducted to measure the time variation of air temperature and tracer gas concentration in a test chamber during the transient natural ventilation driven by thermal buoyancy in some typical cases. The comparison between the experimental results and the predictions of the theoretical models in the literature has been carried out, and the causes of the differences have been analyzed. The effects of the initial temperature difference between the interior and exterior and the vent characteristics on indoor temperature distribution, transient flow rate and the contaminant removal are then investigated in detail.

2. Experimental measurements

2.1. Experimental system

Fig. 1 shows a schematic view of the experimental system used in this work. The experiments were performed in a test chamber with length, width and height of 3.6 m × 3.0 m × 2.6 m, placed in a large laboratory. The wall, floor and roof of the test chamber were made of stainless steel plates which were hollow

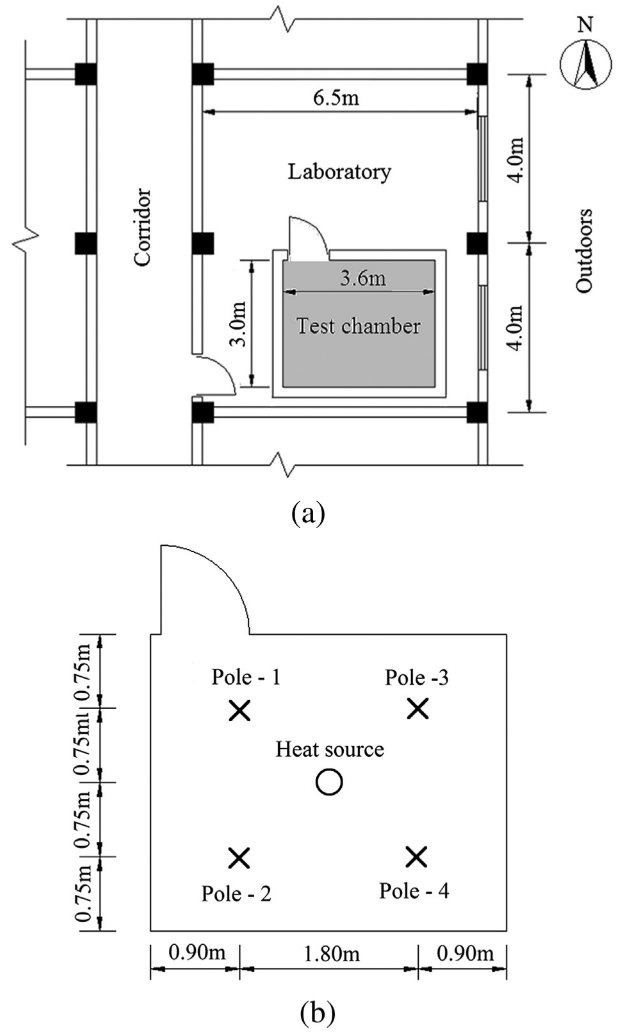

Fig. 1. Schematic view of the experimental system: (a) configuration of the laboratory, (b) heat source and measuring poles in the test chamber.

and filled with foaming polyurethane. Seeing Fig. 2, a vent of 0.57 m (width) × 0.074 m (height) was above the chamber door. The other vent was located on the lower part of the chamber door and its dimensions were changeable. An incandescent bulb of 100 W was placed at the center of the chamber floor to generate buoyancy forces. The bulb was enclosed in a round box with insulating materials to reduce the thermal radiation between the bulb and the interior wall of the chamber. The test chamber and the laboratory space were used to simulate an “indoor” environment and a windless “outdoor” environment, respectively.

The initial indoor temperature can be adjusted to the desired value by an air-conditioning system. Two ceiling fans were designed to mix indoor air and to achieve a uniform interior temperature and contaminant distribution before the measurement. Four vertical poles, each with seven measuring points (Fig. 2(c)), were located on the chamber's floor. The lowest measuring point on every pole was positioned 0.4 m above the floor, and the vertical spacing between the measuring points was 0.3 m. Air temperature and tracer gas concentration were measured simultaneously at all measuring points. The experimental data were recorded every minute throughout all experiments. The layout of the heat source (denoted by “O”) and measuring poles (denoted by “×”) is shown in Fig. 1(b).