International Journal of Heat and Mass Transfer 84 (2015) 521–528

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

A modified turbulent mixing model with the consideration of heat transfer between hot buoyant plume and sidewalls in a closed stairwell

L.J. Li [a,b,d], J. Ji [a,c,*], W.X. Shi [a], Z.H. Gao [a], J.H. Sun [a], J.P. Zhu [a]

[a] State Key Laboratory of Fire Science, University of Science and Technology of China, Hefei 230026, China
[b] School of Civil Engineering and Architecture, Chongqing Jiaotong University, Chongqing 400074, China
[c] Institute of Advanced Technology, University of Science and Technology of China, Hefei 230088, China
[d] State Key Laboratory Cultivation Base for Bridge and Tunnel Engineering in Mountain Areas, Chongqing Jiaotong University, Chongqing 40074, China

ARTICLE INFO

Article history:
Received 19 February 2014
Received in revised form 7 November 2014
Accepted 2 January 2015
Available online 24 January 2015

Keywords:
Closed stairwell
Buoyant plume
Turbulent mixing process

ABSTRACT

Previous studies on the turbulent mixing process in closed shafts did not take into account the heat transfer from the hot buoyant plume to the boundaries such as walls. In this paper, a modified theoretical model predicting the one-dimensional turbulent mixing process in vertical shafts is proposed with the heat transfer from the hot buoyant plume to the boundaries being involved. A set of small scale experiments were conducted to validate this model. A propane gas burner was used as the heat source to provide steady heat release rate. The temperature rise in the stairwell exponentially decreases with the height. The comparison of the model predicted and experimental results show that the Cooper's turbulent mixing model without the boundary effect gives higher predictions to the temperatures in the stairwell compared to the experimental data whereas the current modified model leads to more comparable results. Therefore the heat transfer process between the plume and the boundaries should be included in any modeling for the case of buoyant plume rising in closed shafts.

1. Introduction

The occupant safety and energy efficiency of high rise buildings have attracted lots of attention of researchers owing to many skyscrapers have been constructed with the economic development in the world [1–4]. Vertical shafts in modern high-rise buildings such as stairwells, air ducts and pipe well might become an effective path for heat and mass transfer from each floor. A number of researchers have studied buoyant plume movement and heat transfer in the high rise buildings [5–10]. There are two mechanisms mainly responsible for vertical motion of buoyant plume in a vertical shaft [11]: the turbulent mixing process because of the Rayleigh–Taylor instability [11] and the stack effect due to the pressure differences caused by the temperature difference inside and outside the building. Many studies have been carried out for the stack effect [12–23], including the buoyant plume movement mechanism [12–16], the influence of stack effect on buoyant plume flow [17–20], the rise-time of buoyant plumes [21,22] and the temperature distribution in staircases and shafts [23].

On the other hand, when buoyant plume enters into a shaft where the upper air is colder and denser, the stratification of buoyant plume and air are unstable, which leads to a rapid mixing of the two gases [24,25]. This process is identified as the turbulent mixing process which is relevant to the Rayleigh–Taylor mixing process that has been studied extensively. In a closed shaft the buoyant plume rises with the turbulent mixing simultaneously. In particular, the effect of buoyancy on turbulent mixing is important for fire-driven plume rising in vertical shaft. A few studies have been focused on the turbulent mixing. Zukoski et al. [11,24] studied the turbulent mixing process by using the salt water experiments and heavy-gas/light-gas experiments and developed a model to predict the rise time based on the density difference of fluids. In the case of turbulent mixing process in a high vertical shaft, Cooper [26] modified Zukoski's model by introducing a universal constant K to model. In the previous studies [11,24,26], the temperature of water or gas was the same as the ambient temperature hence there was no heat transfer between water or gas and walls. In the conclusion of Cooper's research [26], Cooper pointed out that his model formulation should be further developed by consider of heat transfer from buoyant plume to the wall and confirmed with data from experiments based on hot-air/cold-air systems.

In the fire research, buoyant plume heat transfer and turbulence flow in compartment were widely studied by field models [27–40].

---

[*] Corresponding author at: State Key Laboratory of Fire Science, University of Science and Technology of China, Hefei 230026, China.

E-mail address: jijie232@ustc.edu.cn (J. Ji).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.01.012
0017-9310/

One of the earliest studies by field models to predict fire behavior in compartments was conducted by Markatos et al. [27–30]. Due to the absence of detailed chemistry, the fire source is represented by volumetric heat source and transport phenomena caused by fire is modeled as buoyancy induced turbulent flow. Pourya Forooghi et al. [31] numerically investigated the effect of buoyancy to heat transfer in inclined pipes. In this study, the mechanism that leads to impairment of turbulence production was discussed. Phillips [32] simulated unstratified turbulent natural convection in a vertical slot at a high Grashof numbers and showed the turbulence originates from shear layer at the channel center. El-Samni et al. [33] studied the turbulent convection in vertical channel, to exploring

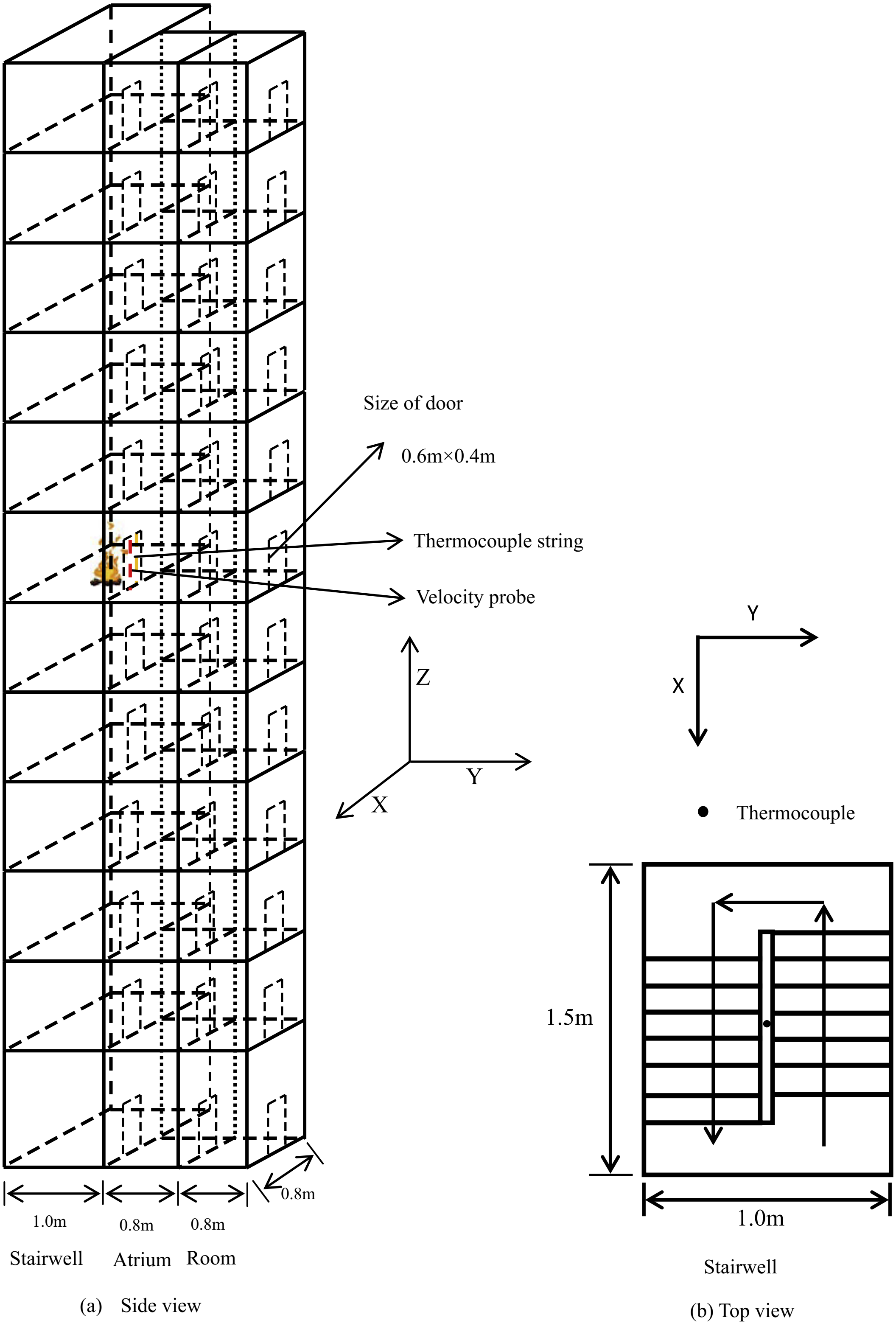

Fig. 1. Schematic of the stairwell.