Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 211 (2018) 681–688

Procedia Engineering

www.elsevier.com/locate/procedia

2017 8th International Conference on Fire Science and Fire Protection Engineering (on the Development of Performance-based Fire Code)

A Simulation Investigation of Fire Smoke Behavior above Urban Street Canyon

Chang-fa TAO[a], Kai WANG[b], Qin LIU[b], Pei-xiang HE[b,*]

[a]School of Automotive and Traffic Engineering, Hefei University of Technology, Hefei, Anhui 230009, China
[b]School of Civil Engineering, Hefei University of Technology, Hefei, Anhui 230009, China

Abstract

This paper presents computer simulation investigations on the fire smoke movement above street canyons under a velocity condition. Two buildings with different aspect ratios, when the range of windward building height is 5 ~ 17 m and the leeward building height is remain at 18 m, have been considered. Besides, the width of the street canyon has been taken as an important factor to influence the critical velocity here. It is shown that the critical velocity of the fire smoke, which moves from windward building to the leeward building, decreases with the height of windward building firstly, but increases with its further increase. According to the theoretical analysis, it is found that the critical velocity is a function of dimensionless width of street canyon. And a developed model has been developed to predict the critical velocity above the street canyon.

Peer-review under responsibility of the organizing committee of ICFSFPE 2017.

Keywords: Building height; Street canyon; Air flow speed; Fire smoke; Critical velocity.

Nomenclature	
h	The height of fire smoke at arbitrary point above the street canyon
H_1	The height of windward building (m)
H_2	The height of leeward building
$\dot{Q}$	Heat release rate
t	Time
t'	Time
u_c	The critical velocity
u_0	Ambient air flow speed
v_h	The velocity of fire plume
W	Street canyon width
Greek symbols	
κ	Parameter
η	Parameter

1877-7058
Peer-review under responsibility of the organizing committee of ICFSFPE 2017
10.1016/j.proeng.2017.12.064

1. Introduction

As part of the urbanization process, ultra high-rise buildings have been becoming increasingly common in the metropolises. Those high buildings are always built beside the roads or streets with different height. Once the fire accident occurs, the street canyon would be filled with pollutant dispersion which injures the body.

At recent years, many studies have been carried out on pollutant dispersion in and above the urban street canyons. Many works have reported the pollutant transportation within the street canyons by CFD simulation [1-15] or experiments [4, 10, 11]. The pollutant diffusion has been often taken as a hot research area for several years. The characteristics of flow and reactive pollutant dispersion in an urban street canyon can vary depending on the degree of street bottom heating [6]. For most cases under an isothermal condition one main vortex in the central and two small vortices in the lower corners of the street canyon are formed [9]. Mohamed [12] studied the distribution of the horizontal velocity and vectors in the canyon for the different roof shapes and its heights. The results showed that the wind velocity increases with the increasing of flat roof, slanted roof and trapezoid-shaped roof and decreases with the downwind and upwind wedge-shaped roofs. The wind velocity decreases as the roof height increases. Considering one street canyon with different aspect ratio (AR), Li [13] found that the distribution pattern of concentration budget in the street canyon of AR 1 is almost the same as that in the street canyon of AR 3, 5, and 10.

For the fire plume, with a strong thermal buoyancy, is very different with the diffusional characteristic of air pollutions produced by automobile. The fire smoke with high temperature rises up quickly, since a strong buoyancy force. It may damages the glass walls or ignites the combustible material of the buildings. Therefore, much more seriously fire accident would be brought. Besides, the fire plume with dangerous smoke soot and gases, such as CO, NOx, SOx, PM, is poisonous to human bodies.

McCaffrey [16] found the variation of centerline plume velocity with height. Few works have considered the pollutant gas dispersion with strong buoyancy [14, 15]. Hu. et al. [14] studied the dispersion of buoyancy driven smoke soot and carbon monoxide (CO) gas, which was ejected out from side buildings into an urban street canyon by fire dynamics simulator (FDS), large eddy simulation (LES) under a perpendicular wind flow. It was reported that with the increase in the wind velocity, the Froude number became higher. They had a good positively linear relationship, as in accordance with a developed model. And a correlation between the critical re-entrainment wind velocity and the characteristic height based on the LES simulation results has been developed. As a further consideration, the width of street canyons, the heat release rate and the building height are taken as impact factors [15], Hu found that the effect of aspect ratio (W/H) on the critical recirculation wind velocity to fall into two behavioral regimes in the skimming flow pattern.

These works considered the effect of street canyon width, building height, heat release rate, fire source location, and so on. But the aspect ratio of the building height has been ignored while it influenced the dispersion characteristic seriously under the air flow condition. In this paper, the dispersion characteristic of fire smoke within the street canyon has been investigated by fire dynamics simulator (FDS), large eddy simulation (LES). A physical model has been developed for the fire smoke which moves from windward building to the leeward building.

2. Numerical method and model configuration

2.1. Numerical CFD methodology

Fire Dynamics Simulator (FDS) has been widely used in the fire simulation, which is developed by the National Institute of Standards and Technology (NIST). The governing equations and solution methods for the gas phase are described in the FDS Technical Reference Manual [17,18]. Both for the fundamental research areas and practical engineering applications, numbers of works have reproduced or predicted the development of fires finished by FDS successfully [14, 15, 19-28]. Based on this, the simulation results by FDS-LES can provide reference significance in a high degree for the investigation which finished difficultly by experiments.

2.2. *Physical model setup*

As shown in Fig. 1, a 3D space with 40 m in height, 24 m in width, and 40 m in length has been considered. The number of grid is 96 × 160 × 160 in the x-, y- and z-direction, respectively. The width of street canyon and the two buildings was 18 m and 3 m, respectively. The height of leeward building was remain a constant (H2 = 18 m), while the windward one varied from 5 m to 17 m. A uniform horizontal air flow was blown into the simulation domain, while the top and other three sides were naturally open. For all the cases, the air flow was set to be perpendicular to the street canyon, and the air flow was blew from left to right with different flow speeds (0 ~ 5 m/s), shown in Table 1. The fire source was set in the central of the