Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 211 (2018) 726–735

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)

CFD Simulations of the Interaction of the Water Mist Zone and Tunnel Fire Smoke in Reduced-scale Experiments

Jun-heng WANG[a,b], Qiao-min NIE[a,c], Zheng FANG[a,b], Zhi TANG[a,b,]*

[a]*Wuhan University, Wuhan 430072 China*
[b]*Engineering Research Center of Urban Disasters Prevention and Fire Rescure Technology of Hubei Provinceb, Wuhan 430072, China*
[c]*Wuhan Economic Development Zone (Hannan) Environmental Protection Bureau, Wuhan 430090, China*

Abstract

Computational Fluid Dynamics (CFD) is a useful tool in tunnel fire safety, providing reliable data as a supplement and extension of experiments. In this work, simulations of reduced-scale tunnel fire experiments, concerning in a concrete tunnel model with a scale of 1/3, are presented. It mainly focuses on evaluating the quality of the CFD results. The results show that the simulations consist with the experiments with natural ventilation and longitudinal ventilation. Extension tests are also presented, considering the length and the working pressure of the water mist zone under longitudinal ventilation. It concludes that the downstream temperatures decrease with the length of water mist zone, and no pronouncedly change is obtained as water pressure increases under longitudinal ventilation.

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

Keywords: *CFD simulations, FDS, tunnel fire, water mist zone*

Nomenclature

c_p	specific heat capacity,(*kJ/(kg ・K)*)
g	gravitational acceleration, (m/s^2)
k	thermal conductivity, ($W/m \cdot K$)
D^*	characteristic length scale of the fire source
$\dot{Q}$	heat release rate of the fire source, (kW)
T	temperature, (K)
Greek symbols	
ρ	density, (kg/m^3)
Subscripts	
∞	ambient

1. Introduction

It is the smoke that mainly causes damages to properties and casualties in tunnel fires [1-3]. Thus smoke controlling is a

* Corresponding author. Tel.: +86 027 6877 2312;.
E-mail address: zhi.tang@whu.edu.cn（Zhi Tang）

1877-7058
Peer-review under responsibility of the organizing committee of ICFSFPE 2017
10.1016/j.proeng.2017.12.069

priority problem to handle in tunnel fire preventing. Because the longitudinal ventilation system, which is wildly applied around the world, may broke the smoke stratification [4], resulting in obscuring evacuation and rescue, people begin to searching alternatives and enhancements. Blocking the smoke by water based system attracts lots attentions as a potential approach.

To date, many research studies have been conducted concerning the use of water based system in blocking fire smoke [5-11]. However, the works about considering the water mist preventing fire smoke in work is relatively rare. Experimental studies were often conducted in reduced-scale model tunnels that were made of fireproof glass (e.g. Sun et al. [6]) or galvanized steel (e.g. Zhang et al [8]). Reports about the experimental studies performed in concrete tunnel model with a considerable larger scale (e.g. 1/3) are rarely read from the documents at hands. It is interesting to discuss the cases that experiment performed in such tunnel model, as the model is closer to the practical. And such experiments are mentioned in this work.

Unfortunately, there are limits in observation when the model tunnel is made of concrete and in a relatively large scale. The variations in temperature filed and smoke stratification are hardly obtained with current instrumentations, as mentioned in section 2. Computational Fluid Dynamics (CFD) then proposes a potential approach as supplement of the experiments. Generally, CFD simulations extend the range of variation discussed. Fire Dynamic Simulator (FDS), one of the CFD tools, is possible to provide reliable and precise results in tunnel fire with water mist operating, according to previous reports [11]. In this work, simulations of the experiments conducted previously are performed with FDS. Based on the comparisons of the simulated and experimental results, the quality of the simulation results obtained by FDS would be assessed first. Afterwards, the length and water pressure of the water mist zone are varied in the simulations to extend the experimental data. Discussions are given at last. .

2. Simulation set-up

2.1 Model details

To compare with the experimental results, the numerical tunnel model is rebuilt first, following the experimental tunnel model. The dimensions of the tunnel are 30 *m* (length) × 3 *m* (width) × 2 *m* (height), corresponding the full-scale dimensions of 90 *m* (length) × 9 *m* (width) × 6 *m* (height) with a scale of 1/3, as shown in Fig.1. All the materials of the tunnel model were set as concrete (ρ =2280 kg/m^3, c_p=1.04 $kJ/kg*K$ and k=1.8 $W/m*K$), following the experiment model. Two steel trays containing methanol are positioned at the central of the tunnel as fire source. The dimensions of each tray are 0.84 *m* (length) × 0.59 *m* (width). The heat release rate (HRR) of single methanol pool is 214 *kW*, according to the measurements in experiments.

Fig.1(a) shows the position of the water mist zone. There are 5 rows of nozzles at each half of the tunnel in the interval of 1.5 *m*. And every row of nozzles contains 3 nozzles in the interval of 0.75 *m*. Thus the length of the water mist zone is determined by the rows of activated nozzles. An axial fan also set at the left side end of the tunnel, supplying the longitudinal air flow.

(a)

(b)

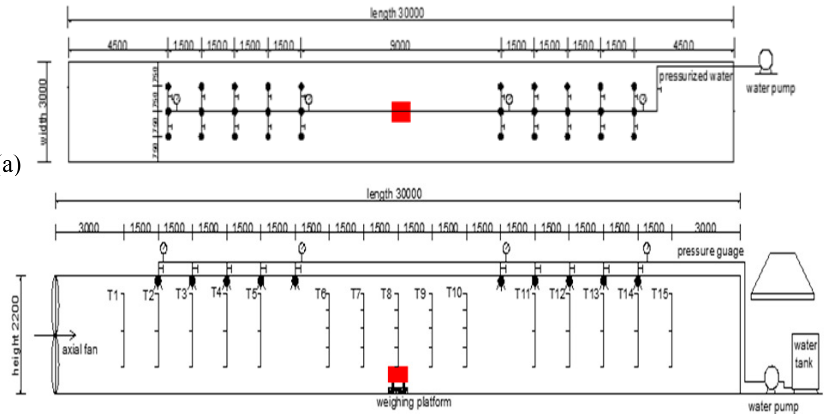

Fig.1 The Schematic diagram of the model tunnel
(a) top view (b) side view

Fig.1(b) shows the thermocouple trees set in the model. Because the tunnel is symmetrical, most discussions are about the temperature downstream the fire source (at the right hand side in Fig.1). All the thermocouples are located at the same