International Journal of Heat and Mass Transfer 73 (2014) 158–169

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Effects of vertical shaft arrangement on natural ventilation performance during tunnel fires

C.G. Fan [a], J. Ji [a,*], W. Wang [b], J.H. Sun [a]

[a] State Key Laboratory of Fire Science, University of Science and Technology of China, Hefei 230026, China
[b] Yangzhou Fire Detachment, Fire Bureau of Jiangsu Province, Yangzhou 225000, China

ARTICLE INFO

Article history:
Received 5 December 2012
Received in revised form 25 November 2013
Accepted 1 February 2014
Available online 3 March 2014

Keywords:
Tunnel fire
Smoke
Natural ventilation
Longitudinal wind
Stack effect

ABSTRACT

Investigation of natural ventilation using shafts in tunnels has been receiving more attentions, however, analyses on how shaft dimension and amount influence the natural ventilation performance have rarely been addressed. For the sake of fire protection and construction of tunnels, the influence of vertical shaft arrangement on natural ventilation performance during tunnel fires is investigated numerically by Large Eddy Simulation. The smoke flow characteristics in the tunnel and shaft under the combined function of longitudinal wind and stack effect of shaft are analyzed. Results show that both plug-holing and boundary layer separation will influence the natural ventilation performance. As a whole, the total mass flow rate of smoke exhausted by shafts increases with the shaft amount under a given total area of shafts. The case with maximum shafts for natural ventilation can gain the best ventilation performance in spite of the longitudinal wind. The case with the largest longitudinal wind velocity will gain the minimum total mass flow rate of smoke exhausted in spite of the shaft amount, due to the fact that a very obvious boundary layer separation occurs in the shaft. It is suggested that the cross-section of one shaft opening in the actual engineering design is oversize in general, which is not in favor of exhausting smoke. The influence of natural ventilation on smoke backflow and a special phenomenon, smoke bifurcation are also investigated.

1. Introduction

In designing green or sustainable buildings, introducing more natural ventilation would avoid costly equipment maintenance and electrical energy consumption, and give similar or even better thermal comfort to those buildings relying on mechanical air-conditioning systems [1]. Air quality might be improved as more fresh air will be delivered. In general, driving forces for natural ventilation are the stack effect and wind-induced action [2]. In recent years, natural ventilation types such as solar chimney and vertical shaft have become popular in relevant constructions.

Tunnel is an effective way to solve urban traffic problems. Nowadays, more and more urban tunnels are under construction all over the world. However, owing to the special structure of tunnels, smoke and toxic gases induced by fires, such as carbon monoxide, which are the most fatal hazard to the people, will not easily be discharged. The smoke will hamper safe evacuation of occupants and hinder firefighters extinguishing the fire.

For tunnel fires natural ventilation using vertical shafts can avoid air fans that will reduce the tunnel section height and does not consume power in the operation process [3]. Shafts not only facilitate airflow exchange in tunnels so as to improve interior air quality at ordinary times, but also contribute to discharge of high temperature smoke and weaken its impact on lining structures and equipments.

Wang [3] investigated the smoke spread and temperature field under natural smoke exhaust system without longitudinal wind in underground tunnel fires. Yan et al. [4] conducted full-scale burning tests in a road tunnel with natural ventilation using shafts and found that large amounts of smoke and heat were released through shafts. Fan et al. [5] carried out a set of burning experiments with heptane pool fires to investigate the air entrainment mode with natural ventilation using shafts in road tunnel fires. Ji et al. [6] conducted a set of burning experiments, to investigate the effect of vertical shaft height on natural ventilation in urban road tunnel fires.

Although the investigation of natural smoke exhaust system has been receiving more attentions and meanwhile providing reliable information in some former studies, delicate quantitative analyses on how the distance between adjacent shafts on the tunnel ceiling and the shaft dimensions influence the natural ventilation performance

* Corresponding author. Tel.: +86 551 3606431(O), mobile: +86 13721101322.
E-mail address: jijie232@ustc.edu.cn (J. Ji).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.02.003
0017-9310/

have rarely been addressed. For the sake of fire protection and construction of tunnels with natural ventilation using shafts, the present work investigated numerically the effect of vertical shaft arrangement on natural ventilation performance during tunnel fires.

2. Numerical modeling

With the development of computational fluid dynamics (CFD) and advent of increasing computer power, it has already been possible to utilize CFD technology to analyze problems involved in three-dimensional flows in buildings [7]. CFD codes such as CFX [8], Fluent [9] and Fire Dynamics Simulator (FDS) [10] are widely applied in simulating building ventilation.

The software package, FDS [11], coupling with a post-processing visualization tool, Smokeview, developed by National Institute of Standards and Technology (NIST), USA, could now be regarded as a practical tool for simulating fire-induced environment. The model has been subjected to numerous validations, calibrations and studies on temperature and velocity fields in fires. There are also some related studies in tunnel fire research field [12–16], which have justified that FDS is a valid tool to model the fire-induced smoke and reliable on modeling natural ventilation in tunnel fires.

2.1. Fire dynamics simulator

FDS solves numerically a form of the Navier–Stokes equations for thermally-driven flow. A description of the model, many validation examples, and a bibliography of related papers and reports may be found on http://fire.nist.gov/fds/. The LES model, which is widely used in study of fire-induced smoke flow behavior, is selected in this study.

The Sub-Grid-Model (SGM) commonly used in LES is developed originally by Smagorinsky [17]. The eddy viscosity is obtained by assuming that the small scales are in equilibrium, by balancing the energy production and dissipation.

The Smagorinsky constant in LES simulation is flow dependent and has been optimized over a range from 0.1 to 0.25 for various flow fields. FDS has been subjected to many verification works and improved since its first release in 2000. According to these validation works, the constants, C_s, Pr and Sc, are set as default values in FDS for current paper as 0.2, 0.2 and 0.5, respectively [10]. It was reported [18] that for simulating buoyancy-drive flow, the predicted values from the filtered dynamics SGM by FDS agreed better with the measured value than those from the original Smagorinsky model and RANS (Reynolds-Averaged Navier–Stokes) models.

2.2. Fire scenario analysis

By taking the aspect ratio of actual tunnels into account [19,20], the model tunnel in current research was specified as 150 m long, 10 m wide and 5 m high. The fire source was located 50 m away from the left end of the tunnel and designed to develop following the t-square curve with a peak heat release rate of 3 MW, representing the typical scenario of car fire as some tunnels, especially urban tunnels, prohibit vehicles with dangerous chemicals or heavy goods from passing by.

In the full-scale experiments conducted by Yan et al. [4], the ambient longitudinal wind velocity (V) was measured between 0.8 m/s and 1.5 m/s. Therefore, to investigate roundly the effect of longitudinal wind on the natural smoke exhaust, the wind velocity range was enlarged and set to be 0–3 m/s with the interval of 1 m/s in current physical model. The ambient temperature was set as 20 °C.

An inlet velocity boundary condition was set at the left opening of the tunnel domain. The top of the shaft and the right side of the tunnel were set to be naturally opened without initial velocity. In order to take into account the interactions of inflow and outflow properly, it is necessary to extend the computation domain beyond the physical boundaries of the openings of the enclosure [21]. Fire simulation results with the extended computational domain yield better agreement with experimental data than those without domain extension. Therefore, the additional computational regions were added near the top opening of vertical shaft and the tunnel outlet. The internal linings of the tunnel and shaft were specified as ‘CONCRETE’. The thermal properties of this material are available in the FDS database documentation.

In order to assess the effect of shaft amount on natural ventilation performance under a given total area of shafts, we investigated flow fields with five different shaft amounts, namely 1, 2, 3, 4, 6. For each case, the total area of shafts on the tunnel ceiling was 12 m^2 and the height of each shaft was 5 m. The schematic diagram of the model is shown in Fig. 1.

In each case, the area of each shaft was the total area of shafts divided by the shaft amount. The shafts were evenly distributed in the region between the fire and the right end of the tunnel and numbered consecutively in the downstream direction of the fire (for example, the shafts in case with three shafts for natural ventilation were numbered as shaft 1, shaft 2 and shaft 3, respectively). Table 1 shows the shaft length and width, distance between adjacent shafts and distance between the fire and shaft 1 in cases with different shaft amounts. For comparison purposes, the case without shafts for natural ventilation was also studied. In total, 24 cases were simulated.

2.3. Sensitivity study on the grid system

In FDS simulations, the grid size is a key parameter to be considered. A $D^*/\delta x$ criterion has been widely used for assessing the grid resolution [11], where δx is the grid size and the characteristic length of D^* is calculated by:

$$D^* = \left(\frac{\dot{Q}}{\rho_\infty c_p T_\infty \sqrt{g}}\right)^{\frac{2}{5}} \quad (1)$$

where $\dot{Q}$ is heat release rate, ρ_∞ is ambient density, c_p is specific heat capacity, T_∞ is ambient temperature and g is gravitational acceleration constant. It was recommended by McGrattan et al. [11] that the value of $D^*/\delta x$ should be in the range of 4–16. Then the size of the finest mesh for a 3 MW fire was calculated to be between 0.09 m and 0.37 m.

Obviously, finer grid will better reflect the heat flow field in detail, but it is also time consuming. So we have to make a choice for an appropriate grid size. In this paper, six different mesh sizes ranging from 0.1 m to 0.33 m were chosen for comparison. Fig. 2 presents the vertical temperature distribution in the tunnel with different grid sizes. With the mesh density increasing, the temperature curve trends to be uniform. The results of mesh with 0.167 m, 0.125 m, and 0.1 m get slightly difference, that is to say, there is no significant improvement but more time consuming when the mesh size is smaller than 0.167 m. Hence, we choose a mesh system with grid size of 0.167 m in this study.

3. Results and discussion

3.1. Cases without longitudinal wind

As is well-known, the longitudinal wind will no doubt influence the smoke movement in the tunnel [22] and even shaft. First of all, the cases without longitudinal wind (V = 0 m/s) are investigated as a benchmark.

Fig. 3 shows the temperature distribution in the shaft and tunnel in the quasi-steady state, namely, when smoke temperature,