Journal of Southeast University (English Edition) Vol. 29, No. 3, pp. 305 - 309 Sept. 2013 ISSN 1003—7985

Smoke distribution in naturally ventilated urban transportation tunnels with multiple shafts

Tong Yan[1] Shi Mingheng[2] Zhai Zhiqiang[3]

([1]College of Urban Construction and Safety Engineering, Nanjing University of Technology, Nanjing 210009, China)
([2]School of Energy and Environment, Southeast University, Nanjing 210096, China)
([3]Department of Civil Environmental and Architectural Engineering, University of Colorado at Boulder, Boulder 80309, USA)

Abstract: The smoke spreading law of urban transportation tunnels with multiple shafts under natural ventilation is studied. A full-scale burning experiment is conducted in an actual tunnel. The study shows that smoke temperatures below the tunnel ceiling reduce rapidly along the longitudinal towards the tunnel exits. A noticeable temperature stratification is observed near the fire source. Most fire smoke is exhausted out of the shafts, while the number of the smoke shafts in the downstream is more than that in the upstream. Large eddy simulation (LES) based on computational fluid dynamics (CFD) is carried out using the fire dynamics simulator (FDS) software with parallel processing in which the grid size of the fire-domain is set to be 0.083 m. The simulation results of temperatures under the ceiling, the smoke fronts and the shafts' smoke exhaust or air supply agree reasonably with the experimental data. Further simulations indicate that the decreasing ambient temperature or shaft spacing might reduce smoke temperatures under the tunnel ceiling and increase mass flow rates out of the shafts. This study provides technical scientific evidence and supports for the design and construction of such kinds of tunnels.

Key words: tunnel; natural ventilation; multiple shafts; large eddy simulation; smoke spreading

doi: 10.3969/j. issn. 1003 - 7985. 2013. 03. 014

One-way urban traffic tunnels with a shallow depth ($\leqslant$6 m) and a short length ($\leqslant$3 000 m) have recently been built in China. Some of these tunnels use a group of shafts on the tunnel ceiling to supply and exhaust air with natural ventilation instead of with mechanical ventilators being installed. This design strategy usually conflicts with the current fire code[1] because the tunnel is longer than the allowed length for natural ventilation. Local construction bureaus and fire departments mostly worry about the potential fire hazards in such a special structure. Tunnel fire smoke has very complex flow characteristics because its physics is affected by the tunnel geometry[2], the ambient air temperature, wind pressure, heat and mass transfer, as well as chemical reactions, etc. But the tunnel slope[3] is often ignored in shallow urban tunnels. Merci et al.[4] studied a small compartment fire with natural roof ventilation and concluded that the total fire heat release rate has the strongest influence on the hot smoke layer average temperature while the influence of the fire source area and the roof opening is smaller. Chen[5] emphasized that the applications of computational fluid dynamics (CFD) models were mainly for studying indoor air quality, natural ventilation, and stratified ventilation as they were difficult to predict with other models. Chow et al.[6] modified a simple buoyancy-driven flow model to determine the smoke layer interface heights for static smoke exhaust systems with natural vents installed in atria, and the simulation predictions of the fire dynamics simulator (FDS), CFAST and the proposed flow model results all verified that a natural vent was effective in keeping smoke above a certain height. The major aims of the present study are to provide insight into the smoke flow phenomena of naturally ventilated tunnels with multi-shafts and to find out whether such passive ventilation provides fire safety or not.

1 Tunnel Experiments

A full-scale fire test is conducted in Nanjing, China. The Xi'anmen Transportation Road Tunnel[7] is located at Nanjing, China which is 1 770 m long and has twin one-way tubes. Each tube is divided into five sections marked as section 1, section 2, section 3, section 4, and section 5 by four groups of multi-shafts. Each shaft is 12.3 m (length) ×2.6 m (width) ×6 m (height). The energy release rate Q (in MW) in fire is given by[8]

$$Q = A_{\mathrm{f}} m' \chi \Delta H_{\mathrm{c}} \tag{1}$$

where A_{f} is the horizontal burning area of the fuel, m^2; m' is the average burning rate per unit area, kg/(s · m^2); χ is the combustion efficiency, %; ΔH_{c} is the complete heat of combustion, MJ/kg. At the initial stages in reality, Q is always proportional to the square of time[8]:

$$Q = \alpha (t - t_0)^2 \tag{2}$$

Received 2012-12-29.

Biography: Tong Yan (1973—), female, doctor, associate professor, njtongyan@njut.edu.cn.

Foundation items: The National Natural Science Foundation of China (No. 51178217), the Natural Science Foundation of Jiangsu Province (No. BK2011804).

Citation: Tong Yan, Shi Mingheng, Zhai Zhiqiang. Smoke distribution in naturally ventilated urban transportation tunnels with multiple shafts. [J]. Journal of Southeast University (English Edition), 2013, 29(3): 305 - 309. [doi: 10.3969/j. issn. 1003 - 7985. 2013. 03. 014]

where α is the increasing coefficient, MW/s^2; and t_0 is the ignition time, s. Vehicles with dangerous goods are not permitted to enter; so the most possible fire is self-ignition of cars or buses, and its peak heat release rate is around 5 MW. A 1.8 m × 1.8 m oil pan containing diesel and gasoline is placed on the ground, as shown in Fig. 1, which is closed by a metal board. After the ignition, A_f increases gradually by dragging the board away from the pan, and then Q follows the t^2 law by controlling the board's moving speed based on Eqs. (1) and (2). The maximum 5 MW is reached at 240 s when the oil pan is entirely opened. Two sets of copper-constantan thermocouples and some colored straps are bound to each steel pole. Two observers walk along the downstream and upstream to record the movement of the smoke fronts respectively. The ambient air temperature is about 5 ℃. The ambient air velocity is about 0.8 to 1.5 m/s towards north.

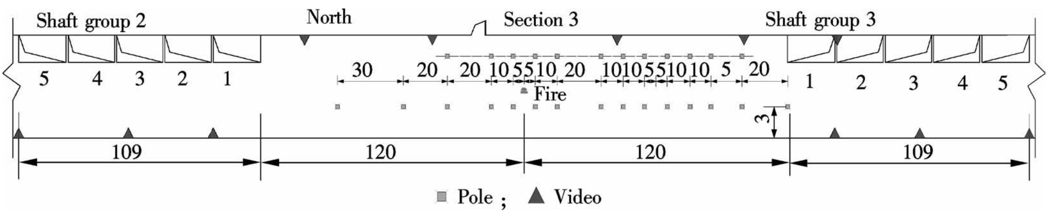

Fig. 1 Overall view of the experimental layout

2 Computational Fluid Dynamics Modeling

The CFD technique is used to simulate the tested tunnel fire and ventilation. An approximate form of the original Navier-Stokes equations appropriate for low Mach number applications is used in the FDS software. To handle sub-grid scale convective motion, the large eddy simulation (LES) technique is adopted here. In the LES, the smallest eddies are filtered out of the governing equations. Their influence is modeled through the introduction of an eddy viscosity μ_{ijk} in the momentum equations. The FDS applies the Smagorinsky model:

$$u_{ijk}=\rho_{ijk}(C_s\Delta)^2S \tag{3}$$

$$S^2=2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+2\left(\frac{\partial w}{\partial z}\right)^2+\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2+\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)^2+\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2-\frac{2}{3}(\nabla\cdot\boldsymbol{u})^2 \tag{4}$$

where Δ is related to the local mesh configuration as $\Delta=(\delta x\delta y\delta z)^{1/3}$, and δx, δy and δz are the sizes of a grid cell[9]. The default value of the Smagorinsky constant C_s in the FDS is 0.2[9–10]. Time step is constrained by the convective and diffusive transport speeds via the Courant-Friedrichs-Lewy (CFL) condition[9]:

$$\delta t\ \max\left(\frac{|u_{ijk}|}{\delta x},\frac{|v_{ijk}|}{\delta y},\frac{|w_{ijk}|}{\delta z}\right)<1 \tag{5}$$

Grid sensitivity is tested in a 50 m section within the studied tunnel. The fire is centrally placed on the ground with its domain size of 6 m (length) × 4 m (width) × 5.75 m (height). The non-fire domain is evenly divided into four meshes. Adjacent meshes just abut each other[11]. Four mesh sizes in the fire domain, 0.167, 0.125, 0.100 and 0.083 m, respectively, are tested, being half of the mesh in the non-fire domain. The simulation is performed for 300 s of physical time using a 2.40 GHz PC. In the scheme of 0.083 m, the plume impinges on the ceiling and flows down along the wall before the ambient air is entrained into the plume, as shown in Fig. 2. The CFL number is less than 1; its minimum quasi-steady time step is set to be 0.005 s; and its heat release rate fluctuates most weakly compared with the other three schemes. As a result, the grid resolution of 0.083 m is chosen for the following studies.

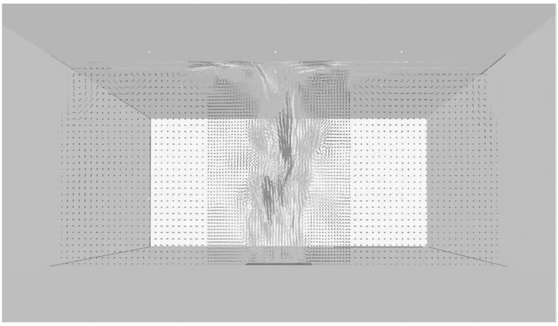

Fig. 2 Cross-sectional velocity contour in scheme of 0.083 m

A 600 m section of the studied tunnel is simulated in the FDS including section 3, shaft groups 2 and 3, as shown in Fig. 1 and Fig. 3. A fire source of 1.8 m × 1.8 m is located centrally on the ground. A heat release rate per unit area of 1 675.9 kW/m^2 is imposed and the maximum heat release rate is 5 MW. The ambient air temperature of 5 ℃ and the wind speed of 0.9 m/s are set. The walls are treated as adiabatic. Fifteen multiple meshes are formed including a fire domain and 14 non-fire domains, so the total number of grid cells is 2 414 880. The simulation is performed for 500 s on a 16 processors, 2.27 GHz supercomputer with parallel processing. Each processor receives one mesh to work on it. The total CPU operating time is 45 h.

Fig. 3 shows the longitudinal smoke propagation at 200 and 450 s, respectively. Stratified smoke is evident. The downstream smoke spreads faster and farther than the upstream smoke, and its layer interface is higher. Backflow occurs clearly at the bottoms of smoke fronts, especially at the late stage of the movement. The findings agree well with the on-site observations. The crew at the upstream near the fire source is not hurt by hot smoke but their evacuation route is somewhat obstructed because of the smoke darkness.

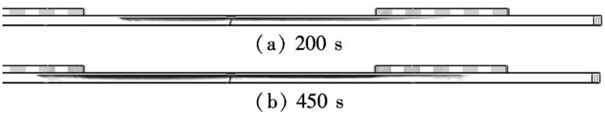

Fig. 3 Smoke spreading at different times in simulation

The temperature distributions under the tunnel ceiling at 240 s are compared with the experimental results as shown in Fig. 4. The temperature decays rapidly when moving away from the fire source in both directions. The CFD predictions are slightly higher, especially near the fire source. This is attributed to the lower heat losses in the CFD simulations than those in the experiments.

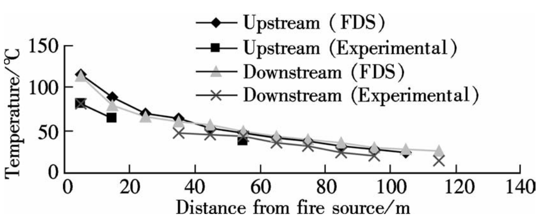

Fig. 4 Smoke temperatures under the ceiling at 240 s

The predicted mass flow rates through the shafts are depicted in Fig. 5, in which smoke exhaust is denoted by "+" and air supply by "−". The downstream shafts 1, 2 and 3 have stable phenomena of exhausting smoke, while in the upstream only shaft 1 works because the smoke movement is obstructed by the ambient wind. All the maximum flow rates are about 40 kg/s.

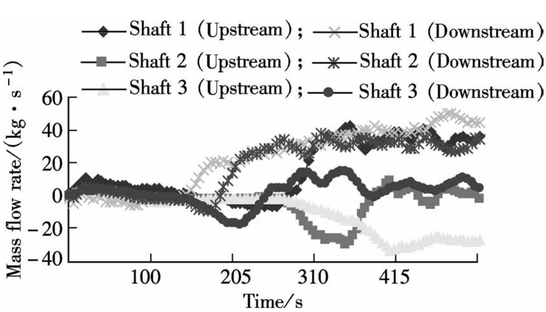

Fig. 5 Mass flow rates through shafts with time in simulation

3 Analysis of Influencing Factors

In order to investigate the influences of the ambient air temperature, shaft group spacing (interval between the downstream shaft 1 and the upstream shaft 1) and the shaft number in a group on the smoke dispersion and fire safety, a 600 m (length) × 4 m (width) × 5.75 m (height) tunnel with two multi-shaft groups is chosen to be simulated under seven schemes (see Tab. 1). In each scheme, the fire source is centrally placed on the ground, and the steady heat release rate is 5.4 MW, and the ambient wind speed is 0.9 m/s. The simulations are performed for 350 s on 15 multiple meshes with parallel processing.

Tab. 1 Schemes of simulation and CPU operating time

Scheme	Ambient temperature/℃	Group spacing/m	Shaft number per group	Length of a shaft group/m	CPU operating time/h
1	5	240	5	109	43
2	20	240	5	109	44
3	35	240	5	109	44
4	20	120	5	109	39
5	20	360	5	109	44
6	20	240	3	61	38
7	20	240	4	85	41

3.1 Ambient temperature

The differences due to the ambient air temperature only exist among schemes 1, 2 and 3. Fig. 6 shows the downstream smoke temperatures under the ceiling at 200 s. It is found that the higher the ambient air temperature, the higher the smoke temperature; all the smoke temperatures are lower than 100 ℃ when 25 m away from the fire, so the high-temperature inside the tunnel is not significant. Fig. 7 shows that the downstream shaft 1 always plays an important role in exhausting smoke and reaches a minimum mass flow rate under 35 ℃, while the upstream shaft 3 does not because the smoke front cannot arrive here. The averag CO concentrations at the safe height are about 0.59 mg/m^3, which is very low and is not a threat to people inside the tunnel.

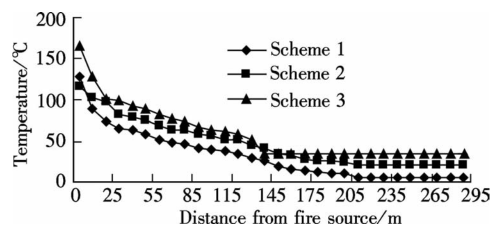

Fig. 6 Smoke temperatures under the ceiling in downstream in schemes 1, 2 and 3

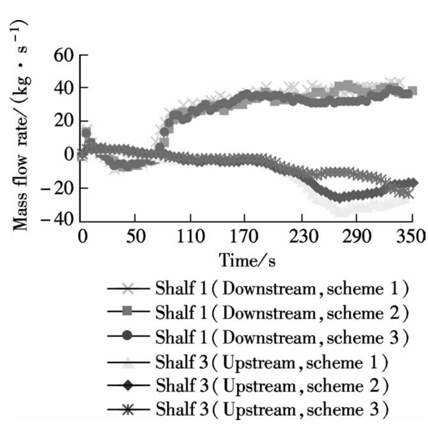

Fig. 7 Mass flow rates of shaft in schemes 1, 2 and 3

3.2 Shaft group spacing

The spacing between the adjacent shaft groups is often limited in an urban tunnel. Under schemes 2, 4 and 5, the distances between the fire source and the upstream/downstream shaft 1 are 120, 60 and 180 m, respectively. Fig. 8 shows that the temperature in scheme 4 is somewhat lower than those in schemes 2 or 5. Fig. 9 indicates that the smoke exhaust rates from the downstream shaft 1 increase with the decrease in the group spacing. Similarly, very low CO concentrations are found at the safe height.

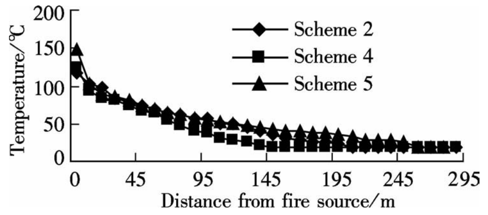

Fig. 8 Smoke temperatures under the ceiling in downstream in schemes 2, 4 and 5

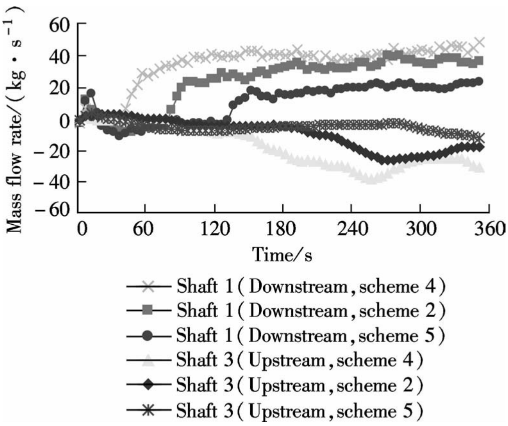

Fig. 9 Mass flow rates of shaft in schemes 2, 4 and 5

3.3 Shaft number in a group

For the shaft numbers of 5, 3, 4, there are no evident differences in the smoke temperatures under the ceiling, as shown in Fig. 10. In addition, only the downstream shafts 1, 2, 3 and the upstream shaft 1 always have an effect on smoke exhaust while the others do not. In general, most heat and smoke leave the three shafts closer to the fire source; hence, increasing the shaft number to 4 or 5 makes no sense.

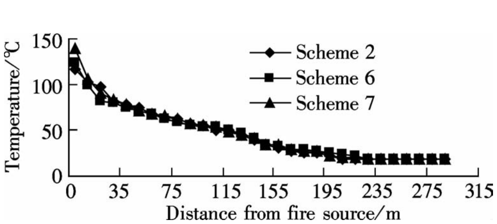

Fig. 10 Smoke temperatures under the ceiling in downstream in schemes 2, 6 and 7

4 Conclusions

The goal of this study is to find out the development of fire smoke in a naturally ventilated urban transportation tunnel with multiple shafts, so as to understand whether such passive ventilation provides sufficient fire safety. A full-scale burning experiment is performed in a real tunnel. LES-based CFD simulations are conducted by using the FDS software with parallel processing in which the grid size of the fire domain is chosen to be 0.083 m. Both the experimental and the simulation results show that the smoke temperatures decay fast towards the tunnel exits; most smoke is exhausted out of the shafts and people inside the tunnel will not be injured by the smoke. The further simulations reveal that:

1) Reducing ambient air temperature contributes to the decreased smoke temperatures below the ceiling and the increased smoke mass flow rates out of the shafts. Hence, the tunnel will be safer when a fire occurs in cold weather than in hot weather.

2) The shaft group spacing plays an important role in controlling fire smoke dispersion. When the spacing decreases, more heat and smoke will leave the shafts readily and the air temperatures below the ceiling are reduced greatly.

3) With a certain shaft group spacing, an optimal shaft number exists in a group. This study reveals that increasing the shaft number to 4 or 5 has no scientific meaning.

References

[1] Ministry of Communications of China. JTJ026. 1—1999 Specifications for design of ventilation and lighting of highway tunnel [S]. Beijing: China Communications Press, 1999. (in Chinese)

[2] Lee S R, Ryou H S. A numerical study on smoke movement in longitudinal ventilation tunnel fires for different aspect ratio[J]. *Building and Environment*, 2006, **41**(6): 719 − 725.

[3] Ballesteros-Tajadura R, Santolaria-Morros C, Blanco-Marigorta E. Influence of the slope in the ventilation semi-transversal system of an urban tunnel[J]. *Tunnelling and Underground Space Technology*, 2006, **21**(1): 21 − 28.

[4] Merci B, Van Maele K. Numerical simulations of full-scale enclosure fires in a small compartment with natural roof ventilation[J]. *Fire Safety Journal*, 2008, **43**(7): 495 − 511.

[5] Chen Qingyan. Ventilation performance prediction for buildings: a method overview and recent applications[J]. *Building and Environment*, 2009, **44**(4): 848 − 858.

[6] Chow C L, Li J. An analytical model on static smoke exhaust in atria[J]. *Journal of Civil Engineering and Management*, 2010, **16**(3): 372 − 381.

[7] Tong Yan, Shi Mingheng, Gong Yanfeng, et al. Full-scale experimental study on smoke flow in natural ventilation road tunnel fires with shafts[J]. *Tunnelling and Underground Space Technology*, 2009, **24**(6): 627 − 33.

[8] Karlsson Björn, Quintiere James G. *Enclosure fire dynamics[M]*. *New York: CRC Press LLC,* 2000.
[9] McGrattan K. Fire dynamics simulator (version 4): technical reference guide [R]. Washington DC: National Institute of Standards and Technology, 2006.
[10] Van Maele K, Merci B. Application of RANS and LES field simulations to predict the critical ventilation velocity in longitudinally ventilated horizontal tunnels [J]. *Fire safety Journal*, 2008, **43**(8): 598-609.
[11] McGrattan K. Fire dynamics simulator (version 4): user's guide [R]. Washington DC: National Institute of Standards and Technology, 2006.

多竖井自然通风城市公路隧道火灾烟气分布

童 艳[1] 施明恒[2] 翟志强[3]

([1] 南京工业大学城市建设与安全工程学院,南京 210009)
([2] 东南大学能源与环境学院,南京 210096)
([3]Department of Civil Environmental and Architectural Engineering, University of Colorado at Boulder, Boulder 80309, USA)

摘要:研究了多竖井城市公路隧道自然通风条件下的火灾烟气扩散规律. 在某真实隧道内实施了全尺寸燃烧试验,试验过程中顶棚下方烟气温度沿纵向朝出口快速衰减,火源附近存在明显的烟气分层现象,大量烟气从竖井排出,下游排烟竖井个数多于上游. 使用 FDS 火灾软件开展基于计算流体动力学的大涡模拟,火源区网格尺寸为 0.083 m. 采用并行计算技术,模拟得到的顶棚下方烟气温度、烟气前锋、竖井进/排烟均与试验数据吻合较好. 进一步的数值模拟结果表明:降低环境温度或竖井组间距有助于降低顶棚下方烟气温度,同时提高竖井出口的排烟质量流量. 本研究为该类隧道的设计与修建提供了理论与技术支持.

关键词:隧道;自然通风;多竖井;大涡模拟;烟气扩散
中图分类号:TU834