The Numerical Study of Fire Thermal Radiation Hazard in Inclined Tunnel Engineering Safety

Zheng Wei[1,2,3*], Feiyang Huang[1,2,3], Peizhong Yang[1,2,3]

[1]School of Mechanical Engineering, Shanghai Jiao Tong University, Shanghai 200240, PR China
[2]Shanghai Key Laboratory of Advanced Manufacturing Environment, Shanghai 200240, PR China
[3]State Key Laboratory of Mechanical System and Vibration, Shanghai 200240, PR China

*wzhz381000799@sjtu.edu.cn

Abstract. The slope of the tunnel has an impact on the distribution of smoke and droplets, which subsequently affects the extent of thermal radiation damage experienced by individuals in the tunnel. Numerical simulation was employed to replicate the progression of fires in inclined tunnels. Using this simulation method, the temperature field, gas radiation properties, and droplet distribution within the tunnel fire were determined. A discrete human mesh was then positioned within the fire field, and the radiant heat flux on its surface was calculated using the backward ray tracing method. The intensity and distribution of radiant heat flux on the human surface, in the direction of the upwind and downwind outlets of the fire source, were simulated at different stages of fire development. By analyzing the changes in radiant heat flux and numerical values, the hazards of thermal radiation to human beings at different stages of fire development and various locations were summarized. The findings of this study indicate that in tunnel fires, during the intermediate stage of fire development when the air heat flow is established and the smoke concentration is not at its maximum, individuals in the downwind direction are at the highest risk of thermal radiation injuries. However, when the smoke concentration reaches a sufficient level, the damage caused by thermal radiation to the human body is significantly reduced.

Keywords: inclined tunnel fire, thermal radiation hazard, backward ray tracing method, numerical simulation, engineering safety

1 Introduction

Tunnel fires have always been one of the research priorities in the area of fire safety, due to the narrow and relatively closed space in the tunnel, the economic losses and casualties caused by tunnel fires are much greater than those caused by ordinary accidents. Therefore, the study of personnel evacuation in tunnel fire is of great significance. The past research mainly focuses on the maximum temperature under the tunnel ceiling [1], the back layer distribution and flame shape [2] and the effect of these factors

Z. Zhang et al. (eds.), *Proceedings of the 2024 6th International Conference on Structural Seismic and Civil Engineering Research (ICSSCER 2024)*, Advances in Engineering Research 246,
https://doi.org/10.2991/978-94-6463-556-0_12

on the personnel evacuation. There are two main research methods: experimental methods and computer simulations.

Tunnel fire experiments are typically categorized into two groups: reality experiments conducted in actual tunnels and scale experiments. The fire experiments conducted in real urban tunnels [3] provide precise temperature and soot distribution within the tunnel, and the radiant heat flux can be measured at specific points. The experimental data is more accurate than simulation results [4][5]. However, both reality experiments and scale experiments are costly, requiring careful planning of the data to be measured and the methods of measurement. Additionally, the available information on fire fields is still limited. Furthermore, fire experiments pose a certain level of danger, particularly in terms of the damage caused by fire to the human body. Due to safety concerns, it is not possible to accurately simulate the real-life scenario of the human body in a fire field. Therefore, the experimental method has significant limitations in studying the thermal radiation damage to humans in fires.

The advancement of computer technology has facilitated research on evacuation in tunnel fires. Computational fluid dynamics (CFD) is utilized to calculate the temperature field and smoke variation in tunnel fires. In the current fire simulation method, thermal radiation is typically calculated using the finite volume method (FVM) to solve the radiation transfer equation (RTE) [6]. However, the human surface is a complex curved surface and is relatively small compared to the mesh size used in fire simulations. The use of FVM to calculate the radiant heat flux on the human surface in a fire will result in a significant ray effect, making it impossible to obtain accurate results.

In general, current research focuses on studying the impact of thermal radiation on personnel evacuation from the perspective of the fire source. However, due to the complexity of the human body and the fire environment, previous methods are unable to accurately calculate the radiant heat flux on the human surface in a fire. In this study, a backward ray tracing method is employed to calculate the radiant heat flux on the human surface in order to evaluate the effect of thermal radiation on the safe evacuation time in tunnel fires.

2 Physical And Mathematical Models

The framework of our proposed model for calculating radiant heat flux on the human surface in a fire consists of three components: radiation properties in fire, a 3D human mesh, and the backward ray tracing method. A numerical simulation method is employed to calculate the development of the fire, and the radiation properties of soot and fire are subsequently extracted. A 3D human mesh is then incorporated into the fire scene, and the backward ray tracing method is utilized to calculate the radiant heat flux on the human body at various times and locations.

2.1 Tunnel Fire Model

According to the fire protection code for building design (2018 edition), the interval between the pedestrian crossing passage and the tunnel leading to the entrance of the pedestrian evacuation passage should be 250m~300m. The length of tunnel is set to be 300m and has a slope with a gradient of 2.9%. The width of the tunnel is 6m and the height is 4m, the cross-section of the tunnel is rectangular. The ignition point is set at the middle of the tunnel and two evacuation exits were set 125m upstream and downstream of the fire source.

In this study, a fire caused by a small car accident was simulated with the steady fire model. The heat release rate of fire is set to be 6.4MW. The fire source is set on the ground of tunnel and the size of fire source is $4m \times 2m$.

The numerical simulation program Fire Dynamic Simulator (FDS) is utilized in this study to simulate the progression of a tunnel fire. Within FDS, the grid size is identified as one of the most crucial parameters that influences the accuracy of the simulation. According to the FDS User Guide [7], the characteristic fire diameter D^* is given as:

$$D^* = \left(\frac{Q}{\rho_a C_p T_a \sqrt{g}}\right)^{\frac{2}{5}} \tag{1}$$

where ρ_a is air density, C_p is the specific heat of air, T_a is the air temperature and Q is the heat release rate of fire source.

In this study, the characteristic flame diameter calculated according to the above formula is 2.02m. As a rule of thumb, the grid size should be 1/4 or 1/16 of D^*. Considering both calculation time and accuracy, the grid size is finally set as 0.25m. Only the boundary conditions at the tunnel entrance and exit are set to open, and the rest of the boundaries are set to close. The simulation result of FDS is extracted in form of PLOT3D document and used in the calculation of radiant heat flux on human body. As show in figure 1.

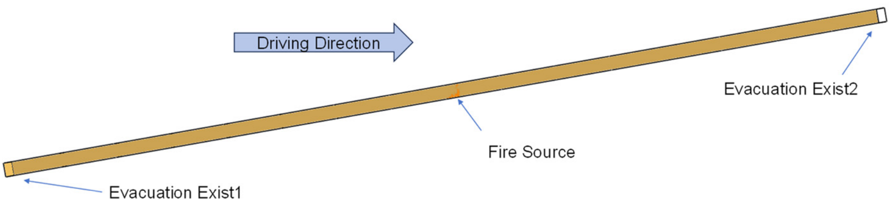

Fig. 1. Physical model of tunnel in FDS

2.2 Human Mesh

the 50th percentile data of adult men in northern China in the Chinese adult body size standard GB/T 10000 2023[8] is elected to complete the three-dimensional human modeling in CATIA. Then, ICEM was used to discretize the surface of the human body with a controlled number of triangular mesh surfaces and a detecting hemisphere is defined at weight center of each surface. The human body surface was divided into 20 parts (head, chest, face, back, etc.) for the analysis of radiant heat flux on human surface.

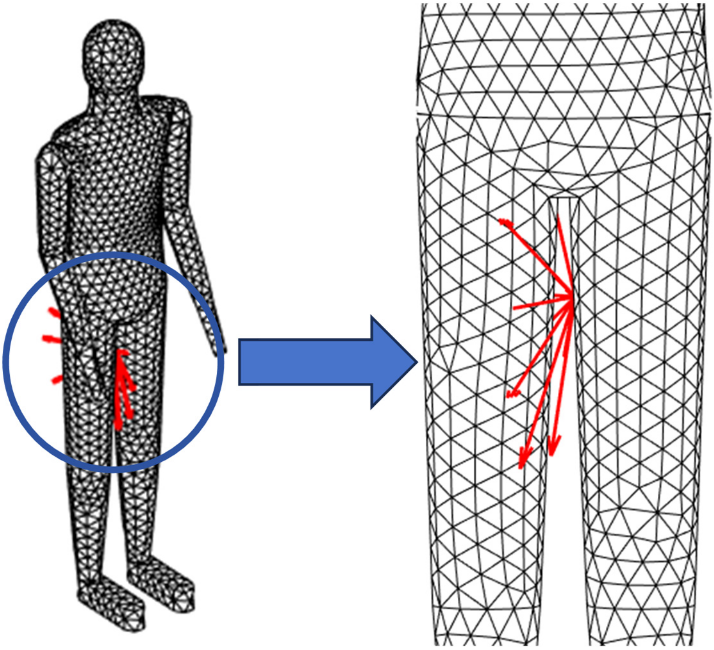

Fig. 2. 3D human mesh and ray generation

In order to improve the accuracy and efficiency of the calculation of radiant heat flux on the human surface, the outward hemisphere of each triangular mesh is divided into solid angles of equal size, and ray vectors are generated at the center of each solid angle. Collision detection is carried out between the rays and the human body, and only the ones that do not collide with the surface of the human body are retained. The resulting human body mesh as well as the rays are shown in the Fig.2. Taking the grid between the legs of a person as an example, it can be seen that after processing, the retained rays do not collide with the surface of the human body.

2.3 Backward Ray Tracing Method

In this study, the gray body hypothesis is applied. Compared to the size of the tunnel, the size of each human mesh is relatively smaller, therefore, backward ray tracing method is more efficient in this case. For radiative heat transfer, the governing equation is shown in Equation 2.

$$s \cdot \nabla I(r,s) = \kappa I_b(T) - (\kappa + \sigma_s) I(r,s) + S(r,s) \tag{2}$$

Where, s, r, κ, $\sigma_s$are the direction and position of radiation propagation, the average absorption coefficient of the radiant gas, and the scattering coefficient of the particle droplets. I, I_b are thermal radiation flux and gas blackbody radiation intensity. $S(r,s)$ is the scattering term.

The volume swept by a ray with a unit solid angle $\Delta\Omega_u$ advancing dl from x_i to x_{i+1} is ΔV_i, the area of the grid emitting this ray is A_n. Then the radiant heat flux emitted by ΔV_i and accepted by A_n can be expressed as

$$q_r'' = \kappa_a \Delta V_i \frac{\sigma T_a^4}{\pi} \frac{A_n}{4\pi (l_i + \frac{dl}{2})^2} \exp\left(-\sum_j^i \beta_j \cdot dl\right) \cdot (\hat{s} \cdot \hat{s}_n) \quad (3)$$

where κ_a is average absorption coefficient of gas in ΔV_i, T_a is the average temperature of the gas. l_i is the distance from the start point of the ray to x_i. β_j represent the extinction coefficient of the gas in each step of the ray. β_j is the vector of the ray and $\hat{s}_n$ is the normal vector of the mesh. The final absorbed heat flux by the surface is

$$q_{ab}'' = \varepsilon q_r'' \quad (4)$$

and ε is the absorption coefficient of the surface.

3 Result and Analysis

Figure 4 shows the distribution of soot in the tunnel, it can be seen that the high temperature soot will spread along the inclined tunnel to both ends. However, due to the effect of buoyancy, the hot soot will eventually flow out of the tunnel in the uphill direction, and in the downhill direction, it will form a boundary between the hot soot and the cool air in the perpendicular direction to gravity, and the angle between the boundary and the tunnel is about the slope of the tunnel.

The ignition source model and the Mudan method were used to evaluate the radiant heat flux at different locations from the fire source, the result is shown in table 1. As show in figure 3.

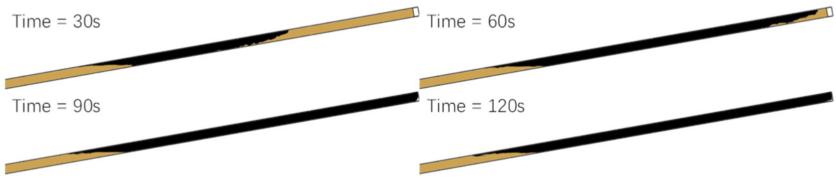

Fig. 3. Distribution of high temperature soot in tunnel at different stages of fire development

Table 1. The evaluation result of radiant heat flux with Mudan Method and Ignition Source Model

Distance to fire source(m)	Mudan Method (kw/m^2)	Ignition source model (kw/m^2)
2.6	9.09	11.3
3.6	5.03	6.04
4.6	3.22	3.77
5.6	2.25	2.58
6.6	1.67	1.88
7.6	1.29	1.43

8.6	1.03	1.13
9.6	0.84	0.91
10.6	0.70	0.75
11.6	0.60	0.64

When the distance exceeds 10 meters, the radiant heat flux is unable to cause harm to the human body. Therefore, this study focused on simulating the distribution of radiant heat flux on the human surface within a 10-meter radius of the fire source.

During the process of escaping from a fire, individuals typically flee with their backs facing the fire source. As a result, the 3D human mesh was positioned with its back towards the fire source during the simulation. The distribution of radiant heat flux on the back of the human body was simulated at four different stages of the fire (30s, 60s, 90s, and 120s). The human mesh was positioned at distances of 1, 5, and 10 meters from the fire source in both the upwind and downwind directions. The results are presented in Figure 4.

During the initial 60 seconds of fire development, a comparison was made between the distribution of radiant heat flux on the human body in the upwind and downwind directions of the fire source. It can be observed that the slope of the tunnel has an impact on the thermal radiation damage to the human body.

When the human body is positioned upwind of the fire source, the maximum radiant heat flux is concentrated on the lower part of the body, such as the legs. However, when the human body is positioned downwind of the fire source, the maximum radiant heat flux is concentrated on the upper part of the body, specifically the upper back and neck. This phenomenon occurs because, in the early stages of fire development, the smoke concentration in the tunnel is not high, and the distribution of radiant heat flux on the human body surface is primarily influenced by the shape of the flame and the relative position and orientation of the human body and the flame. The angle between the flame axis and the tunnel surface is acute due to the combined effects of buoyancy and the flow of hot air in the tunnel.

In the upwind direction of the fire source, as the flame moves farther away from the tunnel floor, the distance to the human body increases, resulting in greater attenuation of the emitted radiation on the human surface. Consequently, the radiant heat flux is mainly concentrated on the lower part of the human body. Conversely, in the downwind direction of the fire source, the distance between the flame and the human body decreases as the tunnel floor height increases, leading to less attenuation of the radiant heat flux emitted. As a result, the radiant heat flux is primarily concentrated on the upper half of the human body.

The rate at which the radiant heat flux decreases with distance from the fire source is also influenced by the accumulation of soot in the tunnel and the stage of fire development. In the early stage of fire development, even though the soot concentration is not high and the fire source is not fully burned, the radiant heat flux decreases rapidly with increasing distance from the ignition source. Similarly, during the full development stage of the fire, when the fire source is fully burned, the high smoke concentration in the environment also causes the radiant heat flux to decrease rapidly with increasing distance from the fire source.

(The fig.4-5 is replaced by the svg form)

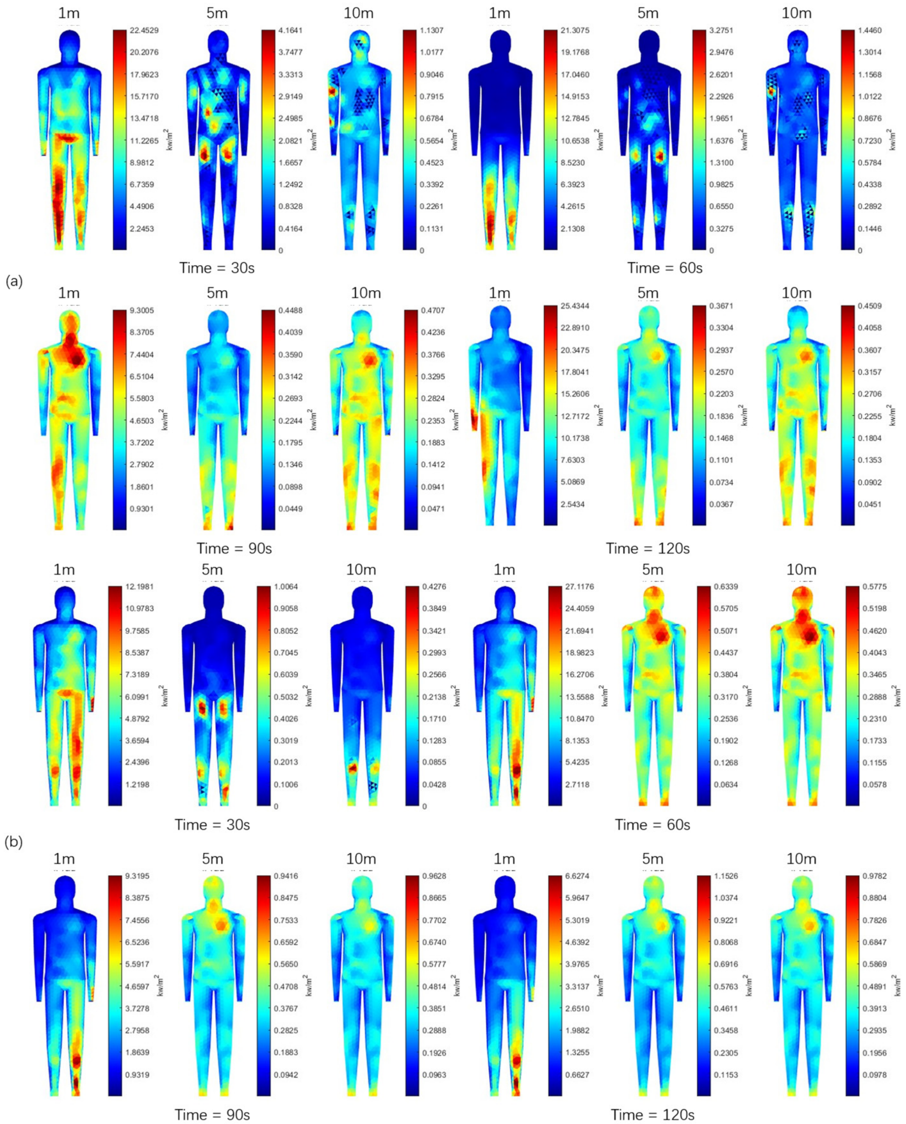

Fig. 4. The distribution of radiant heat flux on human body (a) In the upwind direction of the fire source;(b) In the downwind direction of the fire source

Consider the radiant heat flux on the surface of the human back as an example. At both the upwind and downwind directions and different stages of fire development, the radiant heat flux at each distance from the fire source is shown in Figure 5. In the stages of 30s, 90s, and 120s, whether the human body is positioned in the upwind or downwind direction, the radiant heat flux at 30s is greater than the others. This is because, in the early stage of fire development, even though the fire source is not fully burned and the flame is not fully developed, the soot concentration is much lower compared to that at 90s and 120s. As a result, the absorption and scattering of soot are weaker in the path of thermal radiation propagation, leading to a larger radiant heat flux on the surface of the human body at the same distance from the fire source.

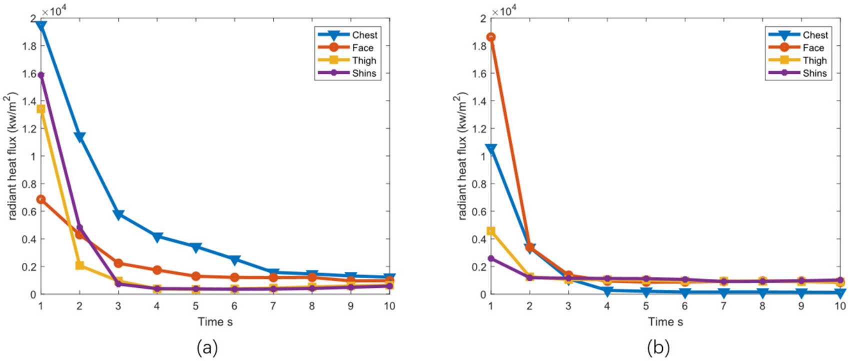

Fig. 5. Radiant heat flux on human back at different distance from the fire source. (a) In the upwind direction; (b) In the downwind direction.

However, the radiant heat flux in the 60s stage does not adhere to the aforementioned patterns. In the upwind direction, it is smaller compared to the other stages, while in the downwind direction, it is the largest. This is due to the fact that, in comparison to the 30s stage, the air in the tunnel is fully heated during the 60s stage, resulting in the formation of an air flow from low to high due to buoyancy in the inclined tunnel. Consequently, the shape of the flame is altered, leading to a reduction in the angle between the flame and the ground. As a result, there is a significant disparity between the radiation heat flux on the back of the human body at the same distance from the fire source in the upwind outlet direction and the downwind direction. At this juncture, the smoke concentration in the tunnel is not sufficiently high, and the shape of the flame still impacts the radiant heat flux on the human surface. As the fire progresses, the smoke concentration near the fire source in the tunnel increases, gradually eliminating this phenomenon.

4 Conclusions

In this study, a comprehensive three-dimensional numerical simulation model is proposed to assess the impact of tunnel slope and fire development stage on the distribution of radiant heat flux on the human body. The key findings are as follows:

1. The tunnel slope has an impact on the evacuation of personnel from a fire. When evacuating in the direction of the upwind outlet and the downwind outlet, the lower and upper parts of the human body are more exposed to thermal radiation from the

fire source, resulting in higher damage. Therefore, personnel evacuating in these directions should cover the corresponding parts of their skin as much as possible to minimize thermal radiation damage.

2. In inclined tunnel fires, the most significant thermal radiation damage to the human body occurs in the downwind direction during the intermediate stage of fire development. Simultaneously, when the fire is fully developed, the smoke concentration in the tunnel becomes excessively high. Although the thermal radiation damage is reduced, the presence of high-temperature soot poses a serious threat to the safety of personnel. Consequently, in the upwind direction, the early and intermediate stages of fire development are considered the safe escape stage, while in the downwind direction, only the early stage is considered the safe escape stage.

However, there are still some issues that need to be addressed in future research. For instance, in order to attain a high level of accuracy in calculations, a significant number of rays must be tracked. Despite the reduction in calculation time, real-time simulation remains unattainable. Additionally, this study does not simulate the actual impact of thermal radiation on the human body, and as a result, the direct thermal response of the human body cannot be obtained. This aspect requires further improvement in subsequent research endeavors.

Reference

1. Lei, P., Chen, C., Zhang, Y., Xu, T., & Sun, H. (2021). Experimental study on temperature profile in a branched tunnel fire under natural ventilation considering different fire locations. International Journal of Thermal Sciences, 159, 106631. https://doi.org/10.1016/j.ijthermalsci.2020.106631.
2. Zhang, X., Chen, L., Lyu, B., Tang, H., Peng, X., Lin, J., ... & Ren, F. (2024). An experimental study on the evolution of flame height and extension length in tunnel fires with arched ceiling confined condition. Tunnelling and Underground Space Technology, 150, 105809. https://doi.org/10.1016/j.tust.2024.105809
3. Król, A., & Król, M. (2021). Numerical investigation on fire accident and evacuation in a urban tunnel for different traffic conditions. Tunnelling and Underground Space Technology,109,103751. https://doi.org/10.1016/j.tust.2020.103751.
4. Barati, E., & Haghani, S. O. (2023). Studying the Synergetic Effect of Point-Extraction and Longitudinal Ventilation on the Maximum Smoke Temperature and Back-Layering Length in Tunnel Fires. Journal of Engineering, 2023(1), 8081874. https://doi.org/10.1155/2023/8081874.
5. Zhang, Y., Yan, Z., Zhu, H., Shen, Y., Guo, Q., & Guo, Q. Experimental investigation of pedestrian evacuation using an extra-long steep-slope evacuation path in a high altitude tunnel fire. Sustainable Cities and Society, 2019, 46: 101423. https://doi.org/10.1016/j.scs.2019.101423.
6. Kim, Y. J., & Trouvé, A. (2023). Evaluation of angular resolution requirements in the finite-volume-method solution of the radiative transfer equation. Fire Safety Journal, 141, 103971. https://doi.org/10.1016/j.firesaf.2023.103971
7. McGrattan, K., Hostikka, S., McDermott, R., Floyd, J., Weinschenk, C., & Overholt, K. (2013). Fire dynamics simulator user's guide. NIST special publication, 1019(6), 1-339.
8. GB/T 10000-2023, Human dimensions of Chinese adults[S].