



Numerical Study on the Effect of Blower Location on the Maximum Temperature and Spread of Smoke In Case Of Fire inside Tunnels

S. O. Haghani, E. Barati*

Department of Mechanical Engineering, Khayyam University, Mashhad, Iran

ABSTRACT: Designing an efficient emergency ventilation system is one of the main approaches to prevent the perilous fire in tunnel phenomenon. One of the most considerable factors in smoke control is critical velocity. In the present work, a parameter is called a critical volume flux, which indicates at least a volumetric flow that prevents smoke from flowing upstream of the fire. In this study, fire in tunnel is simulated using fire dynamics simulator code and the effect of blower location on maximum temperature, spread of smoke and critical volume flux of fire in the tunnel have been investigated. The results show that the blower location has a significant effect on critical velocity and volumetric flux and it can reduce critical volumetric flux by at least 11 percent. Also, considering different conditions in the blower system, the effect of fire source height has been investigated. The results also show that increasing the fire source's height does not have a significant effect on volumetric flux and critical velocity. The results show that the approach of the blower to the fire would reduce the smoke back-layering length and increase the maximum temperature.

Review History:

Received: 2019/03/26

Revised: 2019/07/08

Accepted: 2019/09/02

Available Online: 2019/09/12

Keywords:

Blower location

Fire

Critical volume flux

Maximum temperature

1- Introduction

With the prompt development of road tunnels around the world, the chance of tunnel fires has been boosted, which leads to excessive hazards to human life and the destruction of assets. Tunnels are one of the engineered structures in the roads which have many benefits such as reduction of travel time, and fuel consumption abatement, etc. for transport system users. Each minuscule accident in the tunnel closed medium, which may lead to no hazard in the other locations of the road, has capability to be a serious catastrophe. Accordingly, the transport system designers must proceed to secure the appropriate safety and control and opposite accidents in such engineered structures.

Research has been conducted in recent years to reduce the risk of fire in tunnels. Wu and Bakar [1] studied the flow of smoke in a tunnel fire using longitudinal ventilation. Chow et al. [2] studied the effects of tunnel slope. Mounesan et al. [3] studied the effects of fire source shape. In this study, the effect of blower location on maximum temperature, spread of smoke and critical volume flux of fire in the tunnel have been investigated.

2- Methodology

The numerical modeling is performed in a tunnel model with dimensions of $0.25 \times 0.25 \times 12$ m. There is a fire occurring in a tunnel, a burner is used as the fire source. The squared gas burner with dimensions of 0.1 m is employed as a fire source

*Corresponding author's email: Barati@Khayyam.ac.ir

to supply continuous heat release rates. The burner is placed on the center of the tunnel; ambient temperature is considered 20°C . Fig. 1 shows the geometrical representation of tunnel.

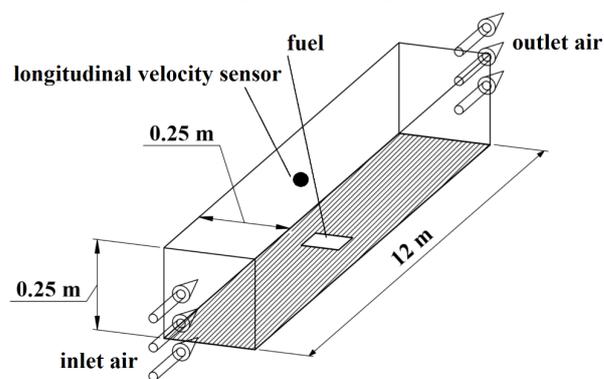


Fig. 1. The geometrical representation of the tunnel

2- 1- Numerical model

The computing power maturity and the extension of Computational Fluid Dynamics (CFD) have directed to the CFD applied to fire modeling. The numerical model is constructed by Fire Dynamic Simulator. The Fire Dynamics Simulator (FDS) has been developed at NIST to explore fire behavior and to analyze the efficiency of fire protection systems. Simulation of fire-driven flow can be conducted in FDS by employing the Large Eddy Simulation (LES) turbulence model. FDS has been extensively utilized in

research of smoke behavior and its validity has been broadly verified. Governing equations are solved numerically in FDS. The physical equations include Navier-Stokes equations for flow analysis, energy conservation equations for temperature distribution and other scalar equations for smoke and particulates transport.

To have a better calculation accuracy, mesh refining is performed. The FDS user guide proposes a non-dimensional expression of $D^* = \left(\frac{\dot{Q}}{\rho_\infty T_\infty c_p \sqrt{g}}\right)^{0.4}$ for assessing a mesh resolution with D^* . The recommended value of $\frac{D^*}{\delta x}$ [4] is in the range of 4-16. Fire grid numbers are studied to confirm that the results are grid independent. Along with the numerical simulations described in the next section, other simulations are conducted with the precise target of verifying the model validity, by examining the agreement between experimental results and model predictions. Fig. 2 shows the distribution of temperature in various computational mesh and Fig. 3 illustrates the variation of critical velocity with Heat Release Rate (HRR) and a remarkable agreement is observed and the simulation is compared with Wu and Bakar [1] and Li et al. [5].

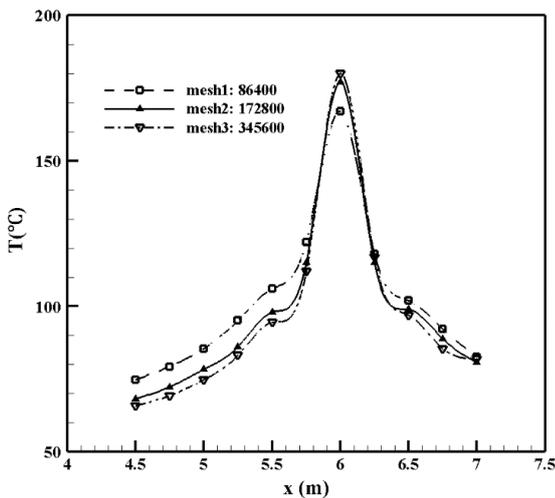


Fig. 2. Distribution of temperature in the various computations

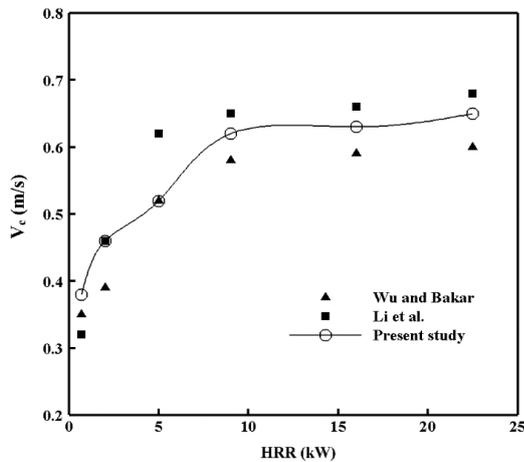


Fig. 3. Variation of critical velocity with HRR

3- Results and Discussion

Table 1 shows critical volume flux at different rates of heat release with four different blower systems.

Table 1. Critical volume flux at different rates of heat release

Critical volume flux (m ³ /s)				
Upper half	linear	parabolic	uniform	HRR (kW)
0.020	0.021	0.046	0.023	0.7
0.025	0.026	0.054	0.028	2
0.028	0.030	0.066	0.033	5
0.034	0.035	0.076	0.038	9
0.035	0.036	0.079	0.039	16
0.036	0.037	0.08	0.040	22.5

The results show that the blower location has a significant effect on critical velocity and volumetric flux and it can reduce critical volumetric flux by at least 11 percent.

Table 2 shows the Critical velocity values at different height of the fire source.

Table 2. Critical velocity values at different height of the fire source

Increasing velocity (m)	HRR (5 kW)		Height of fire source (m)
	critical volume flux (m ³ /s)	critical velocity (m/s)	
0	0.03	0.53	0
7.5%	0.035	0.57	0.1
11%	0.037	0.59	0.15

The results also show that increasing the fire source's height does not have a significant effect on volumetric flux and critical velocity.

Fig. 4 shows Maximum temperature in different conditions. As the blower system gets closer to the fire site, the maximum temperature increases.

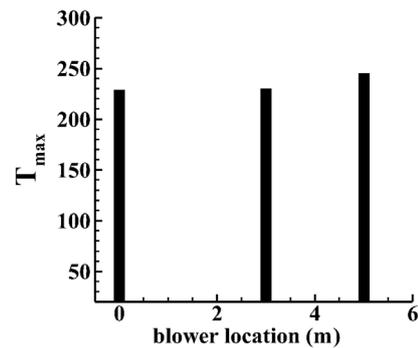


Fig. 4. Maximum temperature in different conditions

4- Conclusion

In this study, the effect of blower location on the critical speed and maximum temperature in fire tunnels was investigated. The results showed that the blower location has a significant effect on critical velocity and volumetric flux and it can reduce critical volumetric flux by at least 11 percent. In the study of the longitudinal distance of the blower system from

the fire source, in the case of back-layering smoke, the results showed that the approach of the blower to the fire would reduce the smoke back-layering length and increase the maximum temperature.

5- References

- [1] Y. Wu, M.A. Bakar, Control of smoke flow in tunnel fires using longitudinal ventilation systems—a study of the critical velocity, *Fire Safety Journal*, 35(4) (2000) 363-390.
- [2] W. Chow, K. Wong, W. Chung, Longitudinal ventilation for smoke control in a tilted tunnel by scale modeling, *Tunnelling and Underground Space Technology*, 25(2) (2010) 122-128.
- [3] M. Mounesan, M.R. Talaei, H. molatefi, Investigation of effective parameters on critical ventilation velocity in underground tunnels, *Amirkabir Journal of Mechanical Engineering*, 48(1) (2016) 41-54 (In Persian).
- [4] K. McGrattan, S. Hostikka, J. Floyd, H. Baum, R. Rehm, W. Mell, R. McDermott, *Fire dynamics simulator technical reference guide*, National Institute of standards and technology, (2010).
- [5] Y.Z. Li, B. Lei, H. Ingason, Study of critical velocity and backlayering length in longitudinally ventilated tunnel fires, *Fire safety journal*, 45(6-8) (2010) 361-370.

