STUDIES ON EVALUATION OF VENTILATION STRATEGIES OF UNDERGROUND METRO RAIL TRANSPORT SYSTEM

S. Jain[1], Shashi Kumar[2], Surendra Kumar[2] and T.P. Sharma[1]
[1] Fire Research Laboratory, Central Building Research Institute, Roorkee, -247667, U.A., India
[2] Department of Chemical Engineering, Indian Institute of Technology, Roorkee, -247667, U.A., India

ABSTRACT

The national capital of India, New Delhi has experienced a phenomenon growth in population in the last few decades. This resulted in development of rail based urban transport system for Delhi. The first phase of the project under execution comprises of 62.5 km of route length of which 12.5 km is underground and is called the Metro corridor. Therefore it becomes essential to study the evacuation procedures in this type of tunnel. Keeping this in mind, the present Research work is undertaken. The Proposed research work will investigate the fire dynamics in a section of underground tunnel, approximately similar to Metro corridor using Computational Fluid Dynamic (CFD) Technique. The objective of the paper is to evaluate the emergency ventilation strategies of a section of tunnel.

All Metro underground stations are approximately 300 m long and 20 m wide. Ventilation fans are located at the stations and not inside the tunnels. For analysis, it is assumed that the fire source of 4 MW is located at the center of the tunnel. In the event of fire it is assumed that the train will pass through and reach the station. Hence, dynamics of fire inside tunnel without train is only studied.

The numerical model used is first verified using experimental results from Steckler experiment. The model is then used to simulate the fire environment inside the tunnel. The results of CFD simulations are compared with those of empirical correlations available in the literature. The effectiveness of smoke ventilation system is then studied. For this the effect of ventilation flow rate, both uniform and non-uniform airflow in tunnel, on thermal environment inside the tunnel has been studied. This helps in determining the critical velocity necessary to prevent back layering. A ventilation scenario where both inlet and exhaust fans are activated is also studied.

KEYWORDS: Tunnel fires, Computational fluid dynamics, Ventilation

NOMENCLATURE

A	tunnel cross-sectional area (m^2)
D_f	diameter of flame (m)
Fr_c	critical Froude number (-)
h_c	heat transfer coefficient (W/m^{2}K)
H	tunnel height (m)
H_D	tunnel hydraulic diameter (m)
K_g	tunnel grade correction factor (-)
$\dot{m}$	smoke mass flow rate (kg/s)
Q	fire heat release rate (kW)
Q_c	convective heat release rate, kW; which is taken as 70% of Q
Q^*	dimensionless fire heat release rate (-)
T_0	ambient air temperature (K)
T_{cp}	plume centerline temperature (K)
T_f	gas temperature from fire (K)
V_c	critical ventilation velocity (m/s)
V^*_c	dimensionless critical ventilation velocity (-)
W	tunnel width (m)

x	distance from the fire source along tunnel axis (m)
Z	height above top of the fire source (m)
Z_0	height of virtual origin relative to the base of fire source (m)
ρ_0	ambient air density (kg/m^3)
C_p	specific heat (kJ/kg K)
g	acceleration due to gravity (m/s^2)
ΔT	average temperature rise at distance x (K)
ΔT_0	Temperature rise near the ceiling over the fire source (K)

INTRODUCTION

In Delhi, after the construction of Metro, millions of people travel through underground railway. Inside a tunnel, in case of fire, because of the limited area it is difficult for fire brigades to get near the ignition sources and control the fire. To ensure that tunnels are safe in case of fire, more fire protection systems are required.

Ventilation systems are one of the most important safety systems to be used in control of fires inside a tunnel. It controls the spread of heat and smoke and is essential in order to create a safe environment for evacuation of people and when fighting fires in tunnels. There are three types of ventilation systems, which are commonly employed. These are the longitudinal, the transverse, and the semi-transverse systems. The longitudinal system is more popular because the piston effect due to moving vehicles provides a cost free force to push air along the tunnel. In longitudinal systems, a series of jet fans are installed along the tunnel ceiling, which pushes the air forward in order to exhaust it to the atmosphere while make up fresh air flows into the tunnel.

The purpose of this study is to study the effect of ventilation flow rate on thermal environment inside the tunnel. This would help in determining the critical velocity necessary to prevent back layering. Two scenarios of ventilation are discussed. In the first case the induced airflow is assumed to be uniform; this corresponds to a case, where fans are located at sufficient distance from the fire source. In second scenario, the fans are located near the fire source resulting in non-uniform airflow at fire site. The numerical methodology is first verified using experimental results from Steckler experiment. The same model is then used to simulate the environment inside the tunnel. The results of CFD simulations are compared with those of empirical correlations available in the literature.

CFD MODELING APPROACH

A CFD model requires that the enclosure of interest be divided into small rectangular control volumes or computational cells. The CFD model computes the density, velocity, temperature, pressure and species concentrations in each control volume based on the conservation laws of mass, momentum, and energy. All the complex physical and chemical processes in fires need to be modeled. The transport equations therefore require sub-models to describe the complex processes of turbulence, combustion and thermal radiation. Approximations are applied for the treatment of heat and momentum transfer to the enclosure boundaries and in the numerical discretisation of the continuous partial differential equations. It is these approximations and sub-models that are to be verified.

Governing Equations of Airflow

The set of equations, which describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations [1-3]. These are partial differential equations, which have no known general analytical solution but have to be solved numerically. CFX uses finite volume technique to solve these equations.

Turbulence Model

For the closure of the above governing equations a turbulence model, typically a two-equation model [4] or a large-eddy simulation model [5] is required. The turbulence model must include the contribution of buoyancy force to the turbulent kinetic energy generation and dissipation.

Fire Source Model

Heat is released in fires through the complex combustion processes. There are two approaches to model the distributed heat release from a fire, namely the volumetric heat source model and various combustion models. The volumetric heat source representation of the fire source is simply modeled as a volumetric source term in the governing equation for enthalpy. It only considers heat and mass transfer caused by fire. VHMS uses a volumetric energy source for the heat release rate. The combustion models gives a more realistic representation of the fire source since it accounts for turbulent mixing of fuel and air, followed by their chemical reaction and heat release in the fire plume. The chemical reaction rate is usually fast in fires, and therefore the controlling mechanism of the reaction is the turbulence mixing of the gas phase reactants. For simplicity, the combustion process is represented as a single, one-step irreversible reaction. Xue et al.[6] compared different combustion models in enclosure fires.

Radiation Model

There exist two models for simulating the thermal radiation transfer in the fire room, namely the six-flux radiation model or P_1 model and the discrete transfer radiation model [3,7]. Between these two radiation models, the discrete transfer model is more accurate but is highly computationally intensive. P_1 model assumes that participating medium is gray and employs spherical harmonic P_1 approximation for solution of Radiative Transfer Equation (R.T.E). The discrete transfer radiation model (DTRM) integrates along a pre-set series of rays emanating from boundary faces, and assumes that surface radiation is isotropic.

STECKLER ROOM FIRE EXPERIMENT

The experiment being considered is one of many performed by Steckler et al., the details of which are given in their publication[8]. The case of burner providing a theoretical heat output of 62.9 kW is considered.

The CFD model consists of the enclosure and an extension of connected space outside the door. The flow is assumed to be steady and in fully turbulent regime. In the current study, the standard two-equation k-ε model of turbulence with buoyancy modification has been used due to its wide application in fire engineering problems [9,10]. Standard wall functions have been used to estimate the turbulence kinetic energy and dissipation at the cell near the wall. The fire source is modeled as VHMS model. In this model, the volumetric fire source is modeled as a cylinder of diameter 0.3 m, which is based on the diameter of burner. The spatial extent of the heat source has been assumed for the VHS model. Thermal radiation was represented with P_1 model. All walls are assumed to be smooth, adiabatic and gray, with an emmisivity of 0.5. The free pressure boundary conditions are applied on the extended domain boundary, where conditions are assumed to be ambient. The extended domain ensures realistic simulation of the plume outside the door opening.

Comparison with Experimental Results

In general, the numerical results are found to be in good agreement with the experimental results for velocity and temperature profiles at the door centerline and at corner [11].

On comparing the spatial distribution of the heat released within the fire plume when the height of the volumetric heat source has been reduced from 1.0 m to 0.05 m, but keeping the same fire base area, it is found that the increased height of the volumetric heat source prevents the tilting of the fire plume towards the wall. This is caused by the cool air jet entrained from the bottom of the door opening. The plume shape however appears more realistic for short heat source height, i.e. 0.2 m. This is because the fire tilts towards the back wall, forced by the makeup air through the door.

Table 1 shows that the peak temperature inside the fire plume increases with decreasing height over which heat is released in the VHS model.

TABLE 1. Maximum temperature in compartment using VHS of various source heights

Fire source volume height (m)	1.0	0.2	0.05
Temperature (K)	660	1160	2400

It has also been observed that the large differences in the predicted maximum fire source temperature in the VHS model (shown in Table 1) do not have significant influence on the predicted temperature distribution away from the fire source, i.e. at the door centerline and at the corner. From Table 1 it is also observed that for low heights of the volumetric heat source, peak temperatures inside the fire plume are unrealistically high because of limiting the heat distribution over very low source volume. Empirically, the height of flame given by Heskestad [12] is defined as:

$$Z_f = 0.235 * Q^{0.4} - 1.02 * D_f \quad [1]$$

For a fire of 62.9 kW and diameter of 0.3 m, this correlation estimates the flame length to be 0.93 m which is close to assumed length of 1 m. This length also gives more realistic temperatures. Thus, a reasonable assumption in case of VHMS model is that the height of the source should be close to that of flame length.

The results presented above illustrate the importance of assuming suitable height and volume of the volumetric heat source, appropriate to the heat release rate of the fire in order to have reasonable prediction of the flow pattern and the flame temperature.

The airflow and heat transfer through the enclosure door are more relevant for fire within a tunnel. Therefore, the agreement with the experimental results provides a verification of the numerical model to evaluate fire environment inside a tunnel.

FIRE INSIDE TUNNEL

The tunnel considered is a visualized one, similar to that of a Metro Rail System. At both ends of a typical station, the tunnels are of double track cut and cover type with a dividing wall to separate the two track ways. The tunnel ventilation system consists of tunnel ventilation shafts located at each end of the station, which are connected to track way through sidewall. Booster fans (jet fans) are located within the track ways to assist in controlling air movement within the track ways. The section considered is 100 m long, 6 m wide and 9 m high. The dimensions taken are approximate. All walls are assumed to be smooth, adiabatic and gray. A view of the computational domain of modeled section is shown in Fig. 1. For all simulations, a fire with a heat release rate of 4000 kW is used to represent vehicle fire. The fire source is assumed to be square, 12.5 m^2 in area and located at the middle of the tunnel. The Volumetric heat source height is taken as 2.5 m. Both the tunnel portals are open and assumed to be at atmospheric conditions.

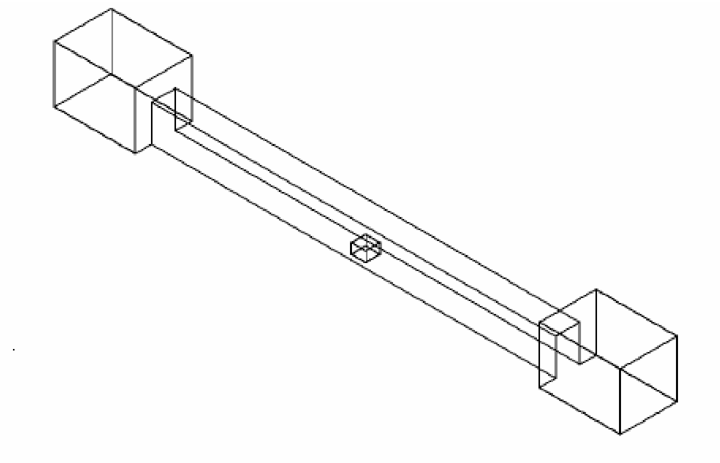
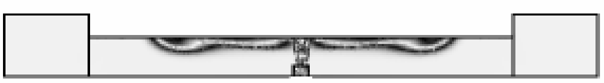


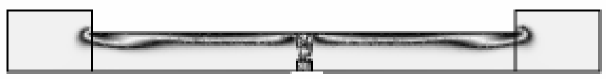
FIGURE 1. Computational domain of tunnel section with extended domain

The most significant effect of fire inside a tunnel is the buoyant effect caused by the difference of density between smoke and fresh air. This effect tends to create a layer of hot smoke and gases flowing away from the fire near the crown of the tunnel, while air supporting combustion moves towards the fire beneath the smoke layer. This can be observed from Fig. 2 which shows predicted temperature distribution on the vertical central plane through the fire source and tunnel portals at various times. It can be seen from the Fig. 2 that the smoke moves symmetrically along the crown in both directions and cool entrained air from bottom of tunnel portals move towards the fire source.

(a) Time = 10 s



(b) Time = 20 s



(c) Time = 30 s

FIGURE 2. Temperature distribution along central plane in the tunnel at various times (a) 10 s, (b) 20 s & (c) 30 s

Heskestad's correlation [12] is used to determine the plume centerline temperature. It is as given below:

$$T_{cp} = T_0 + 9.1\left(\frac{T_0}{g\,C_p^2\,\rho^2}\right)^{1/3} \frac{Q_c^{2/3}}{\left(z-z_0\right)^{5/3}} \qquad [2]$$

The height of virtual origin is defined as:

$$Z_0 = 0.235 * Q^{0.4} - 1.02 * D_f \qquad [3]$$

A comparison of predicted temperature by CFX and Heskestad's correlation is given in Fig. 3. It can be seen from the figure that temperatures are in close agreement away from the fire source, while at points near fire the temperature differs significantly. This can be because of assumed height of volumetric heat source. A lesser height will result in increased temperatures near the source. Also, in Heskestad's relationship, radiation loss is assumed to be 30% while in simulations radiation is being modeled. It also corroborates the fact, as done in Steckler experiment, that better modeling of fire source is necessary to predict the temperatures near the fire source.

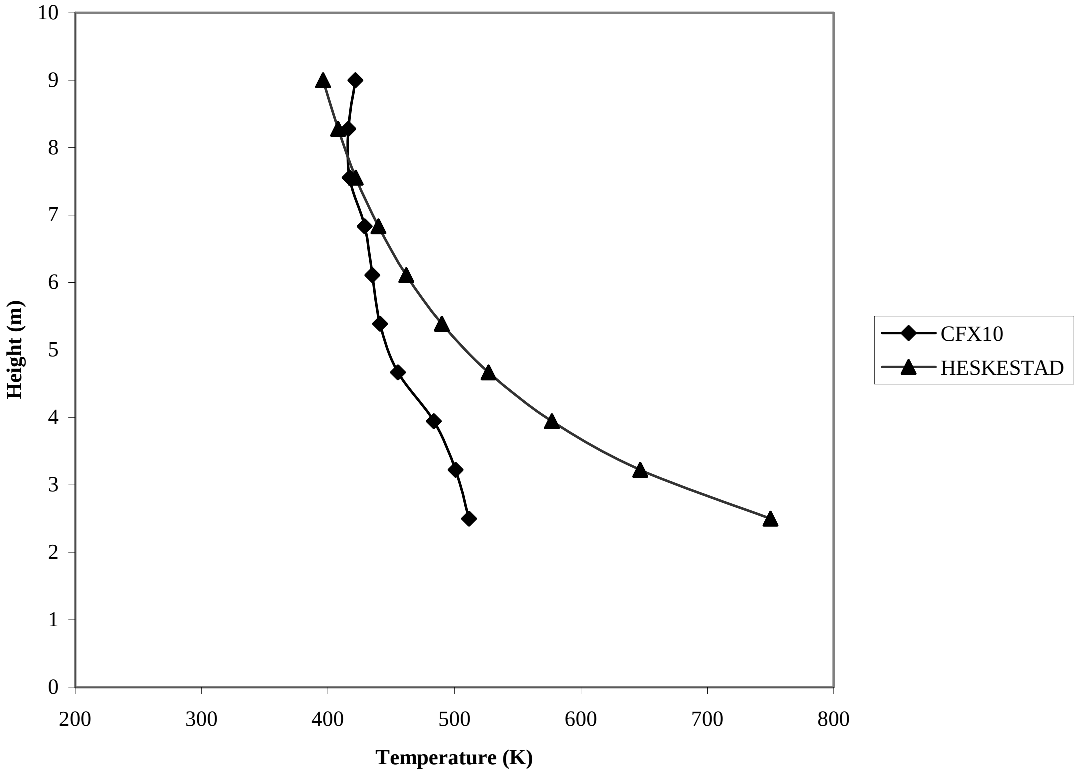


FIGURE 3. Comparison of predicted centerline temperature with Heskestad's correlation

Decay of longitudinal temperature along the tunnel is studied using CFD techniques. In tunnels the smoke temperature decays along the tunnel due to air entrainment and due to heat lost to boundaries. Delichatsios[13] studied spread of smoke under a beamed ceiling and deduced an exponential expression for temperature distribution along a beamed channel. Evers and Waterhouse [13] also established an exponential expression in terms of two parameters K_1 and K_2 as follows:

$$\frac{\Delta T}{\Delta T_0} = K_1 \exp(-K_2 x) \qquad [4]$$

Where the parameter K_2 is related to heat transfer coefficient h_c, corridor width W, smoke mass flow rate $\dot{m}$ and specific heat at constant pressure C_p by:

$$K_2 = \frac{K_1 h_c W}{\dot{m} C_p} \qquad [5]$$

The convective heat transfer coefficient in a tunnel is a combination of both natural and forced convection. Normally, it takes values from 5 to 50 W/m^{2}K for natural convection and 25-250 W/m^{2}K for forced convection in air [14]. It is difficult to determine convective heat transfer coefficients for smoke layer induced by fire, in spite of the fact that there exist a number of empirical formulae to determine convective heat transfer coefficient in air.

Fig. 4 shows the decay of longitudinal temperatures as computed from CFD simulations. For comparison with above formulae which uses zone modeling concepts, average temperature of smoke layer at distance x from the fire source is used. The height of smoke layer is taken to be 5 m from the crown of tunnel. This height is so chosen that below this height the temperature is approximately 40^{o}C. Fig. 4 also shows the exponential fitting of the temperature decay. From curve fitting, this equation comes out to be:

$$\frac{\Delta T}{\Delta T_0} = 0.3807 \exp\left(-0.001 * x\right) \qquad [6]$$

Comparing this equation with 4, assuming h_c to be 27.5 W/m^{2}K (average value of natural convection), $K_2 = 0.3807$ from curve fitting equation, the value of K_1 comes out to be 0.00056.

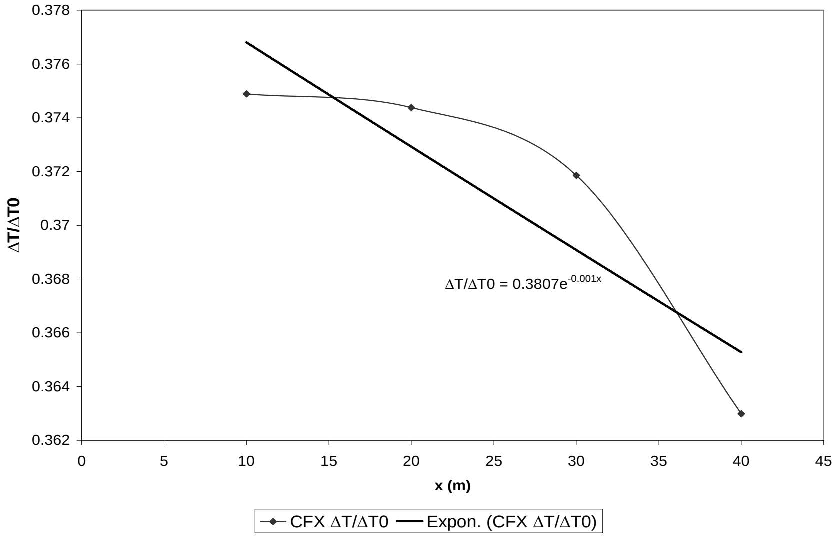


FIGURE 4. Longitudinal temperature decay of smoke layer from the fire source in the tunnel

Smoke Control in Tunnels - Tunnel Ventilation

The longitudinal ventilation system forces air to flow through the tunnel and it thus shifts the balance of heated air in the direction of the forced flow. Under strong ventilation, the hot smoke front tends to mix with surrounding cold fresh air, thus endangering the tunnel users downstream. If the ventilation is weak, the upper layer of heated air may flow in a direction contrary to the forced ventilation: this is the "back-layering” phenomenon. If the ventilation is of sufficient capacity, it will cause all of the heated air to flow towards the downstream direction. This principle is known as smoke control. The design principle of the ventilation system in emergency mode is the ability to prevent back-layering in the upstream area, to control the direction of smoke movement in order to provide a clear and safe path for evacuating people and to facilitate fire fighting operations. In the area, downstream of fire, a well-stratified smoke layer propagating along the tunnel is to be maintained.

The minimum air velocity required to prevent back layering and to force the hot air and smoke in the desired direction is called critical velocity. The critical velocity depends on the heat load of the fire and on the tunnel geometry. The critical velocity is the minimum steady-state velocity of the ventilating air moving towards the fire that would be required to prevent back layering.

In this section, a brief review of the correlation of critical velocity is given first, followed by the verification of the numerical results on a tunnel fire using these empirical formulae.

Correlations of Critical Velocity

The correlation of critical velocity is based on critical Froude number and a one-dimensional (1D) assumption. Two formulae are given here. The first one is from the subway environmental simulation program [15].

$$V_c = K_g \left(\frac{gHQ_c}{Fr_c \, \rho_0 \, C_p \, AT_f} \right)^{1/3} \qquad [7]$$

$$T_f = \frac{Q_c}{\rho_0 \, C_p \, AV_c} + T_0 \qquad [8]$$

where Fr_c is the critical Froude number, $Fr_c = 4.50$.

The other formula is a dimensionless correlation based on that by Oka and Atkinson [16].

$$V_c^* = K * \left(\frac{Q^*}{0.12} \right)^{1/3} \quad for \;\; Q^* \le 0.12 \qquad [9]$$

and

$$V_c^* = K \;\; for \;\; Q^* \ge 0.12 \qquad [10]$$

where the value of K varies from 0.35 to 0.31. For a fire on floor without significant blockage K is 0.35 and for fire that extends to full width of tunnel K = 0.31.

And the dimensionless HRR, Q*, and the dimensionless critical velocity, V_c^*, are given by:

$$V_c^* = \frac{V_c}{\sqrt{g\,H}} \qquad [11]$$

$$Q^* = \frac{Q_c}{\rho_0 \, C_p \, T_0 \, g^{1/2} \, H^{5/2}} \qquad [12]$$

The above correlation has been slightly modified by Wu and Bakar [15]. The correlation uses tunnel hydraulic height instead of height, and the limit of Q* proposed by them is 0.20. Thus the proposed formulae by Wu and Bakar is:

$$V_c^* = 0.40 * \left(\frac{Q^*}{0.20} \right)^{1/3} \quad for \;\; Q^* \le 0.20 \qquad [13]$$

and

$$V_c^* = 0.4 \;\; for \;\; Q^* \ge 0.20 \qquad [14]$$

where the dimensionless HRR, Q*, and the dimensionless critical velocity, V_c^* , are given by:

$$V_c^* = \frac{V_c}{\sqrt{g\, H_D}} \qquad [15]$$

$$Q^* = \frac{Q_c}{\rho_0\, C_p\, T_0\, g^{1/2}\, H_D^{5/2}} \qquad [16]$$

The details of critical velocity correlations and Froude modeling can be found in Grant et al. [17]

Critical Ventilation Velocity for Fire inside Tunnel

Based on the above formulae by Wu and Bakar, the critical velocity is 1.52 m/s. CFD simulations were carried out for different volumetric flow rates to study the influence of the volumetric flow rate on the thermal distribution. Three different volumetric flow rates of 80, 106, 132 m^3/s corresponding to velocities of 1.5, 2, 2.5 m/s respectively are taken. The airflow is assumed to be uniform and fully developed inside the tunnel.

Fig. 5(a-c) shows temperature distribution along the central plane for the case of different velocities. The results indicate that smoke back layering occurred for velocities of 1.5 m/s and 2 m/s and that the smoke layer covers a longer area when the flow rate in the tunnel is low. At flow rate of 2.5 m/s all the smoke moves upstream indicating that this flow rate is more than the critical velocity. This implies that critical velocity lies between 2 m/s and 2.5 m/s. To determine the critical velocity, CFD simulations were carried out with flow rates of 2.1 m/s, 2.18 m/s and 2.25 m/s. Fig. 5(d) shows temperature distribution along the central plane for velocity corresponding to 2.18 m/s. It can be seen from the figure that the critical velocity corresponds to velocity of 2.18 m/s. This velocity is about 43% higher than calculated from formulae.

The increased critical velocity may be because of the manner in which the fire source is defined. The fire source is defined as solid block of height 2.5 m, thereby reducing the tunnel cross section at fire source. This situation can be correlated with a stalled train in a tunnel, thereby reducing the tunnel cross section. Also in the formulae, heat release rate taken is convective which is assumed to be 70% of total heat release rate i.e. 30% heat is assumed to be lost through radiation whereas in CFD computations modeling of radiation are done.

Another set of CFD simulations were carried with the difference that airflow forced inside the tunnel is not assumed to be uniform. This corresponds to a case where jet fans are located near fire source. The area of the jet fan is taken to be 3 m x 2 m. To maintain airflow of 2 m/s in the tunnel, the velocity of jet fan using equation of continuity should be 18 m/s. However, this velocity comes out to be very high and forces smoke downstream rapidly. This becomes a case of strong ventilation. Different simulations are done in which the flow velocity of fan is taken as 6 m/s, 8 m/s, 10 m/s. The temperature distribution along central plane for the case of flow velocity of 8 m/s and 10 m/s are shown in Fig. 6(a,b). It can be seen from Fig. 6(a) that smoke back layering exists for velocity of 8 m/s. When velocity lies between 8 m/s and 10 m/s there is no back layering.

In the next set of simulations, both inlet and exhaust fans are activated with equal flow velocities. Two simulations are carried out for flow velocities of 10 m/s and 12 m/s. Their temperature distributions at the center plane are shown in Fig. 7(a,b). It can be seen from Fig. 7 that the stratified layer of smoke in downstream region is not formed. For inlet and exhaust velocity of 10 m/s the air flow tilts the plume downstream and its trajectory comes to floor in the downstream area from where the exhaust fans sucks this air to exhaust. The smoke thus remains at the bottom of the tunnel. As the flow velocity is increased, inlet air does not touch the floor downstream, rather it gets exhausted to the atmosphere, while smoke layer remains at bottom.

(a) v = 1.5 m/s

(b) v = 2.0 m/s

(c) v = 2.5 m/s

(d) v = 2.18 m/s

FIGURE 5. Temperature distribution along central plane in the tunnel for different inlet velocities when air flow is uniform

(a) v = 8 m/s

(b) v = 10 m/s

FIGURE 6. Temperature distribution along central plane in the tunnel for different inlet velocities when air flow is non uniform

(a) v = 10 m/s

(b) v = 12 m/s

FIGURE 7. Temperature distribution along central plane in the tunnel when velocity of both inlet and exhaust fan is (a) 10 m/s, (b) 12 m/s

CONCLUSIONS

The numerical model used for predicting the environment inside a tunnel is first validated using results of one of Steckler’s experiments. Then, using the same model parameters of fire environment inside a section of tunnel, assuming a fire source of 4 MW is predicted. It is found that the smoke moves symmetrically along the crown in both directions and cool entrained air from the bottom of tunnel portals move towards the fire source. The plume centerline temperature predicted from numerical simulations is compared with empirical relations given by Heskestad. It is found that temperatures are in close agreement away from the fire source. An exponential expression, similar to that given by Evers and Waterhouse for longitudinal temperature decay along the length of the tunnel is devised.

The effect of ventilation flow rate, both uniform and non uniform on thermal environment inside tunnel is analyzed. The critical velocity necessary to prevent backlayering is estimated using CFD and through empirical relations developed by Wu and Bakar. It is found that when flow inside tunnel is fully developed and uniform, CFD predicted higher critical velocity. This may be because of reduced cross-section of tunnel at fire site due to height of fire source or because thermal heat transfer through radiation is modeled. In absence of any data on heat loss due to radiation, it is assumed in empirical formulae that heat lost due to radiation is 30% of total heat release rate. It is also found that still higher critical velocities are required when flow is non-uniform, i.e. a case when fire source is near fan and flow is not fully developed. In such cases, it is observed that in regions downstream of fire, there is no well-stratified smoke layer. The environment inside tunnel is also analyzed when both inlet and exhaust fans are activated with equal flow velocities. It is found that even in this case well-stratified smoke layer in downstream region is not observed. At larger flow rates the air from fans moves at top and stratified layer is not formed in downstream area. It implies that only through upstream areas can escape take place.

REFERENCES

1. Patankar, S.V., Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corporation, Taylor & Francis Group, New York, 1980.
2. Cox, G. and Kumar, S., “Modeling Enclosure Fires Using CFD”, Chapter 3-8, in SFPE Handbook of Fire Protection Engineering, Society of Fire Protection Engineers, Quincy, Mass., USA, 2002.
3. AEA Technology, CFX 5.5.1 User Manual, 2002.
4. Cox, G., “Turbulent Closure and the Modeling of Fire Using CFD”, Phil. Trans. Roy. Soc., London, A356, 2835-2854, 1998.
5. McGrattan, K.B., Baum, H.R. and Rehm, R.G., “Large Eddy Simulation of Smoke Movement”, Fire Safety Journal, 30:2, 161-178, 1998.
6. Xue, H., Ho, J.C. and Cheng, Y.M., “Comparison of Different Combustion Models in Enclosure Fire Simulation”, Fire Safety Journal, 36:1, 37-54, 2001.
7. Kumar, S. and Cox, G., “Radiation and Surface Roughness Effects in the Numerical Modeling of Enclosure Fires”, in Fire Safety Science - Proceedings of the second International Symposium, pp. 851-860, 1987.
8. Steckler, K.D., Quintiere, J.G. and Rinkinen, W.J., “Flow Induced by Fire in a Compartment”, National Bureau of Standards, NBSIR 82-2520, 1982.
9. Kumar, S. and Cox, G., “Mathematical Modelling of Fire in Road Tunnels”, in Proc. 5th Intl Symposium on Aerodynamics and Ventilation of Vehicle Tunnels, BHRA, pp. 61-76, 1985.
10. Cox, G. and Kumar, S., “Field Modeling of Fire in Forced Ventilated Enclosures”, Combustion Science and Technology, 52, 7-23, 1987.
11. Jain, S., “CFD Analysis of Tunnel Fires”, PhD Thesis to be submitted in IIT-Roorkee, India in Dec 2007.
12. Heskestad, G., “Fire Plumes”, Chapter2-2, in SFPE Handbook of Fire Protection Engineering, Society of Fire Protection Engineers, Quincy, MA, USA, 1995.
13. He, Y., “Smoke Temperature and Velocity Decays Along Corridors”, Fire Safety Journal, 33, 71-74, 1999.
14. Drysdale, D., “Heat Transfer and Aerodynamics”, Chapter 2, in An introduction to Fire Dynamics, John Wiley & Sons, 1998.
15. Wu, Y. and Bakar, M.Z.A., “Control of Smoke Flow in Tunnel Fires Using Longitudinal Ventilation Systems – A Study of the Critical Velocity”, Fire Safety Journal, 35:4, 363-390, 2000.
16. Oka, Y. and Atkinson, G.T., “Control of Smoke Flow in Tunnel Fires”, Fire Safety Journal, 25:4, 305-322, 1995.
17. Grant, G.B., Jagger, S.F. and Lea, C.J., “Fire in Tunnels”, Phil. Trans. R. Soc., London, A356, 2873-2906, 1998.