International Journal on Engineering Performance-Based Fire Codes, Volume 5, Number 4, p.194-198, 2003

NUMERICAL STUDY OF REVERSAL FLOW IN TUNNEL FIRES

Jiangguo Chen, Haixin Chen and Song Fu
Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China

ABSTRACT

In the fire safety design of a tunnel, a longitudinal ventilation system is often installed to create a safe route for evacuation and fire fighting, which is clear of smoke and hot gas, in the case of a fire. In ventilated tunnel fires, smoke and hot gases may flow in the direction opposite to the ventilation air flow. The existence of the reverse flow endangers the fire fighting and evacuation in underground mine roadways, tunnels and building corridors. In this paper, a computational fluid dynamics (CFD) method is used to model floor-level fires in a ventilated tunnel. The computed velocity and temperature profiles with different fire source intensity and different tunnel width are obtained. These results show that, to provide an evacuation path, an appropriate ventilation air flux must be maintained. The "critical velocity", which is defined as the minimum air velocity required to suppress the smoke spreading against the longitudinal ventilation flow during tunnel fire, is also predicted and compared with the results obtained experimentally. Based on the tunnel hydraulic height, a relationship between non-dimensional critical velocity and non-dimensional fire heat release rate is proposed.

Keywords: tunnel fire, CFD, critical velocity

1. INTRODUCTION

Although there is increasing attention being paid to life safety during fire emergency in tunnels and mine roadways, there is no universally accepted design and operation criterion for fire emergency life safety systems. Consider the scenarios such as a vehicle on fire stopped in a tunnel, jamming the traffic and requiring worker and passenger evacuation, or a conveyer-belt fire in an underground mine entry, producing smoke and toxic combustion products. In the design for such a fire or smoke emergency, a main concern is maintaining an evacuation path that is free of smoke and hot gases. In ventilated tunnel fires, the risk from fires as well as the subsequent smoke movement depends largely on the ventilation flux applied. If the ventilation velocity is low, the smoke produced from the fire can travel in the upstream direction against the direction of the ventilation flow. Smoke may form a layer near the ceiling and flow in the direction opposite to the ventilation stream. This reversal of flow is called "backlayering". In some realistic fires, smoke and other combustion products have been observed to fill the tunnel for a considerable distance upstream the fire source. When the ventilation is not enough, the reverse stratified layer has an important effect on fire fighting and evacuation. It is of practical importance to understand the physical parameters and flow conditions under which the reverse stratified flow occurs.

The "critical velocity" is used to represent the value of the ventilation velocity which is just able to eliminate the "backlayering", and force all the smoke to move in the downstream direction. This value has become one of the prime criteria for the design of tunnel ventilation systems. Previous works employed simple empirical models to predict the critical velocity [1-5]. Generally, these models consider the buoyancy head and the dynamic head in the system, and deduce appropriate quantities for correlation. A recent review of tunnel fires by Grant et al. [6] pointed out that the existing experimental data still show an inadequate fundamental understanding of the interaction between buoyancy-driven combustion products and forced ventilation, the validity of extrapolating small-scale results to large scales, the influence of slopes and the tunnel geometry on smoke movement. They also pointed out that CFD-based models would be the best way for prediction purposes, because CFD models make few prior assumptions. They are based on the fundamental conservation equations in fluid mechanics, and provide, in theory, fine resolution of the problem in terms of both space and time for all the parameters of interest. With increasing availability of powerful computers, considerable attentions are given to CFD modeling of fire in buildings and ventilated tunnels [7-11].

In this paper, a CFD code is developed as an analysis tool to model floor-level fires in a ventilated tunnel. The fire source is simulated by a diffusion flame of propane issuing from a hole on the floor of a tunnel. The standard k-ε turbulence model is used in the three-dimensional channel flow. The distribution of flow and thermal physics parameters are obtained under different fire intensity. The relationship between "critical velocity" and fire heat release rate is also studied.

2. MODEL DESCRIPTION

2.1 Governing Equations

Three-dimensional, time-dependent viscous flow equations, the Navier-Stokes equations are used to calculate the flow and heat transfer characteristics within the tunnel fire. These equations describe the conservation laws of mass, momentum, energy, turbulence parameters and species, subject to the given boundary conditions.

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \mathbf{u} = 0 \qquad (1)$$

Conservation of species:

$$\frac{\partial}{\partial t}(\rho Y_l) + \nabla \cdot \rho Y_l \mathbf{u} = \nabla \cdot \left(\rho \frac{\mu}{Sc} \right)_l \nabla Y_l + W_l''' \qquad (2)$$

Conservation of momentum:

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} \right) + \nabla p = \rho \mathbf{g} + \mathbf{f} + \nabla \cdot (\mu \nabla \mathbf{u}) \qquad (3)$$

Conservation of energy:

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot \rho h \mathbf{u} - \frac{Dp}{Dt} = \dot{q}''' + \nabla \cdot \left(\frac{\mu}{\Pr} \right) \nabla h \qquad (4)$$

State relation:

$$p_0 = \rho R T \qquad (5)$$

Energy equation:

$$h = c_p T \qquad (6)$$

The above equations are written for a weakly compressible flow, which means that the fluid density changes as a result of changes in temperature or composition of the mixture, but not pressure. Pressure is, therefore, supposed to be thermodynamically constant and influence the fluid motion only through its spatial derivatives present in the momentum equations. This assumption filters out acoustic waves from the flow.

For the enclosure of Reynolds stress, the standard two-equation k- ε turbulence model is employed [12], in which additional equations for the kinetic energy of turbulent fluctuations k and the dissipation rate of turbulence energy ε are solved.

2.2 Combustion Modeling - Representing of Fire

A popular and rather successful example of combustion modeling is the Eddy-Break-up (EBU) model, originated from the works of Spalding [14-16] and subsequently developed by Magnussen and Hjertager [17]. Further research works have suggested that combining EBU turbulence combustion models with laminar Arrhenious models gives better prediction results. Therefore EBU-Arrhenious models have been adopted in the present code. The two-step chemistry reaction mechanism of propane/air was used.

$$C_3H_8 + 7/2O_2 = 3CO + 4H_2O, \quad CO + 1/2O_2 = CO_2 \qquad (7)$$

In the Arrhenious model, the methane and carbon monoxide reaction rates can be expressed as:

$$w_{C_3H_8,Arr} = -1.0 \times 10^{12} e^{-\frac{30 \times 4184}{RT}} \rho Y_{C_3H_8}^{\ 0.1} Y_{O_2}^{\ 1.65} \times \overline{M}_{C_3H_8} \qquad (8)$$

$$w_{CO,Arr} = -\left[12.35 \times 10^{10} e^{-\frac{40 \times 4184}{RT}} \rho^{1.75} Y_{CO} Y_{H_2O}^{\ 0.5} Y_{O_2}^{\ 0.25} - 5 \times 10^{8} e^{-\frac{40 \times 4184}{RT}} Y_{CO_2} \rho \right] \times \overline{M}_{CO} \qquad (9)$$

While in the Eddy Break Up model, the methane and carbon monoxide reaction rates have the form of

$$w_{C_3H_8,EBU} = -C_{r1} \left(\rho \frac{\varepsilon}{k} \right) \min \left(Y_{C_3H_8}, \frac{Y_{O_2}}{\beta_{C_3H_8}} \right) \qquad (10)$$

$$w_{CO,EBU} = -C_{r2} \left(\rho \frac{\varepsilon}{k} \right) \min \left(Y_{CO}, \frac{Y_{O_2}}{\beta_{CO}} \right) - \frac{28}{16} w_{C_3H_8,EBU} \qquad (11)$$

In EBU-Arrhenious model, the methane and carbon monoxide reaction rates were decided by:

$$w_{C_3H_8} = -\min[|w_{C_3H_8,Arr}|, |w_{C_3H_8,EBU}|]$$

$$w_{CO} = -\min[|w_{CO,Arr}|, |w_{CO,EBU}|] \qquad (12)$$

Here, $C_{r1} = 3.0$, $C_{r2} = 4.0$.

2.3 Boundary Conditions and Numerical Method

A steady-state simulation was performed in the present study. For the walls, the non-slip and adiabatic boundary condition is used. The wall-function is used to solve near-wall turbulence. At the upstream (inflow) side of the channel, a uniform air velocity *u* is specified (forced ventilation) with $v = 0$ and $w = 0$.

The equations were solved using Finite Volume Method. The difference scheme is the HLPA [18]. In the solving process, the SIMPLE method is used.

3. RESULT AND DISCUSSION

The channel used in the modeling is 7.1 m in length, 0.25 m in width, and 0.25 m for height, the same as the geometry in the experiment of Wu [13]. For a fixed channel geometry, the dominant parameters for the flow field are the ventilation velocity *u*, the fuel-exit velocity, the fuel-exit diameter and the channel inclination angle. The quantity is proportional to the rate of heat generation at the source fire. The fuel is injected through a 0.1 diameter floor opening located at a distance of 3.0 m downstream the ventilation air inlet. A three-dimensional body-fitted grid with 108 x 27 x 23 cells in length, width and height direction respectively was used.

Fig. 1 shows that when the value of u_{in} increases, the length of the reverse stratified layer decreases. Fig. 2 is the comparison of the CFD result and the experimental measured critical velocity versus the fire heat release rate in the tunnels. It can be seen that the numerical result have good agreement with the experimental result of Wu [13]. Fig. 3a shows numerical result under different tunnel width. Oka and Atkinson [5] used tunnel height *H* as the characteristic length in their analysis. Here tunnel A to C has the same height of 0.25 m, the tunnel width are 0.25 m, 0.5 m, 1.0 m respectively. A general expression presented by Oka and Atkinson [5]:

$$V' = 0.38\left(\frac{Q'}{0.12}\right)^{1/3},\ for\ Q' \le 0.12,$$

$$V' = 0.38,\ for\ Q' > 0.12 \qquad (13)$$

is also plotted in Fig. 3a. Here,

$$V' = \frac{V}{\sqrt{gH}},\ Q' = \frac{Q}{\rho_0 C_p T_0 \sqrt{gH^5}} \qquad (14)$$

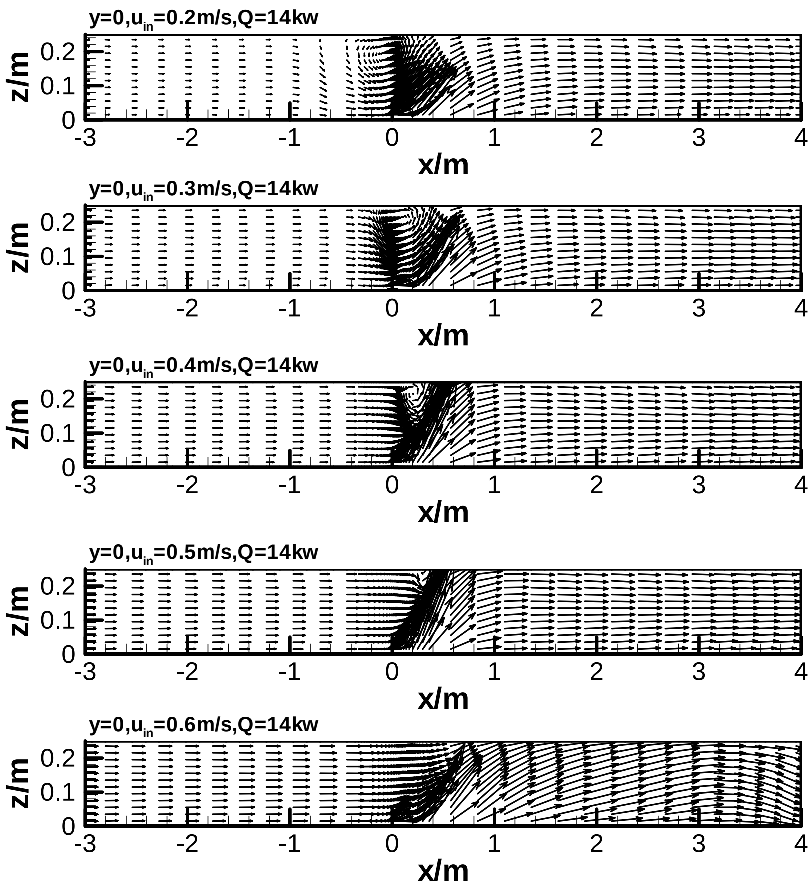


Fig. 1: Velocity vector plots showing the extent of reverse stratified layers for various flow conditions

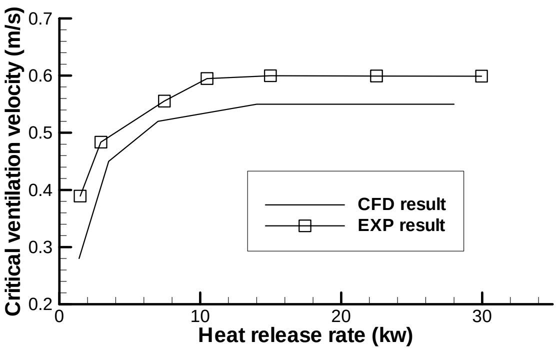


Fig. 2: Comparison of the CFD result and the experimental measured critical velocity vs. the fire heat release rate in the tunnels

Ris [19] find the mean hydraulic diameter $\overline{H}$, which is defined as the ratio of 4 times the cross-sectional area to the tunnel wetted perimeter, to be a better characteristic length to normalize the problem. Wu [13] has successfully used this length scale to normalize their experimental results. The calculated relation between dimensionless velocity V'' and dimensionless heat release Q'' is shown in Fig. 3b. The formula in this figure is presented by Wu [13]:

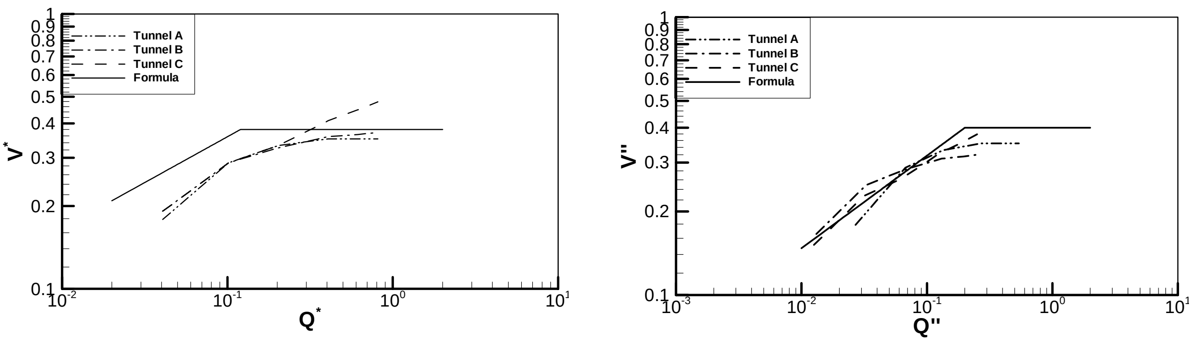
$$V''=0.40[0.2]^{-1/3}(Q'')^{1/3},\ for\ Q''\leq 0.20,$$
$$V''=0.40,\ for\ Q''>0.20 \qquad (15)$$

Here,

$$V''=\frac{V}{\sqrt{g\overline{H}}},\ Q''=\frac{Q}{\rho_0 C_p T_0\sqrt{g\overline{H}^5}} \qquad (16)$$

It has a good agreement with our computed results. From the two figures, we can see that the mean hydraulic diameter $\overline{H}$ is a better characteristic length than the tunnel height H to normalize the problem.

Fig. 4 shows that the distribution of CO_2 in CFD result can be compared with the actual smoke distribution in real fires.



(a) (b)

Fig. 3: The dimensionless critical velocity V* vs. the dimensionless heat release rate Q*
(a)The tunnel height is used as the characteristic length
(b)The hydraulic tunnel height is used as the characteristic length

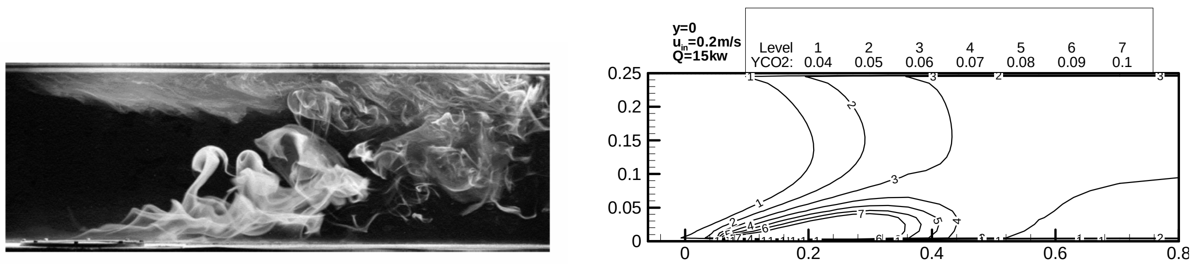


Fig. 4: Comparison of experimental photograph of smoke and CFD result of CO_2 contour distribution in a tunnel fire

4. CONCLUSION

A CFD method based on the standard *k-ε* turbulence model was used to model a floor-level fire in a ventilated tunnel. It has been shown that how CFD analysis can be used to determine a correlation between the length of the reversal flow region and tunnel ventilation. The resultant correlation can be used as the guidance for smoke control measures. For example, the correlations indicate that ventilation flux obeying to the one-third power of the fire intensity must be maintained to provide a clear and safe evacuation path from the fire source. This is an example of how a CFD modeling, when interpreted carefully and employed correctly, can become a useful design tool for fire protection. (Future investigation with other CFD codes and additional experiments should result in improved reverse flow length correlations.)

ACKNOWLEDGMENTS

This work was supported by the China NKBRSF project, no.2001CB409600.

REFERENCES

1. P.H. Thomas, "Movement of smoke in horizontal corridors against an air flow", Institution of Fire Engineers Quarterly, Vol. 30, pp. 45-53 (1970).
2. P.L. Hinkley, The flow of hot gases along an enclosed shopping mall: a tentative theory, Fire research note, No. 807, Fire Research Station (1970).
3. A.J.M. Heselden, "Studies of fire and smoke behavior relevant to tunnels", Proceedings of 2nd International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, March 1976, Cambridge, UK, pp. 23-25, BHRA Fluid Engineering (1976).
4. A. Guelzim, J.M. Souil et al., "Modeling of a reverse layer of fire-induced smoke in a tunnel", Fire Safety Science – Proceedings of the Fourth International Symposium, pp. 277-288 (1994).
5. Y. Oka and G.T. Atkinson, "Control of smoke flow in tunnel fires", Fire Safety Journal, Vol. 25, pp. 305-322 (1996).
6. G.B. Grant, S.F. Jagger and G.J. Lea, "Fires in tunnels", Philosophical Transactions of the Royal Society of London, Vol. 356, pp. 2873-2906 (1998).
7. P.J. Woodburn and R.E. Britter, "CFD simulation of a tunnel fire - Part 1", Fire Safety Journal, Vol. 26, pp. 35-62 (1996).
8. P.J. Woodburn and R.E. Britter, "CFD simulation of a tunnel fire - Part 2", Fire Safety Journal, Vol. 26, pp. 63-90 (1996).
9. D.F. Fletcher, J.H. Kent, V.P. Apte and A.R. Green, "Numerical simulations of smoke movement from a pool fire in a ventilated tunnel", Fire Safety Journal, Vol. 23, No. 3, pp. 305-25 (1994).
10. S. Kumar, G. Cox, "Mathematical modelling of fires in road tunnels", Proceedings of the Fifth International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels, May (1985).
11. C.J. Lea, "CFD modeling of the control of smoke movement from tunnel fires using longitudinal ventilation", The First International Symposium of IAFSS, August, ETH, Zurich (1995).
12. B.E. Launder and D.B. Spalding, "The numerical computation of turbulent flow", Computer Methods in Applied Mechanics and Engineer, Vol. 3, pp. 269-289 (1974).
13. Y. Wu and M.Z.A. Bakar, "Control of smoke flow in tunnel fires using longitudinal ventilation systems – a study of the critical velocity", Fire Safety Journal, Vol. 35, pp. 363-390 (2000).
14. D.B. Spalding, "Mixing and chemical reaction in confined turbulent flames", 13th Symposium (International) on Combustion, The Combustion Institute, pp. 649-657 (1971).
15. H.B. Mason and D.B. Spalding, "Prediction of reaction rates in turbulent premixed boundary-layer flows", Combustion Institute European Symposium, New York, Academic Press (1973).
16. D.B. Spalding, 16th Symposium (International) on Combustion, The Combustion Institute, pp. 1657-1663 (1976).
17. B.F. Magnussen and B.H. Hjertager, "On mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion", Proceedings of 16th International Symposium on Combustion, The Combustion Institute, pp. 719-29 (1976).
18. J. Zhu, "A low diffusive and oscillation-free convection scheme", Communications in Applied Numerical Methods., Vol. 7, pp. 225-232 (1991).
19. J. De Ris, "Duct fires", Combustion Science and Technology, Vol. 2, pp. 239-58 (1970).
20. C.K. Lee, R.F. Chaiken and J.M. Singer, "Interaction between duct fires and ventilation flow: an experimental study", Combustion Science and Technology, Vol. 20, pp. 59-72 (1979).
21. P.L. Hinkley, The flow of hot gases along an enclosed shopping mall - A tentative theory, Fire research note, No. 807, March (1970).