COUPLING SIMULATION ON SUBWAY TUNNEL SMOKE PROPAGATION

Wei Deng*, Xiaofeng Li, Yingxin Zhu

1 Department of Building Science, Tsinghua University, Beijing 100084, China

ABSTRACT

Subway is consisted of tunnels, platforms and ventilation shafts. The air flow between these structures forms a ventilation network. There is a need for fire modeling tools capable of both rapid and precise simulation for fire and smoke propagation in such ventilation network. Computational Fluid Dynamics (CFD) is widely used in subway fire performance design. However, the boundary conditions used in models are always assumed to be constant. The interactions between fire and ventilation network cannot be considered. In other word, we only know it is safe under a certain boundary condition but whether the emergency system can run effectively and provide such a boundary condition is unknown. This paper presents a new fire model to solve this problem. TNFIRE3 is a coupling model in which network model is combined with CFD model. Smoke propagation in fire tunnel and air flow in the whole line can be simulated simultaneously. TNFIRE3 capabilities include different kinds of fire sources, radiation, train motion and operations of emergency ventilation system. A demo case of subway tunnel fire has been carried out to illustrate how to design the emergency ventilation modes with TNFIRE3.

KEYWORDS

TNFIRE; coupling simulation; fire; subway tunnel; network

INTRODUCTION

The restrictions imposed by intensive city land use today require efficiency transport system. As a convenient and efficiency means of transport, subway has been developed rapidly in the past several decades. Many countries have built a sound subway system. With the continuing urbanization process, subway construction in China is in full swing. During all the potential dangers, fire can be the most serious one. Although large scale fire rare happen in subway, the event often leads to disastrous consequences once it happens. The most recent fire accident in the Daegu subway station in Korea caused 198 dead and the earlier one which killed 155 people occurred in Australia in 2000.

In order to control fire and smoke propagation, ventilation system is commonly adopted to help evacuation. In subway, emergency ventilation system is distributed in each platform and tunnel, once fire break out, some of which may work together because only one fan may be not enough. It is very important to understand whether the system capacity and the emergency mode can make smoke under control. However, fire scenarios are different from one case to another. Fire modeling tool for predicting is required. Compared to full-scale experiment, fire modeling tools are less expensive and more time saving. In a recent survey(Raymond 1992, Stephen 2003), about one hundred and sixty eight computer models were identified. They can be roughly divided into field model, zone model, network model and coupling model.

Field model is base on Computational Fluid Dynamics (CFD), Field model is quite capable in simulating the effects of fire and providing a detail and visual smoke movement. As for subway fires, CFD tools are commonly applied to evaluate the security in platform and tunnel fire. F. Chen(Chen 2003, Chen 2003) analyzed the smoke movement in a real platform in both natural ventilation case and mechanic ventilation case by commercial CFD software CFX4; P.Z.Gao et al.(Gao) compared the results predicted by LES and a k-ε model in a tunnel fire. D. Borello et al.(A CFD 2002) used PHOENICS to simulate a road tunnel fire, the result was compared to a full scale test data. Theoretically, CFD can be used for all fire scenarios, but because of the limitation of computational resources, CFD can only simulate a single tunnel or platform in previous papers, while the interactions between fire and ventilation network are neglected. Zone model is the simplified method of field model. Zone models usually divide a room into the upper hot gas layer and the nether cool gas layer. Within each of the two layers, the temperature, smoke and gas concentration are assumed to be exactly the same. It gets huge success in compartment fire simulation for its requirement of much shorter CPU time but reasonable prediction. W.K. Chow(Chow 1996) used CFAST to simulate a tunnel fire. It is found that prediction accuracy increases by dividing tunnel into smaller segments. The same as CFD model, however, zone model cannot deal with the emergency ventilation system neither.

Network model, which has a widely used in mine heading and subway environment simulation is the simplest model with one-dimensional assumption. It treats the air flow path as branches with uniform parameters while the joints of branches as nodes. It is easy to involve the whole ventilation system into consideration. Floyd et al.(Floyd 2005) developed a network fire model FSSIM. It can process a multiple compartment fire with complex HVAC system. However, the empirical equation for fire source may not fit in other conditions. SES(Subway 1997) and STESS(STESS 2000) are both subway environment simulation tools, although they can predict the air flow in ventilation system, the fire model are not available for detail prediction.

Coupling models which combined the two or three of field, zone and network model together is the best way for the simulation of interactions between fire and ventilation network. Network model can predict the air flow rate in each tunnel and provide an accurate boundary condition for CFD model, while CFD provide a mass, momentum and energy source for network model. Yao and Fan (Yao 1999) proposed FZN(Field-zone-network) model for fire simulation of high rise building. The model was applied to a 5-floor building fire. The result showed that FZN model could give a promised prediction. However the ventilation system cannot be considered in Yao's model. Li and Yan(Li 1995) evolved a field-network model into fire simulation in subway ventilation network with quasi-steady assumption.

This paper proposes a transit coupling approach which use CFD model near fire source and network model for the other parts of the ventilation network. The mathematic formulation and numerical algorithm is discussed and a demo case is carried out to illustrate the application of coupling model.

MATHEMATICS FORMULATION

CFD Model

Fire and smoke movement near fire source can be described by a set of partial differential equations which are derived from the conservation of mass, momentum, energy and species. The general form of control equation can be written as:

$$\frac{\partial(\rho\phi)}{\partial t} + div\left(\rho\vec{U}\phi - \Gamma_{\phi} grad\phi\right) = S_{\phi} \quad （1）$$

Where φ is the dependent variable, Γ_φ is the diffusion coefficient of φ, S_φ is the source term of φ.

In general, the velocity will not exceed

15m/s in fire case, Mach<0.1, however, the fluid density varied with temperature and species. It cannot be treated as incompressible flow. Thus, A set of so called low mach flow equations proposed by Rehm-Baum(Rehm 83)0 and Sivashinsky(Sivashinsky) are adopted here.

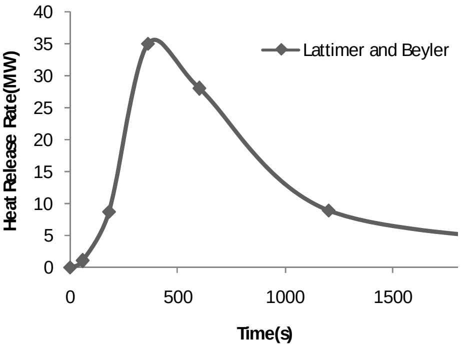

Fig 1. Heat release rate of tunnel fire

Commercial software package Phoenics is used for solving equations. For turbulence model, a modified k-ε turbulence model is selected. The radiation model and combustion model are neglected so as to save cost and time. Heat release rate is assumed as a super fast t square curves as Fig 1 shows.

In this study, the boundary conditions of solid walls, vents connected to network need to be specified. The solid is assumed as a non-slip wall with a fix temperature, and standard power law wall function was applied. For vent opening, because it is the interface between field model and network model, the boundary conditions are determined by coupling model.

Network Model

In subway ventilation system, tunnels, platforms and ventilation shafts are considered as the branches of the network, while the conjunctions of those structures became nodes of the network. Thus the network topology can be written as an incidence matrix A which represents the relation between nodes and branches and a basic circuit matrix B which represents the relation between circuits and branches. The element in matrix A and B is:

$$a_{ij}=\begin{cases}1 & \text{i node is at j branch, and air flow from i node to j branch}\\ -1 & \text{i node is at j branch, and air flow from j branch to i node}\\ 0 & \text{i node is not at j branch}\end{cases}$$

$$b_{kj}=\begin{cases}1 & \text{j branch is a part of loop k, and with the same direction}\\ -1 & \text{j branch is a part of loop k, and with the opposite direction}\\ 0 & \text{j branch is not a part of loop k}\end{cases}$$

So there are n×b elements in matrix A and (b-n)×b elements in matrix B, n is the number of nodes except reference node, b is the number of branches.

One-dimensional flow momentum equation in a branch can be written as:

$$\frac{d(\rho u)}{dt}=f_x-\frac{1}{\rho}\frac{dP}{dx} \qquad (2)$$

For a tunnel with uniform cross area, linear distribution of pressure and body force is assumed. So equation 2 can be written as:

$$\frac{d(\rho u)}{dt}=f_x-\frac{1}{\rho}\frac{DP}{L} \qquad (3)$$

Where f is the body force along flow direction:

$$f=\frac{Dh-{SM_i^2}/{\rho_i}-Dz_i(\rho_i-\rho_0)g}{\rho_i L_i} \qquad (4)$$

Where the first term is momentum source such as fan or train, the second term is the resistance of the tunnel and the last term is the buoyancy force.

Sustain equation 3 into 2 and write in vector form:

$$\overline{DP}=\overline{DH}-S|G|\overline{M}-g\Xi\overline{DZ}-E\frac{d\overline{M}}{dt} \qquad (5)$$

Where:

$$S = \begin{bmatrix} S_1 & & & 0 \\ & S_2 & & \\ & & \cdots & \\ 0 & & & S_b \end{bmatrix}$$

$$|G| = \begin{bmatrix} M_1/\rho_1 & & & 0 \\ & M_2/\rho_2 & & \\ & & \cdots & \\ 0 & & & M_b/\rho_b \end{bmatrix}$$

$$\Xi = \begin{bmatrix} \rho_1 - \rho_0 & & & 0 \\ & \rho_2 - \rho_0 & & \\ & & \cdots & \\ 0 & & & \rho_b - \rho_0 \end{bmatrix}$$

$$E = \begin{bmatrix} L_1/F_1 & & & 0 \\ & L_2/F_2 & & \\ & & \cdots & \\ 0 & & & L_b/F_b \end{bmatrix}$$

Apply backward difference scheme to equation 5 can get a set of difference equations:

$$\begin{aligned} \overline{DP} &= \overline{DH(t)} - S|G(t)|\overline{M(t)} \\ &\quad - g\Xi(t)\overline{DZ} - E\frac{\overline{M(t)} - \overline{M(t-\Delta t)}}{\Delta t} \end{aligned} \qquad （6）$$

The air flow in the network obeys two basic laws:

mass conservation in node：$A\overline{M} = \overline{Q}$ （7）

Kirchhoff law in a circuit：$B\overline{DP} = \Delta P$ （8）

$$\begin{cases} \Delta P = 0 & \text{without momentum source} \\ \Delta P = P_{ref,1} - P_{ref,2} & \text{with momentum source} \end{cases}$$

Where $\overline{M}$ is the branch mass flow rate vector, $\overline{Q}$ is net mass flow rate vector of node, $\overline{DP}$ is the branch pressure drop vector. Solving for $\overline{M}$ from eqs. (6) and (8) yields a set of non-linear equations as

(9): $$B\left[\overline{DH(t)} - S|G(t)|\overline{M(t)} - g\Xi(t)\overline{DZ} - E\frac{\overline{M(t)} - \overline{M(t-\Delta t)}}{\Delta t}\right] - \Delta P(t) = 0$$

（9）It can be derived from eqs. (7) that there are (b-n) dependent variables while there are (b-n) equations in eqs. (8), the equations have a unique solution. A steady algorithm MMKP(Zhu 1989) is adopted here for solving eqs. (7) and (9) simultaneously.

One-dimensional temperature dependence heat transfer is assumed at the wall. The temperature distribution in a branch is a function of both time and distance from inlet. The analytic solution is given as below:

$$\begin{cases} T = T_{soil} + (T_0 - T_{soil})\exp\left(-\dfrac{\alpha}{C_p \rho A} t\right) & t < \dfrac{x}{u} \\ T = T_{soil} + (T_{in} - T_{soil})\exp\left(-\dfrac{\alpha}{C_p \rho A u} x\right) & t \ge \dfrac{x}{u} \end{cases} \qquad （10）$$

Where T_{soil} is the temperature of soil leave wall for a distance with a constant value, T_0 is the initial temperature of branch which equal to ambient temperature here, Tin is the inlet temperature which is a function of time, x is the distance from inlet, α ,Cp, ρ ,A, u is the equivalent heat transfer coefficient of tunnel wall, thermal capacity, density, cross area and air velocity, for it takes not very significant effect to temperature change, they can be assumed to constant.

Supposed the air with different temperature can mix completely in nodes. The energy balance of a node can be represented as:

$$\sum_{j=1}^{n}\left[M_j \max\left(a_{i,j}T_{j,up}, 0\right) - M_j \min\left(a_{i,j}T_{j,down}, 0\right)\right] = 0 \qquad （11）$$

Where M_j is the mass flow rate at branch j, a_{ij} is the element of incidence matrix, $T_{j,up}$ is the temperature at the upstream of a branch while $T_{j,down}$ is the temperature at the opposite position.

Solving the simultaneous equations (10) and

(11) can get the temperature of all nodes and branches.

Coupling Approach

Coupling approach is the algorithm that links the field model and network model. The relationship between field model and network model can be described as the boundary conditions exchange. On one hand, the fire plume generates a throttle resistance to the air flow, and buoyancy force induced by hot gas in inclined tunnel should also be taken into account. Thus the fire tunnel can be treated as a common branch with an momentum source(sink) together with a mass source at its downstream node for a consist equation form.

On the other hand, for field model, the inlet and outlet boundary should be specified through network result. Network model provide mass flow rate at inlet and pressure distribution at outlet.

$$\begin{cases} \varphi_{inlet,\,field} = \varphi_{net} & if\,(net->\ field) \\ \dfrac{\partial \varphi}{\partial x}\Big|_{outlet,\,field} = 0 & if\,(\,field->net) \end{cases} \qquad （12）$$

Where

$\varphi = u, v, w, T, c...$

The network model of TNFIRE3 is developed in fortran90 language and embedded into the ground.for file of PHOENICS V3.5.1. as a module. The coupling simulation is realized by calling network module at each time step and changing the data.

NUMERICAL CASES

Fig 2 illustrates a part of subway including four platforms. Consider a fire occurred in the upstream tunnel between platform B and platform C. Suppose the fire occurs at the afterbody of the train with a heat release as Fig 1 shows. The new air is required to have a direction from platform C to platform B. The Fan capacity is 60m3/s each. As Fig2. shows, tunnel model in CFD is set to 300m long by 4.2m width by 5.5m height with a grid of 300 ×10×12. Total simulation time is 10 min, time interval is set to 1 sec. Two types of ventilation modes are simulated. The detail fan operation can be seen in Table.1.

Table 1 Ventilation modes

	Supply Fan	Exhaust Fan
Case1	C1,C2 parallel connection	B3,B4 parallel connection
Case2	C1,C2 parallel connection C3,C4	B3,B4 parallel connection B1,B2

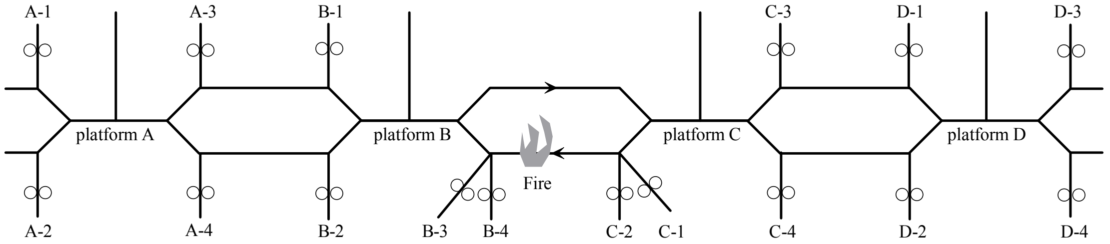

Fig 2. Illustration of subway ventilation network

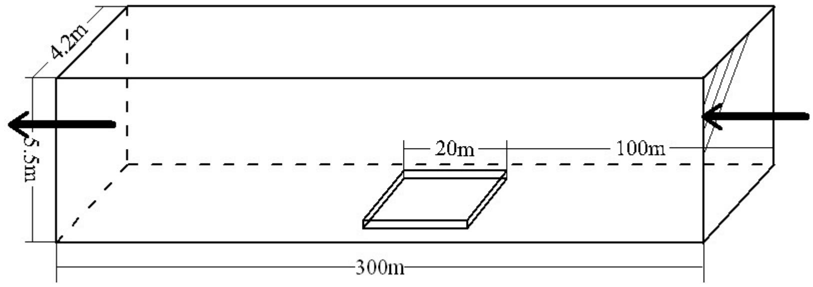

Fig 3. CFD model of fire tunnel

RESULTS AND DISCUSSION

Fig 4~8 shows the propagation of hot smoke represented by air temperature, all the left picture is from case 1 and the right one is from case 2. For each case, the temperature distribution at center section of 120s, 240s, 360s, 480s and 600s are given. It can be seen that with more fan active, the temperature of downstream decreases. The back layer of case 1 is obviously occur after 180s, while that of case 2 delay to occur at 420s. Although both of the back layers can be suppress in ten minutes, hot smoke goes 50m back in case1 which produce more risk for evacuation. Critical velocity which is function of heat release rate and tunnel dimension is always used to evaluate the security. If the inlet velocity is higher than the critical velocity, smoke will not form a back layer. The inlet velocity and critical velocity is drawn out in Fig 9. The velocity increases rapidly at first 60 seconds when the emergency ventilation system starts working from a static status or a reversed ventilation model and decreases with the fire heat release rate increasing. Although the inlet velocity of both cases do not satisfy the critical velocity, Case 2 provide a much higher inlet velocity because a higher pressure head generated by fans.

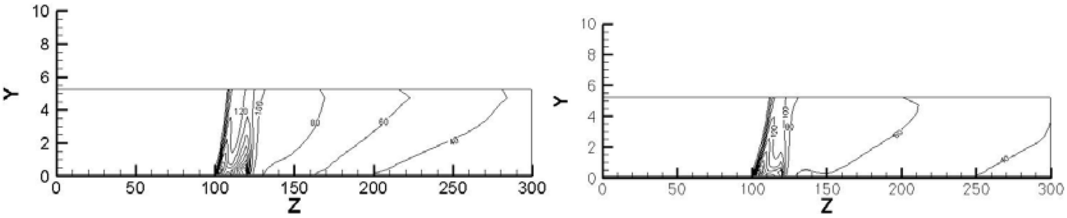

Fig 4. Temperature distribution and back layer at T=120s

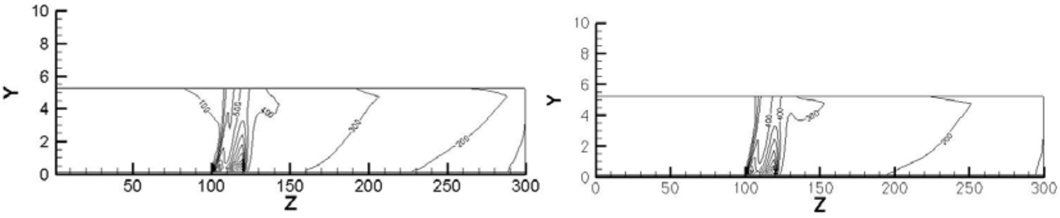

Fig 5. Temperature distribution and back layer at T=240s

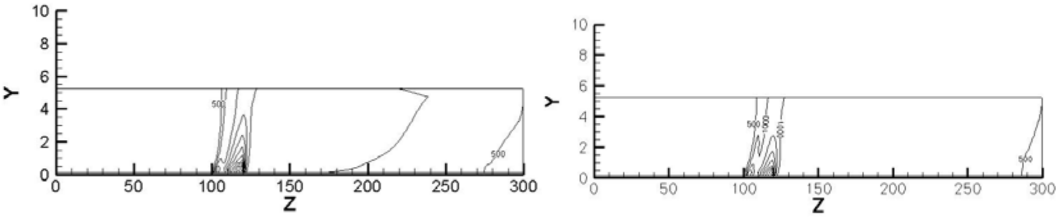

Fig 6. Temperature distribution and back layer at T=360s

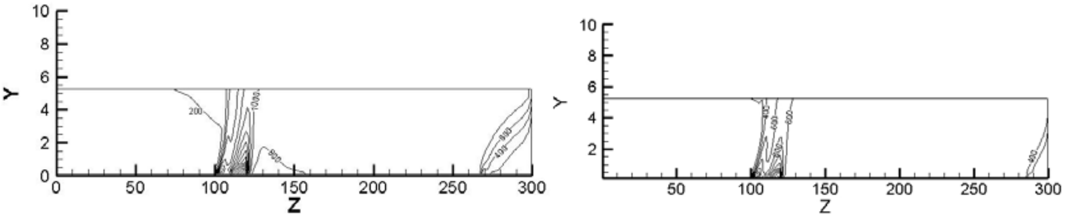

Fig 7. Temperature distribution and back layer at T=480s

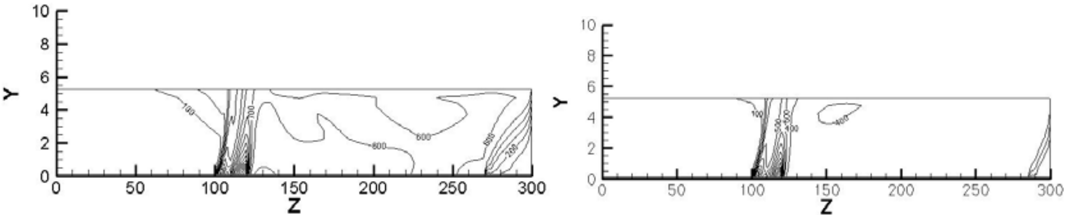

Fig 8. Temperature distribution and back layer at T=600s

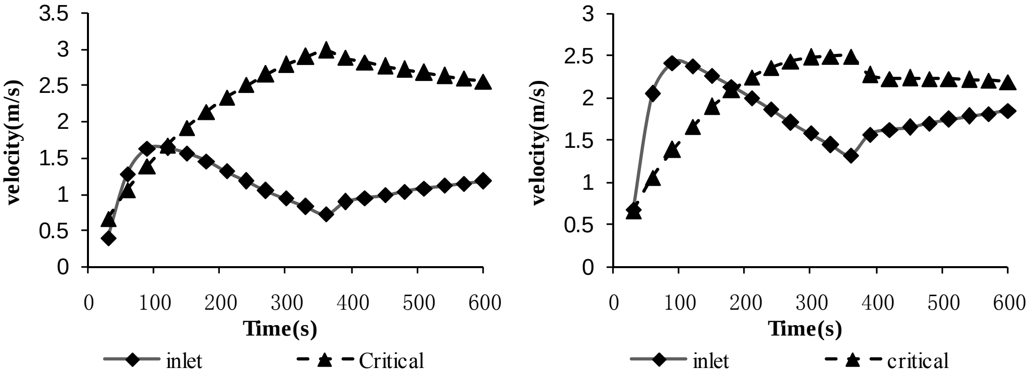

Fig 9. Velocity profile at inlet of field model and critical velocity

CONCLUSIONS

The theory of a new coupling model TNFIRE3 is present in this paper. The treat of coupling interface is also fulfilled. Two cases are carried out for a demonstration of using coupling model for subway tunnel emergency ventilation mode design. The results show that the new coupling model TNFIRE3 can give reasonable prediction of tunnel fire. Both the detail distribution of temperature, concentration of fire tunnel and air flow rate of the whole network can be got simultaneously. The interaction between fire and ventilation system can obviously be seen. Though the inlet velocity cannot reach critical velocity at some time, the back layer can be control if the critical velocity does not exceed the inlet velocity very much.

This paper has set a start of field-network coupling model for fire at complex space. The work of this paper mainly focuses on the modeling algorithm. Further work will be focused on the model validation and external capacity for field-network model with multiple interfaces, and more reliable CFD model should also be taken into account.

REFERENCES

Raymond Friedman. An international survey of computer models for fire and smoke. Journal of fire protection engineering. 1992 4(3):81-92

Stephen M. Olenick, Douglas J. Carpenter. An updated international survey of computer models for fire and smoke. Journal of fire protection engineering. 2003 5(13): 87-110

F. Chen, S.W. Chien, H.M. Jang, W.J. Chang. Stack effects on smoke propagation in subway stations. Institute of Applied Mechanics, National Taiwan University, Taipei, Taiwan, China. 106. Continuum Mech. Thermodyn. (2003) 15: 425–440

F. Chen, S.W. Chien, H.M. Jang, W.J. Chang. Smoke Control of Fires in Subway Stations. Institute of Applied Mechanics, National Taiwan University, Taipei, Taiwan, China. 106. Continuum Mech. Thermodyn. (2003) 15: 425–440

P.Z. Gao, S.L. Liu, W.K. Chow, N.K. Fong. Large eddy simulations for studying tunnel smoke ventilation.

A CFD Methodology for fire spread and radiative effects simulation in longitudinal ventilation tunnels: application to the memorial tunnel. International PHOENICS User Conference 23 – 27 September, 2002 Moscow

Chow W.K. Simulation of tunnel fire using a zone model. Tunnelling and Underground Space Technology ,1996 , 11(2) : 221-236

J.E. Floyd, S.P. Hunta. Network Fire Model for the Simulation of Fire Growth and Smoke Spread in Multiple Compartments with Complex Ventilation. Journal of fire protection engineering, Vol.15(2005): 199-229

Subway environmental design handbook, vol. II, Subway environmental simulation computer program, Version 4, Part 1, User's manual. DOT of USA, 1997.

STESS design handbook. Subway tunnel environment simulation software, Version 3, User's manual. Department of building science, Tsinghua university, 2000.

Jianda Yao. Verification and application of field-zone-network model in building fire. Fire Safety Journal; 33 (1999): 35-44

Xianting Li. Analysis of flame spreading over rod fuel and smoke movement in tunnel network. Phd thesis. Tsinghua university, Beijing, China. 1995

R.G. Rehm and H.R. Baum, The equations of motion for thermally driven buoyant flows, Journal of Research of the National Bureau of standards, 83(3), 298-308

G.I. Sivashinsky. Hydrodynamic theory of flame propagation in a enclosed volume, Acta Astronautica, 6, 631-645

Yingxin Zhu. Calculation method for dynamic process of flow in hydrodynamic network. Journal of Tsinghua university, Vol.29(1989): 72-78