International Journal on Engineering Performance-Based Fire Codes, Volume 1, Number 3, p.168-177, 1999

DESIGN AND OPERATION OF TUNNEL VENTILATION SYSTEM UNDER FIRE SCENARIO

S. C. Kot
The Hong Kong University of Science and Technology, Hong Kong, China

Raymond M. H. Yau and Dennis C. W. Tang
Advanced Engineering, Ove Arup & Partners Hong Kong Limited, Hong Kong, China

ABSTRACT

The approach to tunnel ventilation system design under fire scenarios has been a controversy for engineers for many years. This paper discusses the advantages and disadvantages of different approaches like full-scale fire test, empirical method, tunnel network modelling and computational fluid dynamics technique. Example will be used to compare the results obtained from different approaches.

CFD simulations have been attempted for fire simulation inside tunnels. Much of the current fire modelling techniques and assumptions are based on simulations and experience established from other areas. Validations against full-scale tunnel fire experiment are few. This paper presented one of the recent researches in the area, which form part of the continuous 'validation' of the CFD code for this kind of application. Recommendation based on the results will be made on the operational requirement of tunnel ventilation system under fire scenario.

1. INTRODUCTION

In the East Asia Region, particularly Hong Kong, there is a rapid development in the building industry over the last decade. The growth is expected to continue in the coming years. Infrastructure projects involving the increasing number of tunnels are growing in an even more rigorous rate. Many of the new works involve both longer and wider tunnels with sophisticated tunnel ventilation systems. As a result of increasing number of vehicles passing through tunnels, the risk of tunnel fire is higher than ever before. Furthermore, there is a significant reduction in vehicular emissions over the past decades, the fresh air ventilation rate required to meet the contaminant level criteria in road tunnels is on a decreasing trend. Therefore, the determination of ventilation control for a tunnel under fire scenarios will in many cases become critical in the design for the capacity of tunnel ventilation system.

The approach to tunnel ventilation system design has been a controversy for engineers for many years. The present methodologies and criteria used for quantifying the capacity of emergency tunnel ventilation in tunnel fire scenarios are mostly based on a simple empirical method, one-dimensional fluid dynamics network model and individual judgement and experience, without comprehensive testing. As a result, a number of opinions prevail regarding the capabilities of various types of ventilation systems to effectively manage heat and smoke.

To overcome this concern, the Computational Fluid Dynamics (CFD) technique, which has been well-established and applied in the design of building ventilation systems, enables the prediction of fundamental field variables of pressure, velocities, temperature and smoke concentration at any location within the computational domain of the system being simulated. However, due to the unique environment inside the tunnel, validation of simulation results is currently one of the most important issues in applying the CFD technique as a tool for emergency tunnel ventilation design.

2. DESIGN CRITERIA

There are three major limiting conditions for tenability under fire scenario, namely, temperature, carbon monoxide concentration and visibility (smoke). Previous researches have shown that the obstruction in visibility due to smoke will be the most critical factor among the three. According to CIBSE Guide E [1] and Technical Memoranda TM19:1995 [2], the visibility should be kept above 8 m to 10 m under all conditions for escape. In addition, temperature should be keep at below 80°C, which can be tolerated for 15 minutes to make evacuation possible [3].

In longitudinal tunnel ventilation system, directional control of smoke is usually adopted to provide a smoke free path for the evacuation of passengers [4]. The minimum air velocity required to control the smoke from backlayering is known as critical velocity.

3. EXPERIMENTAL METHODS

Validations of the current practice, especially against full-scale tunnel fire experiment, are few. Very few full-scale tunnel-fire experiments have been carried out and also in many cases, only temperature was measured. In addition, in many previous experiments, the number of data recorded was always far less than that generated from simulation. A detailed comparison between experiments and CFD was therefore impossible.

Recently, the Memorial Tunnel Fire Ventilation Test Program (MTFVTP) [5], being managed by the Massachusetts Highway Department, was undertaken to provide full-scale fire test results to examine the effectiveness of different tunnel ventilation systems on smoke control. The Memorial Tunnel, located near Charleston, West Virginia, is a two-lane, 853m long, mountain tunnel having a 3.2 percent upgrade from the south to the north tunnel portal. A total of 98 fire ventilation tests were conducted with different fire loads in the tunnel. Various tunnel ventilation systems and configurations were operated to evaluate their respective smoke and temperature control capabilities.

For example, there are a total of 15 tests related to longitudinal ventilation tests with jet fans in the MTFVTP for different fire load, fan starting time, fan reversibility, the operation of foam suppression system and the effect of vehicle silhouettes. Results suggested the critical velocity required to prevent backlayering of smoke is between 2.54 m/s to 2.95 m/s for fire load ranged from 10 to 100MW respectively.

4. EMPIRICAL METHOD

The theoretical critical air velocity [6-8] was calculated based on a one-dimensional formulation and Froude number concept. The model couples the energy equation, mass conservation and scale model test results, and inserts them into a Froude number formulation. Lee, Chaiken and Singer [7] stated that if the Froude number is kept at or below a certain critical value that smoke and hot air from the fire will not spread opposite to the direction of the ventilating air. Because the lower the Froude number, the weaker the buoyant forces become. The resultant equations used to evaluate the critical velocity are as follows:

$$V_c = K_1 K_g \left(\frac{gHE_C}{\rho c_p A T_f} \right)^{1/3} \quad (1)$$

$$T_f = \frac{E_c}{\rho c_p A V_c} + T \quad (2)$$

$$K_1 = Fr_c^{-1/3} \quad (3)$$

$$K_g = 1 + 0.0374(grade)^{0.8} \quad (4)$$

And the grade is the absolute value of the slope grade downward expressed as a percent.

The more conservative value of 4.5 [7] for Froude number is commonly adopted for the calculation of critical velocity in practice.

The calculation results based on Memorial Tunnel configuration are showed in Fig. 1. The increase in fire load will lead to the increase in critical air velocity in order to prevent smoke backlayering. For fire less than 20 MW, smoke backlayering can be controlled by an air velocity of 2.38 m/s. For fire up to 100 MW, the backlayering can be controlled by a longitudinal air velocity of 3.4 m/s.

Some latest researches [e.g. 9] proposed a revised set of empirical equations to determine the critical velocity. Although their new sets of equations showed a better agreement with experiment, comprehensive tests have not been carried out for the validation and therefore have not been widely adopted in the industry.

5. NUMERICAL SIMULATION

There are two major types of numerical simulation model adopted for the design of tunnel ventilation system, namely, the one-dimensional tunnel network model like SES or THERMOTUN, and the Computation Fluid Dynamics (CFD) model.

The one-dimensional tunnel network model provides estimate of transient airflow, temperature, humidity and pressure for the subway tunnel network system in a one-dimensional manner. It permits the user to simulate a variety of train propulsion and braking systems; various environmental control systems including forced air ventilation, station air-conditioning, and trackside exhaust; airflow in any given network of interconnected tunnels and stations; any sequence of train operation; various steady-state and non steady-state heat sources; emergency situations with trains stopped in tunnels and air movement by mechanical ventilation and buoyant forces.

Although detailed dynamical flow behaviour, like hot smoke flow from a tunnel fire could not be fully visualised by using one-dimensional network model, it has an advantage of being extensively used and well validated. For example, the SES program was field validated in Montreal and Toronto and has been applied to rapid transit or rail systems in many cities in the USA and the East Asia region including Hong Kong.

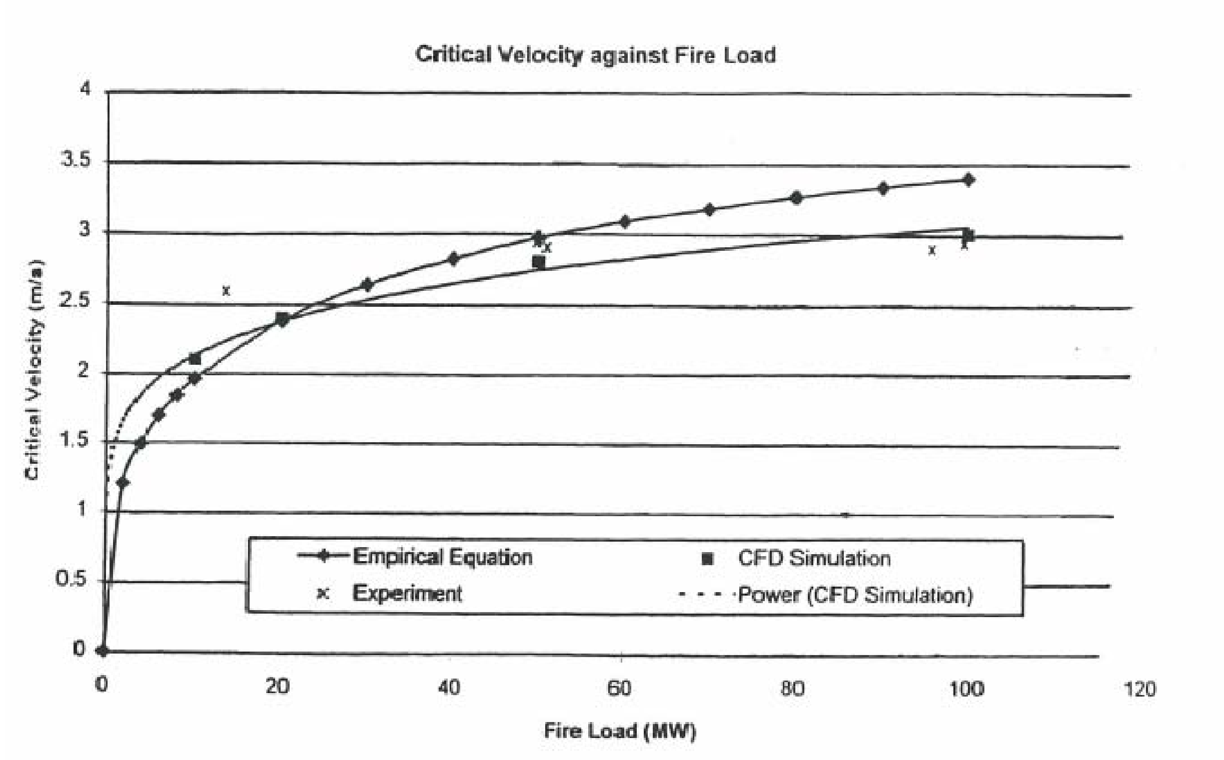


Fig. 1: Critical velocity against fire load

CFD simulations have been attempted for fire simulation recently due to the advances in computer hardware and software reliability and efficiency. The CFD technique enables the prediction of fundamental field variables of pressure, velocity, temperature and smoke concentration at any location within the three-dimensional computational domain. Much of the current fire modelling techniques and assumptions are based on simulations and experience established from other areas. Most of them use standard k-ε turbulence model, with convective heat transfer and wall model. Sometimes radiative heat transfer and combustion models are also included.

In one of our recent researches, the Star-CD V2.3 was used to simulate the Memorial Tunnel under fire emergencies. Star-CD is a commercial CFD software written by Computational Dynamics Limited, United Kingdom and has been used previously for other tunnel fire simulations [10].

6. CASE STUDY USING CFD TECHNIQUE

6.1 The Model

A CFD model consisted of about 28,000 cells using standard k-, turbulence model, with convective heat transfer and wall model was constructed to simulate a car accident inside a tunnel where fire was involved. The tunnel is modelled by the three-dimensional finite volume unstructured mesh. Simulation was carried out in a transient manner (Refers to appendix for the governing equations). The details of accident are described below:

- Before time equal to 0, the tunnel was under normal operation with two lanes traffic with different types of vehicles.
- At time equal to 0, a car and a bus on opposite lane crashed into each other at location about 240m away from the south portal. All the cars ahead of the accident location left the tunnel and those behind the accident location stopped one after another. Simulation started at time equals to 0.
- A 50 MW fire then started within the tunnel at the car-crash location. The fire was assumed to burn at a constant heat release rate of 50 MW. Vehicles of different sizes which represented by blockages were included to simulate the effect of stopped cars in the real situation. No tunnel ventilation fans were in operation between time equal to 0 to 60 seconds.
- Tunnel ventilation jet fans upstream (on the north portal side) of the fire site were then brought into operations at full speed 60 seconds after the start of the fire. The tunnel ventilation fans produced a longitudinal air velocity of 4.0 m/s (which is approximately equal to the recommended minimum ventilation rate for smoke and heat management by the ASHRAE, in its HVAC Applications Handbook) along the tunnel which

exceeded the critical velocity in order to control the flow of smoke and prevent smoke backlayering.

- The simulation was continued until the tunnel environment (in term of smoke flow, temperature and velocity) has reached its steady state.

Fig. 2 shows a view into the location of car accident CFD model. The blockages represent the stalled cars behind the incident location.

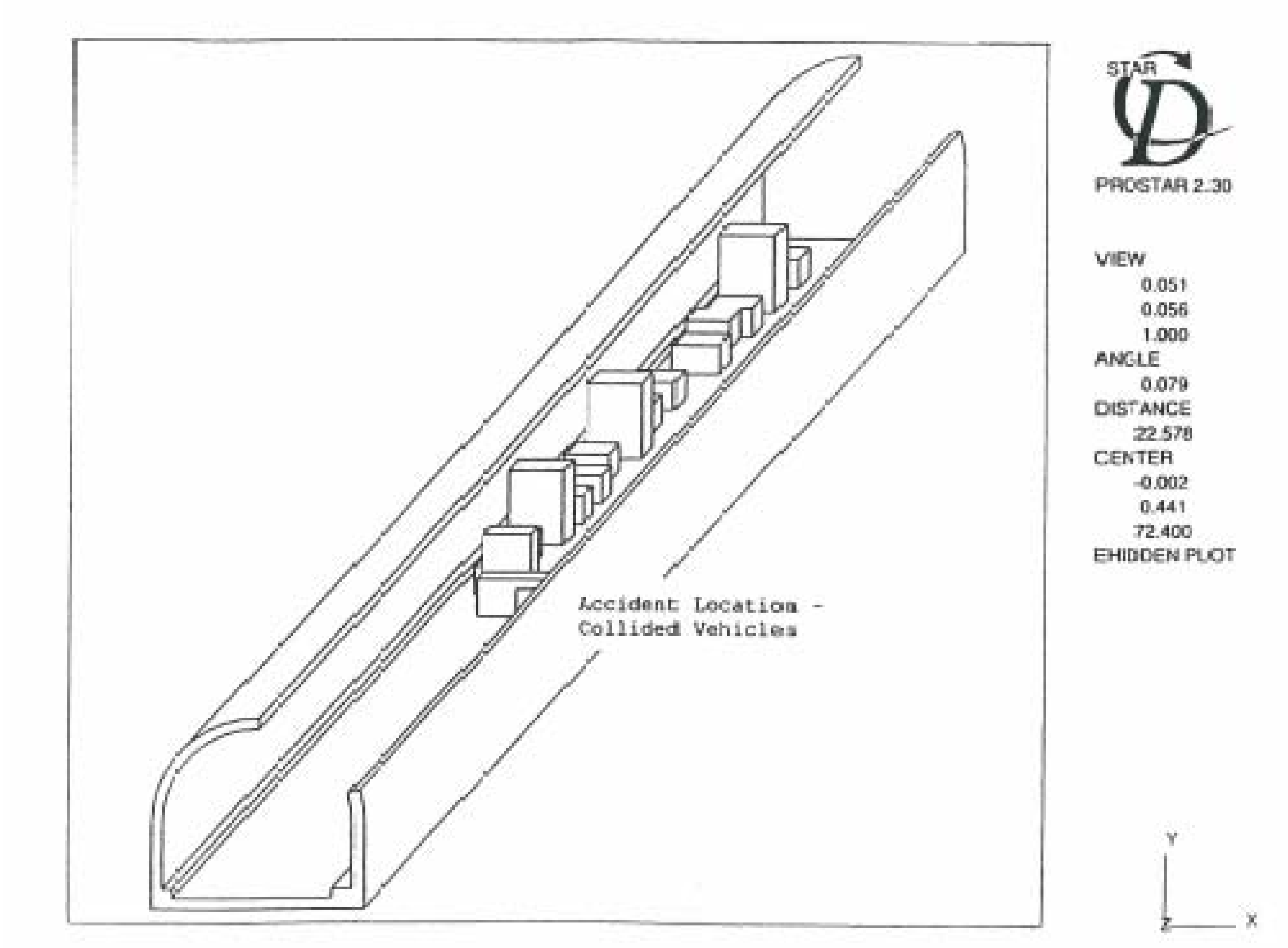


Fig. 2: A three dimensional view across the incident site of the CFD model

6.2 Simulation Results

Fig. 3 to Fig. 5 show the smoke concentration, temperature and velocity profiles within the tunnel in the first 3 minutes after the fire started.

In the first minute, large amount of smoke was produced and flowed in both directions toward the north and south portals. After the tunnel ventilation fans have brought into operation, the bulk tunnel airflow, and hence the smoke began to flow in the same direction as that of the ventilation fans. Smoke was first cleared near the north portal end when fresh air was drawn-in by the negative pressure created by the longitudinal air movement. Gradually the areas upstream of the fire site became clear of smoke within approximately two minutes after the operation of ventilation fans. In addition, temperature at location upstream of fire was brought down to ambient condition (20°C) within the same period of time. Steady state was then reached with all smoke was controlled to flow downstream of the fire site toward the south portal.

During the simulations, the tenability criteria for passengers were always first exceeded by the smoke visibility before the high temperature convective heat from fire have significant impact on the passengers.

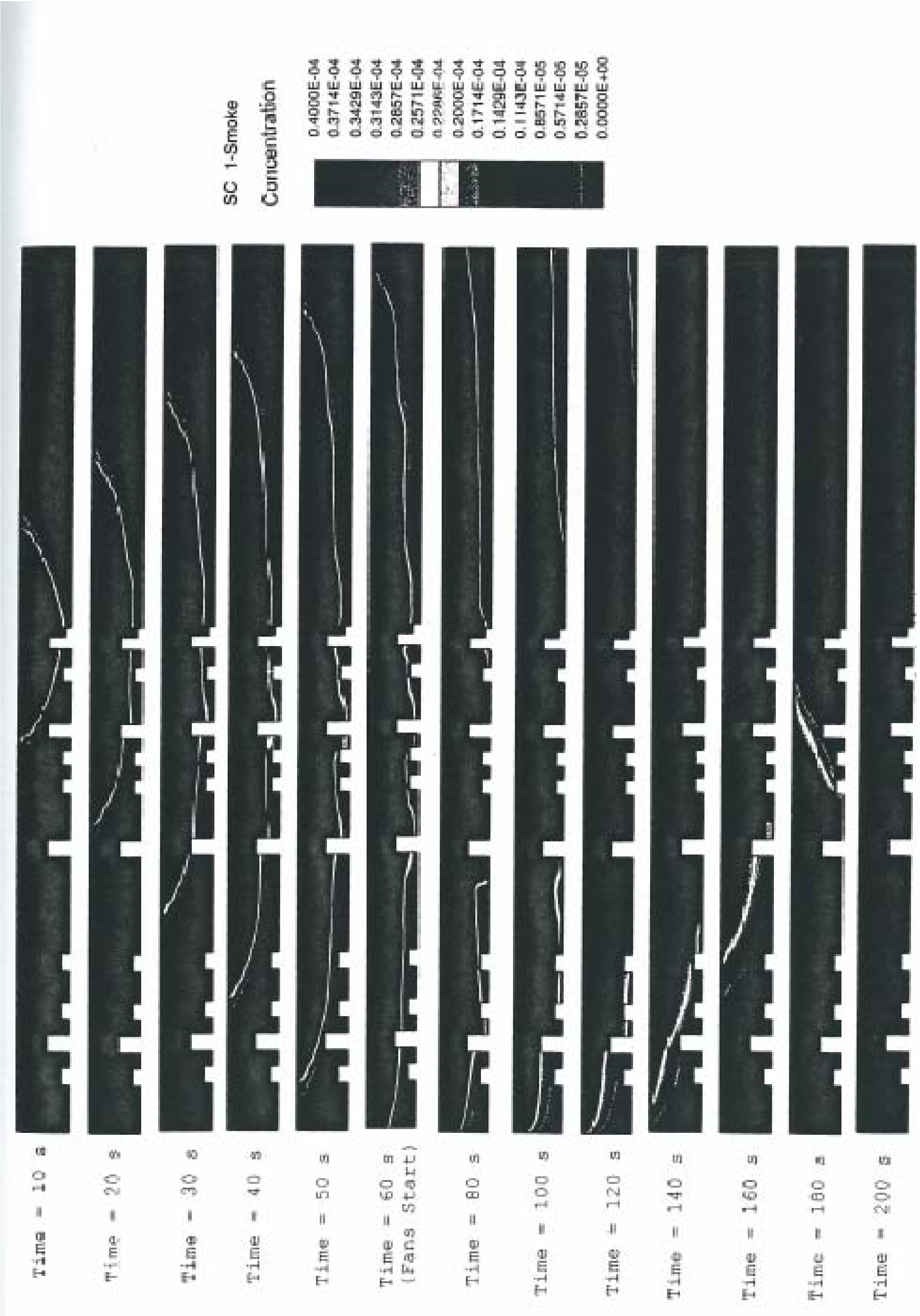


Fig. 3: Transient smoke concentration profile along the tunnel after the car accident

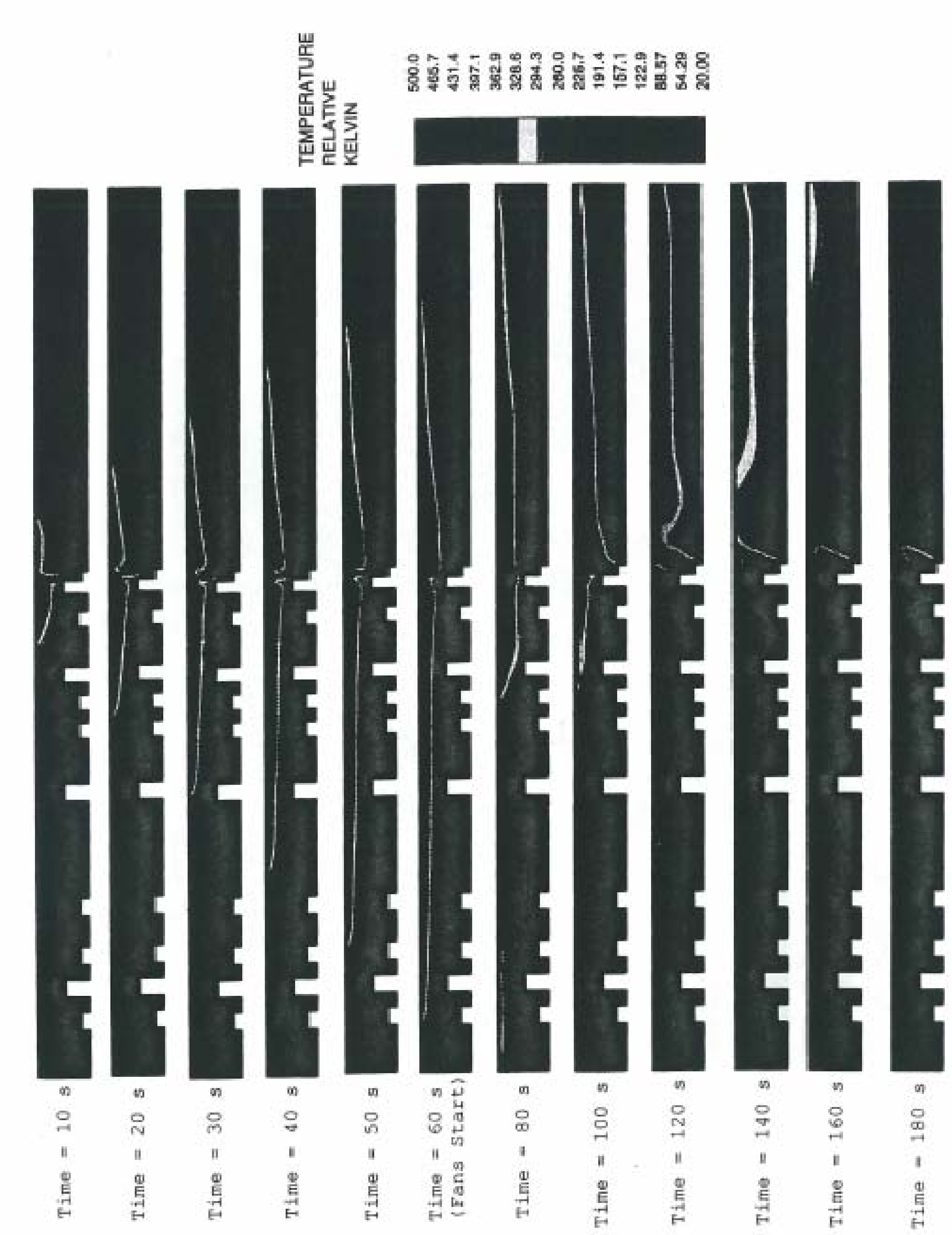
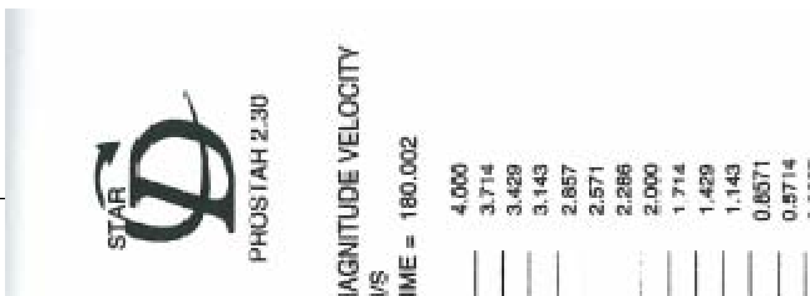


Fig. 4: Transient temperature profile along the tunnel after the car accident



The following summarises the spread rate of smoke in both uphill and downhill directions in the first minute before the tunnel ventilation fans came into operation:

Time after accident (seconds)	10	20	30	40	50	60
Propagation of Smoke (exceeding the visibility criteria) in Uphill Direction						
Length (m)	65	118	174	227	279	-*
Velocity (m/s)	6.5	5.9	5.8	5.7	5.6	-*
Propagation of Smoke (exceeding the visibility criteria) in Downhill Direction						
Length (m)	65	99	141	168	196	219
Velocity (m/s)	6.5	5.0	4.7	4.2	3.9	3.7

* Smoke flow out of the model boundary

Results showed that the propagation of smoke in the first minute was about 300m in the uphill direction and 200m in the downhill direction. This agreed with the experiment [5] that the hot smoke layers would spread up to 580 m in the initial two minutes of a fire (i.e. 290m/min). In addition, the difference in smoke flow velocity in uphill and downhill direction was due to the fact that uphill gradient is favourable to the buoyant smoke spread in that direction.

Below shows the temperature at locations upstream and downstream of fire site:

Time after accident (seconds)	10	20	30	40	50	60	80	100	120	140	160	180
Air temperature at 5.5m Height at 20m upstream of fire site (°C)	200	380	380	380	380	390	300	200	50	20	20	20
Air temperature at 2.5m Height at 20m upstream of fire site (°C)	20	20	25	28	28	24	120	90	40	20	20	20
Air temperature at 5.5m Height at 20m downstream of fire site (°C)	120	250	350	400	450	510	460	480	400	300	160	140
Air temperature at 2.5m Height at 20m downstream of fire site (°C)	20	25	30	30	32	36	40	150	160	140	90	80

Results showed that the air temperature at upstream of fire was in general within the short-term tenability limit (120°C) at occupancy level (2.5m height). However, it was noted that there was a short instance after the start of ventilation fans (i.e. Time = 80 seconds) when the fan-induced air flow disturbed the hot smoke layer at high level resulted in the temperature at low level rise to above 140°C (at location 60m upstream of fire). In addition, the tunnel ventilation fans which generated a bulk air velocity of 4.0m/s could bring down the temperature at upstream of fire to ambient within 60 seconds after it came into operation.

6.3 RESULTS FOR DESIGN AND OPERATION CONSIDERATION FOR TUNNEL UNDER FIRE SCENARIO

Simulations show that the hot smoke could spread up to 300m/minute for a 50MW fire before any tunnel ventilation fans came into operation and therefore the start-up time of tunnel ventilation fans should be minimised. Any delay in the fan start up time will endanger the life of the passengers who were forced to stay behind the accident location. The overall fan response time comprises of the fire detection time, operator reaction time and fan actuation time. Therefore in order to minimise the overall ventilation system response time, it will require a faster fan start-up control and sequencing logic, better training to the operator and a quicker response tunnel fire and smoke detection system.

CFD simulation [11] and the experiment [5] also demonstrated that the critical velocity determined by empirical formulas may sometimes underestimate the required air flow rate in order to prevent the smoke backlayering. This is particular important when the fire size is small compared to the width of the tunnel.

Owing to the assumption in the derivation of the empirical formulas that all of the ventilating air acts to cool the fire. This point is of increasing importance nowadays with our new tunnel become wider and larger in order to cope with the needs of heavy traffic and the design fire load is on the other hand reducing due to the improvement in vehicle and material design. Prediction of critical velocity using empirical equations may on the other hand oversize the fans at a large design fire case. These show the need for a CFD type of modelling for design purpose.

In addition, the pressure force required to generate the same airflow rate at fire scenario will be very much higher than (exceeding 60%) that needed for non-fire scenario. Fan sizing based only on friction loss through tunnel and ductworks at normal operation will be inappropriate for emergency situation. The designer should ensure the tunnel ventilation fan can deliver the required air flow rate and thrust under the emergency fire condition.

7. CONCLUSION

Different approaches to tunnel ventilation system design for fire scenario have been discussed. Example was used to compare the results obtained from different approaches.

CFD simulations were carried out for the longitudinal tunnel ventilation using jet fans under different fire scenarios in a road tunnel. The simulation results were compared with the results obtained from traditional method and a series of full-scale fire tests conducted in an abandoned road tunnel in the USA, known as the Memorial Tunnel Fire Ventilation Test Program (MTFVTP).

The transient environmental behaviour of a tunnel under fire scenario for the design and operation of tunnel ventilation system have also been addressed to demonstrate the capability of using CFD technique for tunnel ventilation system design.

NOMENCLATURE

σ_k Turbulent Prandtl/Schmidt number for turbulence kinetic energy
σ_ε Turbulent Prandtl/Schmidt number for turbulence dissipation energy
σ_f Turbulent Prandtl/Schmidt number for mixture fraction
ε turbulence dissipation rate
μ molecular viscosity
μ_t turbulent viscosity
μ_0 the dynamic viscosity at 273.15K and 101.325kPa
ρ density
τ_{ij} stress tensor components
A Net cross-sectional area of tunnel
C_p Specific heat of air
C_s Sutherland constant, equals to 116 for air
D_m molecular diffusivity of component m
E_c convective heat release rate from fire
$F_{m,j}$ diffusional flux component
Fr Froude number for a flow ventilating a fire
$\sqrt{g}$ determinant of metric tensor
g_i the gravitational acceleration component in direction x_i
h_m static enthalpy
h_t thermal enthalpy
H height of the tunnel at the fire site
H_m heat of formation of constituent m
I local relative turbulence intensity
k turbulence energy
K Von Karman's constant
m_m mass fraction of mixture constituent m
m_m' fluctuation of mass fraction of mixture constituent m
p piezometric pressure
s_h energy source
s_m mass source
s_i momentum source components
t time
T absolute temperature
T_f Average temperature of the fire site gases
x_i Cartesian coordinate (i=1,2,3)
u_i absolute fluid velocity component in direction x_i
u_i' turbulence normal stresses
u_j u_j-u_{cj}, relative velocity between fluid and local coordinate frame that moves with velocity u_{cj}
V_c critical velocity of approaching air to prevent backlayering of smoke

REFERENCES

1. CIBSE Guide E, Fire engineering, CIBSE (1997).
2. Technical Memoranda TM 19:1995, Relationships for smoke control calculations, CIBSE (1995).
3. PIARC (Permanent International Association of Road Congresses) world road congress, Working group no. 6 on "Fire and smoke control", Draft contribution to the committee report for the Montreal congress (1994).
4. "Enclosed vehicular facilities", Applications Handbook, Chapter 12, ASHRAE (1995).
5. Parsons Brinckerhoff / Bechtel, Massachusetts Highway Department, Memorial tunnel fire ventilation test programme comprehensive test report CD complete version (1996).
6. C.C. Hwang and R.F. Chiken, Effect of duct fire on the ventilation, Bureau of mines report of investigation #8311 (1978).
7. C.K. Lee et al., "Interaction between duct fires and ventilation flow: An experimental study", Combustion

Science and Technology, Vol. 20 (1979).

8. C.D. Litton et al., Calculating Fire-throttling of mine ventilation airflow, Bureau of mines report of investigation #9076 (1987).

9. Y. Oka, G. T. Atkinson, “Control of smoke flow in tunnel fires”, Fire Safety Journal, Vol. 25, pp. 305-322 (1995).

10. W.U.A. Leong et al., “Computer analysis of emergency tunnel ventilation on the Jubilee Line Extension”, Fire Engineers Journal, Vol. 1, pp. 32-37 (1997).

11. S.C. Kot, R.M.H. Yau and D.C.W. Tang, “Application of computational fluid dynamics in tunnel ventilation system design and operation under fire scenario”, 2nd International Conference - Tunnel Management, Operation and Maintenance, Hong Kong (1998).

APPENDIX: DETAILS OF FORMULATION FOR THE CFD MODEL

Flow Field

$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho)+\frac{\partial}{\partial x_j}(\rho u_j)=s_m \qquad (5)$$

$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_j u_i-\tau_{ij})=-\frac{\partial p}{\partial x_i}+s_i \qquad (6)$$

Ideal Gases Law

$$\rho=\frac{p}{RT(\Sigma\frac{m_m}{M_m})} \qquad (7)$$

Viscosity

$$\mu=(\frac{T}{273.15})^{3/2}\frac{273.15+C_s}{T+C_s}\mu_0 \qquad (8)$$

Heat Transfer

$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho h)+\frac{\partial}{\partial x_j}(\rho u_j h-F_{h,j})= \frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}p)+u_j\frac{\partial p}{\partial x_j}+\tau_{ij}\partial\frac{u_i}{\partial x_j}+s_h \qquad (9)$$

Diffusional Heat and Mass Flux

$$F_{h,j}\equiv k\frac{\partial T}{\partial x_j}-\overline{\rho u_{j'}h'}+\Sigma_m h_m\rho D_m\frac{\partial m_m}{\partial x_j} \qquad (10)$$

Turbulence Energy Equation

$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho k)+\frac{\partial}{\partial x_j}(\rho u_j k-\frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial x_j})= \mu_t(P+P_B)-\rho\varepsilon-\frac{2}{3}(\mu_t\frac{\partial u_i}{\partial x_i}+\rho k)\frac{\partial u_i}{\partial x_i} \qquad (11)$$

Turbulence Dissipation Rate

$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho\varepsilon)+\frac{\partial}{\partial x_j}(\rho u_j\varepsilon-\frac{\mu_{eff}}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j})= C_{\varepsilon1}\frac{\varepsilon}{k}[\mu_t(P+C_{\varepsilon3}P_B)-\frac{2}{3}(\mu_t\frac{\partial u_i}{\partial x_i}+\rho k)\frac{\partial u_i}{\partial x_i}] -C_{\varepsilon2}\rho\frac{\varepsilon^2}{k}-C_{\varepsilon4}\rho\varepsilon\frac{\partial u_i}{\partial x_i} \qquad (12)$$

The empirical modelling constants for the turbulence model are listed below:

C_μ	σ_k	σ_ε	$C_{\varepsilon1}$
0.09	1.0	1.22	1.44
$C_{\varepsilon2}$	$C_{\varepsilon3}$	$C_{\varepsilon4}$	K
1.92	0.0 or 1.44*	-0.33	0.42

* $C_{\varepsilon3}$ =1.44 for $P_B>0$ and is zero otherwise

Smoke Dispersion

$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho m_m)+\frac{\partial}{\partial x_j}(\rho u_j m_m-F_{m,j})=s_m \qquad (13)$$