

RECENT ACHIEVEMENTS IN MODELLING THE TRANSPORT OF SMOKE AND TOXIC GASES IN TUNNEL FIRES

Suresh Kumar
Building Research Establishment, United Kingdom

ABSTRACT

Fire models are increasingly being used for various aspects of tunnel fire safety design that includes ventilation, smoke management, fire suppression, structural and egress analyses. This paper reports on some recent achievements in the modelling of smoke and toxic gases resulting from fires in tunnels. It discusses the role of simple engineering calculation methods, zone and network models, and advanced computational fluid dynamics (CFD) models in the design and analysis of the ventilation system and the associated operational strategy for fire safety in tunnels.

1. INTRODUCTION

Public awareness of the consequences of fire in road tunnels was raised by the recent series of major incidents involving significant loss to human life. In particular, the fires in the alpine road tunnels at Mont Blanc and Tauern in 1999, Gotthard in 2001, and a subway fire in Daegu in 2003 resulted in total of nearly 200 deaths. The ventilation system is an integral part of the safety in tunnels of significant length, and has the dual role of supplying fresh air to limit the concentrations of vehicle exhaust gases to acceptable levels within the tunnel traffic space (normal operation) and to control movement of smoke and toxic gases in the event of a fire (emergency situation). The design of the ventilation system and associated operational strategy requires careful consideration of a number of important factors. These include the influences of length, slope and cross-section of the tunnel, the distribution, power and response time of any powered fans and the considerations of what might be the appropriate 'design fire'. There may be other important factors to consider also, e.g. the presence of any natural airflow in a sloping tunnel due to a temperature or pressure differential between the tunnel portals. The contribution of each of the above parameters is very complex, and recourse to computer modelling may be required in order to achieve a sound design.

This paper discusses the role of different mathematical modelling approaches ranging from simple empirical formulae, through network and zone models to the more sophisticated computational fluid dynamics (CFD) models for fire safety assessment in tunnels. Four examples are chosen here to demonstrate the capability of the CFD models. They simulate the transient characteristics of full-scale fire tests conducted in tunnels with different ventilation systems. These experimental fires were conducted for the purposes of evaluating the use of jet fan ventilation in controlling 'backlayering' in the Memorial Tunnel, reconstruction of the Mont Blanc Tunnel fire incident, optimisation of fire detection and ventilation design in the Channel Tunnel shuttle wagon, and to examine the beneficial effect (cooling) of sprinklers as well as any adverse interaction (down-drag) that they may cause on fire gases.

2. MODELLING APPROACHES

2.1 Empirical Calculation Methods

Algebraic equations have been developed to estimate the longitudinal airflow velocity, at the fire source, required to prevent the upstream ‘backlayering’ of heat and combustion products (see, for example, Heselden¹ and Danziger & Kennedy²). These empirically based models provide only limited insight, and do not predict conditions downstream of the fire or address issues such as the interaction of the jet fans, fire plume and external wind conditions. The tunnel gradient has a significant influence on the backlayering of combustion products and should be taken into account in determining the critical velocity in a sloping tunnel. Although not addressed in this study, vehicles and other tunnel obstructions will also reduce further the mean volume flow rate of airflow (although the local velocity over an obstruction is likely to increase).

2.2 Zone and Network Models

Network models are an extension to the algebraic equations discussed above. A tunnel, or network of tunnels and passageways, is divided into a circuit of one-dimensional ventilation pathways. Solving for the conservation of mass and energy at each junction, a network model predicts the time dependent movement of air (and smoke) through the network. The strength of these models is their ability to predict air movement within the entire network within relatively short computing time-scales. Their weakness, as for the algebraic engineering equations, is the missing detail at the fire source and the absence of any three-dimensional effects. However, used in combination with a zone or CFD model in the vicinity of the fire source, they provide potentially a very powerful design tool. TRANSIT³ and RABBIT⁴ are recent examples of one-dimensional network models.

In a zone model the tunnel is divided into a number of zones, in which conditions in each zone are assumed to be uniform. Whilst widely used in the study of building fires, zone modelling has enjoyed less application in tunnel fires. FASIT⁵ is an example of a time-dependent zone model, where the fire source is represented as a Gaussian thermal plume. Zone models provide some of the information missing in the algebraic engineering equations and the network models. However, the assumptions required in the plume shape, ventilation flow pattern etc, means that care must be taken that they are not used out of context.

2.3 CFD Models

Based on first principles, CFD fire models solve the underlying conservation equations for mass, momentum, energy and species concentrations (e.g. fuel mixture fraction). This allows the important physical and chemical processes and their interactions, describing the production and movement of smoke and heat, to be simulated realistically. Physical sub-models describe the complex physical processes of turbulence, combustion and thermal radiation. In addition to general purpose commercial codes such as CFX, STAR-CD, FLUENT and PHOENICS, there are a number of special purpose CFD codes for application to tunnel fire such as TUNFIRE⁶, COMPACT-3D⁷ etc. The fire specific codes such as TUNFIRE may employ simpler meshes and numerical schemes, but they have considerable fire science and engineering knowledge built in.

3. FIRE MODELLING IN TUNNELS – SOME RECENT EXAMPLES

3.1 Modelling large fires in the Memorial Tunnel with longitudinal ventilation

The development and validation of computer models for fire hazard analyses in tunnels has been undertaken by simulating physical experiments at reduced-scale, and to a lesser extent at full-scale. Of particular note is the series of fire tests in the disused Memorial Tunnel in the USA⁸, which have provided full-scale data for model validation purposes in a variety of ventilation conditions. Figure 1 shows the plan and end views of the tunnel. As an illustration, figures 2 and 3 show some results of the CFD model TUNFIRE⁶ for the longitudinal ventilation test number 615B that used 6 jet fans for controlling combustion products due to a pool fire growing to approximately 100 MW. The full 853 m length of the tunnel was modelled using approximately 200,000 grid cells with combustion and radiation sub-models. The transient predictions of gas temperatures in figure 2 show smoke ‘backlayering’ for an initial naturally ventilated period of two minutes, which was controlled by activation of a set of 6 jet fans that forced the smoke and heat out of the portal in the down-gradient direction. Figure 3 illustrates the capability of the CFD model in reproducing the evolution of thermal flow fields generated by a 100MW fire.

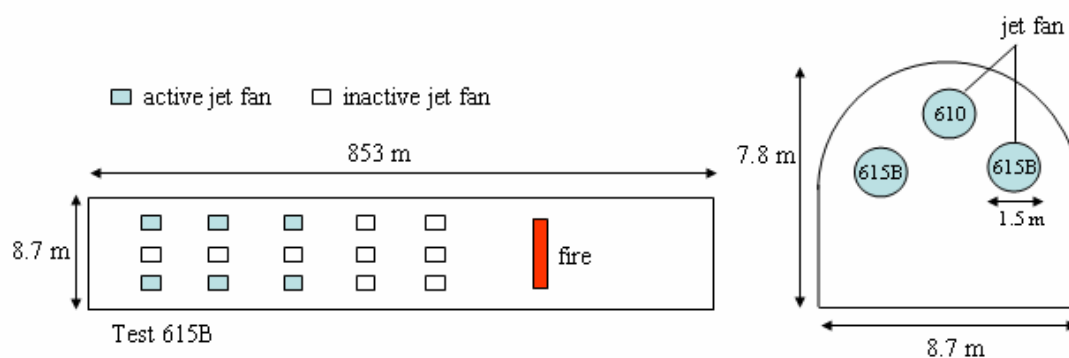


Figure 1 Plan and end view of the Memorial tunnel showing a pool fire and a set of jet fans

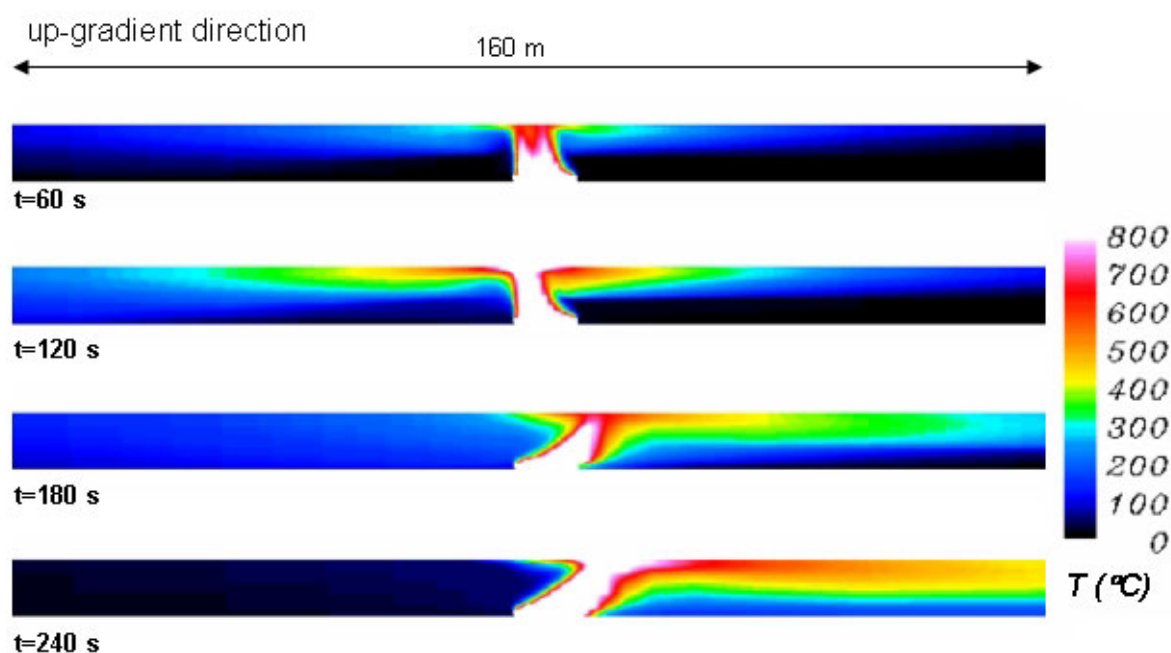


Figure 2 Evolution of predicted gas temperature contours for Memorial Tunnel Test 615B

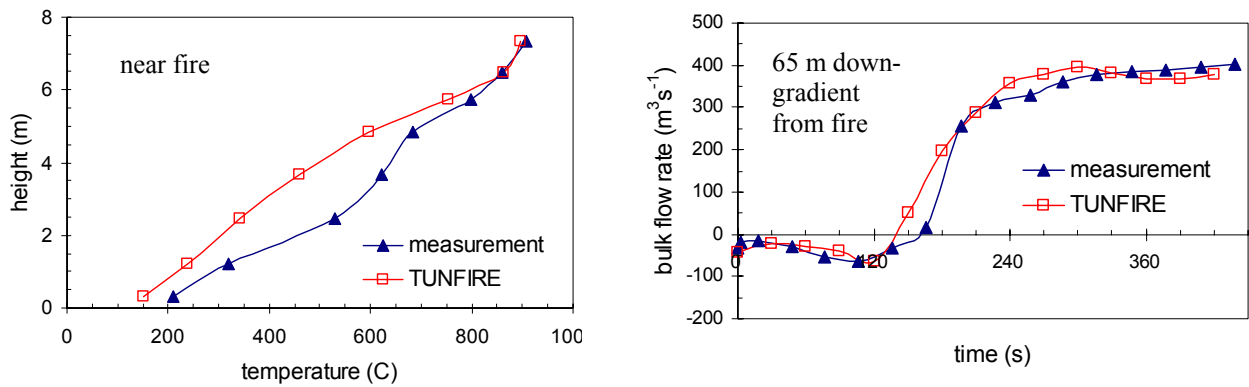


Figure 3 Predicted and measured gas temperatures and bulk air flow rates for Test 615B

3.2 Modelling large fires in the Mont Blanc Tunnel with semi-transverse ventilation

Recently, TUNFIRE was used to reconstruct the major fire incident in the Mont Blanc Tunnel. The CFD modelling was very challenging here because the full 11.6 km length of the tunnel with variable gradient had to be modelled to correctly account for pressure boundary conditions at the tunnel portals. The model was also used to reproduce a series of experimental fire tests⁹ that were performed to gain a better understanding of the existing tunnel ventilation performance prior to restoration work. The tests involved pool fires up to 8 MW under various ventilation arrangements. The ventilation system was semi-transverse, with air supplied at floor level and exhausted through the portals, and augmented by additional ceiling level supply or extract. Tests 1 and 2 approximately replicated, respectively, the ventilation settings that would have been expected to produce the most favourable conditions and those that were actually employed on the day of the disaster. A 3-km long section of the Mont Blanc tunnel with approximately 200,000 grid cells was used to reproduce the pool fire tests within the tunnel. Figure 4 on the left shows the damage to the tunnel after the incident and on the right the CFD results for Test 1, illustrating six pool fires and the effect of a net airflow from the Italian to French portals, and the presence of smoke ‘backlayering’ on the Italian side of the fire source. Predicted temperature and velocity profiles were compared to those measured in both tests. Figure 5 compares, as an example, predicted and measured temperature profiles for Test 2 at 20m and 180m from fire towards French portal, where it can be seen that the model has reproduced stratification of the smoke layer up to 180m from fire.

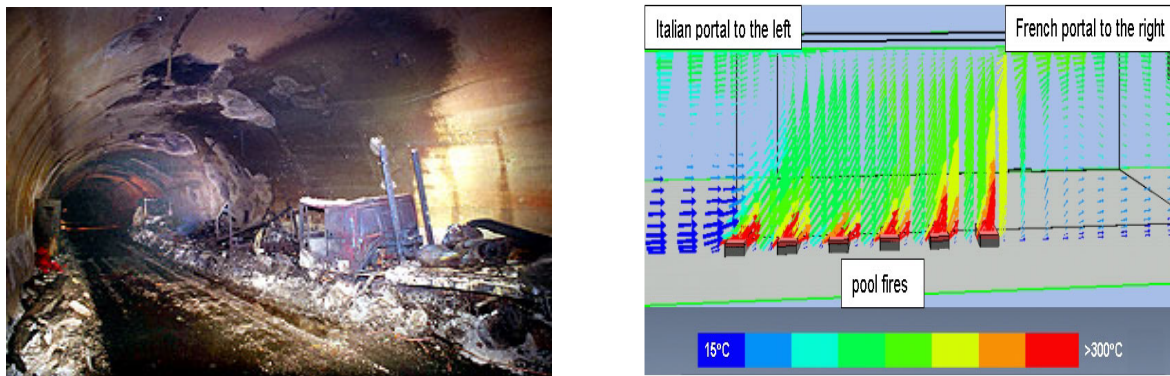


Figure 4 Mont Blanc tunnel fire and the predicted thermal flow fields near fire for Test 1

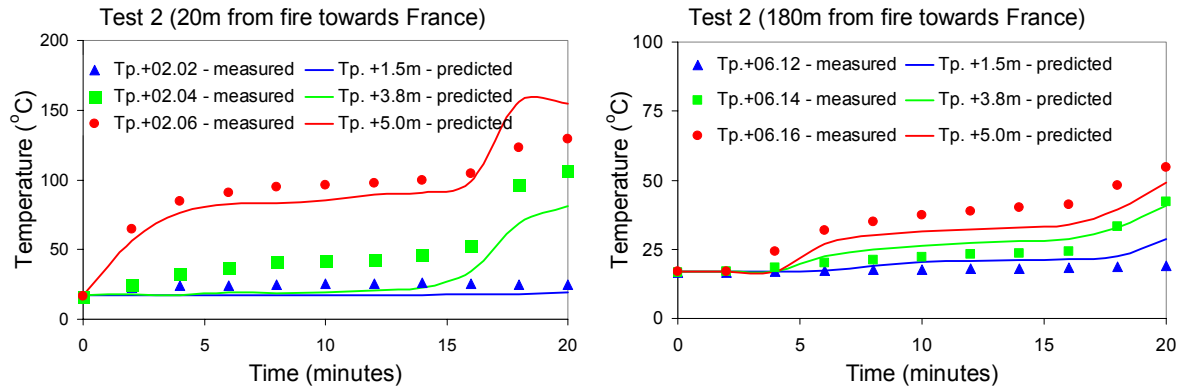


Figure 5 Comparison of predicted and measured temperature profiles in Mont Blanc Tunnel

3.3 Using CFD for optimising smoke ventilation system in a Channel Tunnel shuttle wagon

The tourist shuttle trains to be used in the 50km long Channel Tunnel had been designed on the basis of car passengers travelling with their vehicles within the shuttle wagons. Figure 6 illustrates a schematic of a shuttle wagon with five cars denoted by dotted rectangles, and ceiling level ionisation and optical detectors denoted centrally by solid circles and rectangles respectively. Each wagon was equipped with a fire detection and suppression system and a ventilation system. The fire detection comprised a range of detector types and three levels of fire detection. The detection for levels 1 and 2 corresponded respectively to the operation of first and second detectors with an opacity level of 7% (i.e. OD/m of 0.0315 m^{-1}) and level 3 on an optical detector operating at an opacity level of 70% (OD/m of 0.5 m^{-1}) when fire is due to be extinguished by a suppression system. The ventilation system was designed to provide a comfortable atmosphere under normal operating conditions, and was shut down on fire detection.

CFD modelling was first validated¹⁰ against full-scale fire tests and was then used to optimise the performance of the ventilation system in the event of a fire such that earlier fire detection is achieved so that the time available for evacuation (between level 2 and level 3) is maximised. Figure 7 shows the generation flow chart of selected scenarios used for this purpose. The impact on time to detection of earlier ventilation shutdown at level 1 was examined by comparing scenario G with F for the rear fire and scenario H with I for the central fire, where the ventilation was shut down at level 2 for scenarios F and H and at level 1 for scenarios G and I. As an illustration, figure 8 shows the comparison of the transient predictions of OD/m at nose and ceiling levels for scenario G (with ventilation shutdown at level 1) with scenario F (with shutdown at level 2). It can be seen that level 2 (indicating fire detection) is reached at 65s for scenario F and 50s for scenario G, i.e., some 15s earlier for scenario G than for scenario F. This illustrates that earlier shutdown of the ventilation system has beneficial effect on fire detection and hence on the evacuation time.

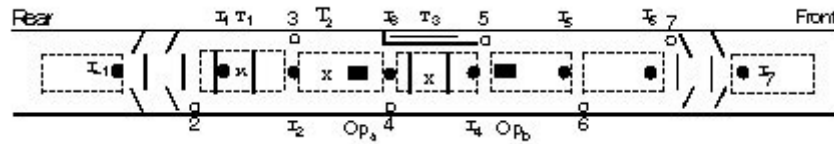


Figure 6 Schematic of Channel Tunnel shuttle wagon

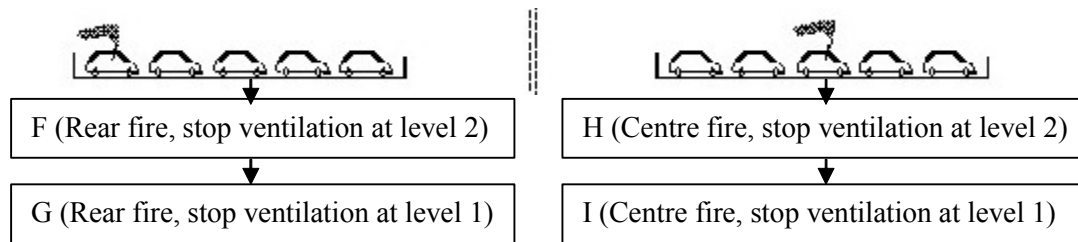


Figure 7 Scenario generation flow chart

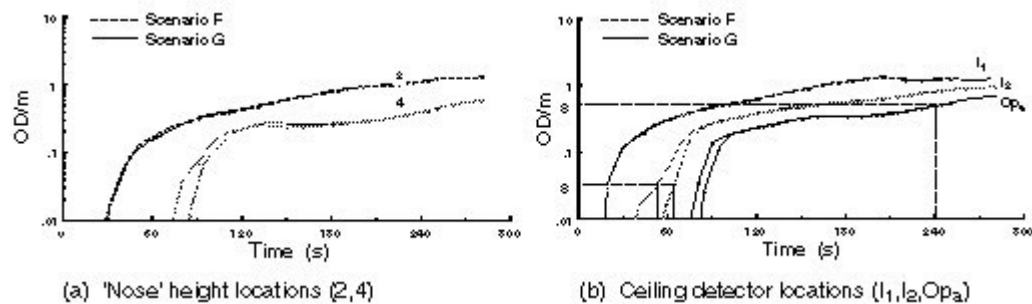


Figure 8 Predicted optical densities at 'nose' and ceiling levels for scenarios F and G

3.4 Modelling sprinkler-fire gas interaction in tunnels

There is currently a growing interest in the use of water suppression systems in tunnels because it is acknowledged that they can help protect the tunnel structure and, if operated soon enough, may limit fire spread; but their benefits for life safety aspect are yet to be demonstrated. Sprinklers may cause 'smoke downdrag', where the downwards momentum transfer from the droplets and the cooling of the gases may combine to result in the smoke layer descending, possibly onto evacuation paths. These issues are currently being addressed by physical experiments at reduced-scale and extrapolated to additional scenarios and to full-scale using a coupled sprinkler-CFD (JASMINE-SPARTA) model. Figure 9 illustrates a typical 'deluge head' spray pattern and the effect that it can have in causing 'downdrag' of a smoke layer.

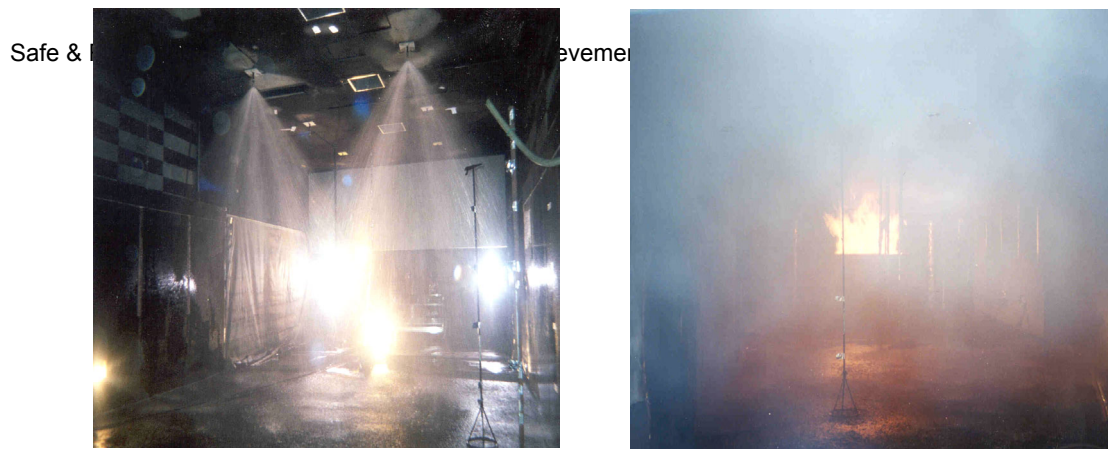


Figure 9 Typical 'deluge head' sprinklers sprays and their effect on 'smoke downdrag'

Numerical simulations of the reduced-scale experiments have been undertaken to verify in figure 10 that the JASMINE-SPARTA model can reproduce the measured level of cooling in the smoke layer¹¹. Figure 11 shows a typical graphical output from the model, including, gas temperature contours, a predicted spray pattern and a 'map' of delivered water density on the floor surface. The gas temperature contours in figure 11 (a) illustrate that 'open head' and 'automatic head' sprinklers have similar 'downdrag' effect on fire gases. The sprinkler-CFD model is now being extended to water mists, and to incorporate the interactions of the water spray with the radiation field. This latter issue is important for both water mists, where it is one of the main hazard mitigation mechanisms, and also for water curtains, which have been suggested for application to tunnels.

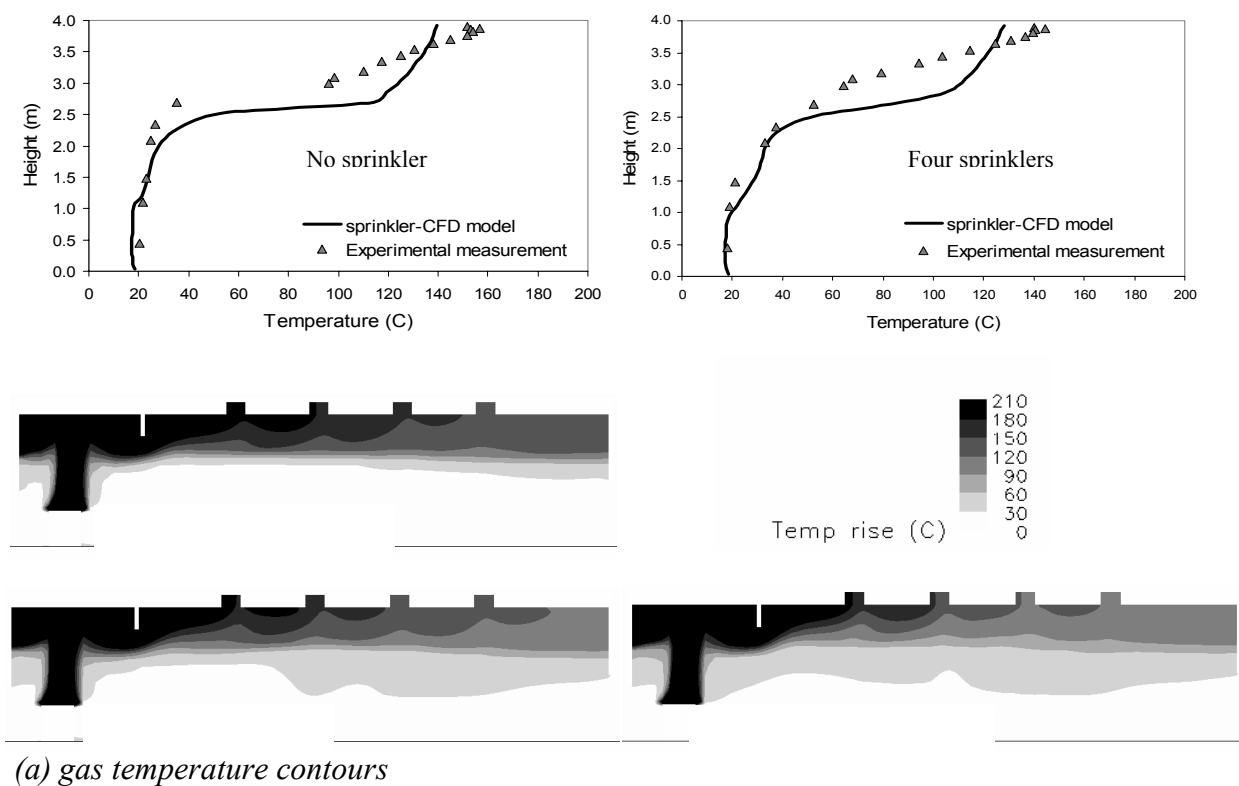


Figure 10 Effect of sprinklers on cooling of fire gases by JASMINE-SPARTA model

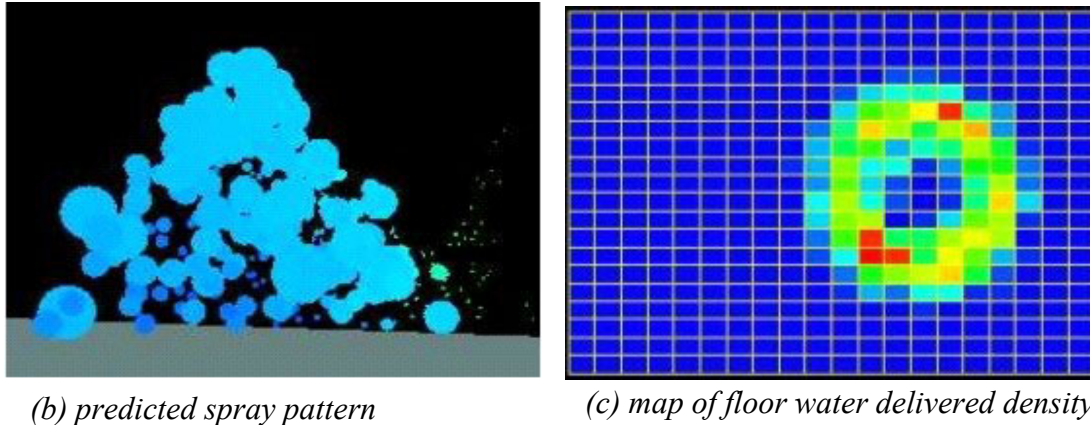


Figure 11 Graphical output from the JASMINE-SPARTA model

4. CONCLUDING REMARKS

It has been demonstrated that the fire modelling, particularly CFD, is capable of reproducing the evolution of the thermal and chemical characteristics of large fires in tunnels with different ventilation systems. The CFD approach has an important role to play in the design and analysis of the tunnel smoke ventilation system and its interactions with water suppression system (e.g., cooling and down-drag). It complements other predictive methods, including 'simple' engineering calculations, zone and network models, and lends itself to detailed parametric studies that would be too expensive to undertake by full-scale experiment. It is particularly attractive to study smoke 'backlayering' and the associated issue of critical velocity. When addressing the ventilation issues surrounding a long tunnel, a combination of CFD and airflow network model may provide an appropriate compromise between resource and accuracy.

5. REFERENCES

1. A.J.M. Heselden: Studies of fire and smoke behaviour relevant to tunnels. In *Proc. 2nd Int. Symp. on the Aerodynamics and Ventilation of Vehicle Tunnels*, Cambridge, UK, 23-25 March 1976, BHRA Fluid Engineering, paper J1.
2. N.H. Danziger, W.D. Kennedy: Longitudinal ventilation analysis for the Glenwood Canyon tunnels. In *Proc. 4th Int. Symp. on the Aerodynamics and Ventilation of Vehicle Tunnels*, 1982, BHRA Fluid Engineering, pp. 169-186.
3. P. Chasse, J.M. Apvrille, J-M: A new 1D computer model for fires in complex underground networks. In *Proc. 1st Int. Conf. on Tunnel Fires and Escape from Tunnels*, Lyon, France, 5-7 May 1999, Independent Technical Conferences Ltd, pp. 201-211.
4. I. Riess, M. Bettelini: The prediction of smoke propagation due to road tunnel fires. In *Proc. 1st Int. Conf. on Tunnel Fires and Escape from Tunnels*, Lyon, France, 5-7 May 1999, pp. 213-222. Independent Technical Conferences Ltd.
5. D.A. Charters, W.A. Gray, A.C. McIntosh: A computer model to assess fire hazards in tunnels (FASIT). *Fire Technol.*, vol. 30, 1994, pp. 134-154.

6. S. Miles, S. Kumar, R. Andrews: Validation of a CFD model for fires in the Memorial Tunnel, In *Proc. 1st Int. Conf. on Tunnel Fires and Seminar on Escape from Tunnels*, 5-7 May 1999, Lyon, France. Independent Technical Conferences Ltd., pp. 159-168.
7. K.C. Karki, S.V. Patankar: CFD model for jet fan ventilation systems. In: 10th Intl. Symposium on Aerodynamics and Ventilation of Vehicle Tunnels-Principles, analysis & Design. BHR Group Conf. Series, Pub. No. 43, 2000, pp. 335-80.
8. Massachusetts Highway Department & Bechtel/Parsons Brinckerhoff: Memorial Tunnel Fire Ventilation Test Program – Test Report, 1995.
9. B. Brousse, A. Voeltzel, Y. Le Botlan, E. Ruffin: Ventilation and fire tests in the Mont Blanc Tunnel to better understand the catastrophic fire of 24 March 1999. In *Proc. 3rd Int. Conf. on Tunnel Fires and Escape from Tunnels*, 9-11 October 2001, Washington, USA. Independent Technical Conferences Ltd., pp. 211-222.
10. S. Kumar, M. Yehia: Application of JASMINE to the modelling of vehicle fires in a Channel Tunnel shuttle wagon, In *Proc. 2nd International Conference on the Safety in Road and Rail Tunnels*, Granada, Spain 1995, p. 321
11. S. Miles, S. Kumar, K. Chong: Development of a CFD methodology for predicting the combined effect of sprinklers and smoke ventilation in tunnels. In *Proc. 4th Int. Conf. on Tunnel Fires*, 2-4 December 2002, Basel, Switzerland, pp. 329-338.