Proceedings of the 12th International Fire Science & Engineering Conference, Interflam 2010, 5-7th July 2010, University of Nottingham, UK, Volume 2, pp. 1091-1102. ISBN 978 0 9541216-6-2, 2010.

MODELLING FACTORS THAT INFLUENCE CFD FIRE SIMULATIONS OF LARGE TUNNEL FIRES

Fuchen Jia, Zhaozhi Wang and Edwin R. Galea
Fire Safety Engineering Group, University of Greenwich, London SE10 9LS

ABSTRACT

A series of CFD fire predictions of a 100 MW Memorial tunnel fire test are performed to investigate several factors that influence the outcome of CFD fire simulations of large scale tunnel fire scenarios. This study shows that at the critical ventilation velocity, the occurrence of back-layering is very sensitive to factors such as variation in ventilation velocity, tunnel surface condition, the width of the modelled fire source and the presence of relatively minor obstacles in the vicinity of the fire source. A serious question has also been raised over the suitability of the use of inlet boundary conditions to represent the ventilation flow generated by jet fans. To a large extent, the success of simulating a tunnel fire test such as the Memorial Tunnel test relies on correct replication of the test conditions and setup. However, for most large scale tunnel tests, not all the information required to correctly specify a CFD fire simulation is available and hence sensitivity analyses over critical model setup parameters is essential to correctly interpret differences between predictions and experimental measurements.

INTRODUCTION

The critical velocity of a longitudinal ventilation system for a tunnel is defined as the minimum ventilation velocity that is capable of preventing upstream travel (back-layering) of combustion products of a fire in the tunnel. The prevention of back-layering is essential for both fire fighting and the safe evacuation of people in the event of a tunnel fire. The relation between back-layering and heat release rates within tunnel fires was first studied in the late 1960s[1] and has been the subject of extensive experimental and numerical studies[2-7] over the past two decades. With the advance of CFD fire modelling and improvements in computing power, CFD investigation of tunnel fires has gained considerable popularity in recent years due to its obvious advantages over experimental fire tests, namely, relatively low costs and its ability to provide considerable information concerning both fire behaviour and the flow field. A number of CFD fire modelling studies[5-8] of tunnel fires have demonstrated that it is a useful technique to corroborate the findings from tunnel fire experiments and empirical models and provide explanations for the relation between the critical velocity of longitudinal ventilation system and the heat release rates (HRR) of tunnel fires. In most of these studies, the HRRs of the tunnel fires are no more than 50 MW. However, some studies[2,4] have shown that HRRs of large tunnel fires can be over 100 MW and CFD simulation of tunnel fires in excess of 100 MW is still considered a challenge today.

Esther Kim et al.[9] have studied one of the Memorial tunnel tests with a 100 MW HRR using the CFD fire simulation software FDS. In their study, a series of CFD simulations of the test were performed with measured HRRs and transient ventilation velocities. Their study suggested that even with ventilation velocities 20% in excess of measured values, back-layering occurred much earlier than observed in the experiment and unlike in the experiment, was maintained throughout the simulation. In the experiment back-layering occurred only when the longitudinal ventilation was relatively small and it was stopped when the ventilation speed was increased. This inability to produce good agreement with this set of experimental results is not necessarily a weakness of CFD but may be a result of uncertainties involved in setting up the CFD simulation.

It is suggested in this paper that there are a number of uncertainties in the Memorial Tunnel fire test data which will be reflected in the setup of any CFD fire model to simulate the Memorial Tunnel fire test that will eventually affect the outcome of the simulation. Among these factors the following will be investigated in this study: the ventilation velocities, the width of the fire source, obstacles in the vicinity of the fire source, and the roughness of the tunnel surface. The measured velocities which have been used as boundary conditions to specify the ventilation flow were subjected to an error of approximately 0.1 m/s, which causes 3%-4% changes from the measured velocities at 107 m upstream from the fire[9]. The width of the fuel source used in CFD simulations of the Memorial tests varied widely in different studies[7,9] from 2.6 m to 4.7 m. Within the Memorial fire tests there were a number of experimental instruments of various sizes and shapes scattering on the floor of the tunnel. These obstacles inevitably affect the flow pattern and flow speed. The roughness of the tunnel surface is also unknown. Woodburn and Britter[5] have revealed in their study that the level of wall roughness can have a significant effect on the results of CFD simulation of tunnel fires.

The objective of this study is to investigate the impact of these four uncertainties on the CFD fire simulation of the Memorial Tunnel fire test. The study focuses on reproducing the experimental observation of back-layering as this is one of the primary concerns of tunnel smoke control systems.

THE MEMORIAL TEST

For the selected Memorial test, test 621A, the tunnel was 8.8 m wide, 7.9 m high and 853 m long with an arched ceiling. The tunnel has a 3.2% upgrade against the ventilation flow. The longitudinal ventilation was provided with a total of 24 reversible axial flow fans. Air temperature, air velocity and gas concentrations were measured at various stations. The fire size varies around 100 MW throughout the duration of the test. The fuel used is No. 2 fuel oil filled in four steel pans with about 0.15 m of water at the bottom. The pans are approximately 0.9 m above the tunnel floor. The fire is placed at 0 m with the tunnel entrance at -615 m upstream and the exit at 238 m downstream. Back-layering occurred during the test between 696-840 seconds as the longitudinal ventilation flow was reduced to less than the critical velocity at approximately 11 minutes but the back-layering was stopped when the ventilation speed increased again after 13 minutes. Back-layering was deemed to be prevented when smoke was contained approximately 11 m upstream from the fire. In this study, the occurrence of back-layering means that back-layering appears beyond 11 m upstream from the fire.

THE CFD SIMULATIONS

In the coordinate system used in this study, the fire is placed at 0 m which is consistent with the one used in the Memorial tunnel tests. To reduce the overhead of the computation, only the section of the tunnel, -107 m upstream from the fire to 115 m downstream, is simulated. At -107 m, the boundary is set as an inlet with the axial velocity profile specified using the measured time varying values at this location. The downstream end is set as an outlet. The tunnel surface is modelled with the wall function and the fire source is modelled as a fuel source having the same area as the fuel pan. The fuel source is 0.9 m above the floor with a solid base representing the steel pans. The burning of the fuel which is assumed to be diesel is modelled with the Eddy Dissipation model with infinitely fast complete chemical reaction. The CFD simulations are carried out with a research version of SMARTFIRE[10,11,12], a CFD fire modelling software. Within SMARTFIRE an unstructured mesh was used to correctly represent the curved nature of the tunnel ceiling. In this research version, an algebraic multi-grid algorithm is employed to solve pressure field of the flow domain. Algebraic multi-grid algorithms are as fast, efficient and accurate as the FFT algorithms. This algorithm is essential to obtaining accurate solutions of CFD fire simulations with very large geometries such as the Memorial test investigated in this study. Also in this version, the wall roughness is modelled with an extended wall function[13]. Within this extended wall function, the law of the rough wall maintains the same form of the standard wall function but with different parameter values corresponding to the specified wall roughness height.

Scenarios Simulated

To investigate the four factors stated before, the following scenarios are simulated

Table 1: Specifications of scenarios studied

	Ventilation velocity	Width of the fire	Surface roughness	Obstacles around the fire
Base Case	Measured velocity profile	4.5 m	smooth	None
Case I	Measured velocity profile + 0.1 m/s	4.5 m	smooth	None
Case II	Measured velocity profile + 0.1 m/s	4.5 m	1 mm	None
Case III	107% Measured velocity profile	4.5 m	smooth	None
Case IV	107% Measured velocity profile	4.5 m	1 mm	None
Case V	Measured velocity profile + 0.1 m/s	4.5 m	smooth	Three columns
Case VI	Measured velocity profile + 0.1 m/s	2.6m	smooth	None

The measured transient velocity profile at -107 m and its variations in the different scenarios are used as inlet velocity in the CFD simulations. The measured velocity profile at the inlet is divided into an upper value and a lower one. The height of the lower part is 2 m from the floor and the upper part is from 2 m above the floor to the full height of the tunnel. The average value of the measured velocities over the height of each part is set as incoming air velocity at the corresponding part. The 4.5 m of the width of the fuel source is estimated from photos of the fuel pans, which is close to the width used in[9]. To investigate the effect of the width of the fuel source on back-layering, in Case VI, a width of 2.6 m is used, which is the same as the width used in[7]. The total surface area of the fuel pans are maintained by extending the length of the fuel source in this case. The instrument trees in the vicinity of the fire source were insulated and as such presented obstructions to airflow. The three trees close to the fire source are represented as obstacles with a width of 0.3 m at the central line of the tunnel in Case V. These obstructions are located at -11 m, 5 m and 12 m respectively.

Location of the Downstream Boundary

To investigate the effect of the downstream length on the outcomes of the simulations, an additional simulation with the downstream length extended to the tunnel exit was conducted. This additional simulation has the same setup as that of the Base Case. The results from the two simulations with the two different downstream lengths show that the effect of the downstream length on velocity and temperature was negligible. Back-layering occurs almost at the same time in the two simulations. Therefore, the shortened downstream length is employed in the numerical simulations conducted in this study. Other numerical studies[14,15] have also shown that when the locations of the downstream boundary are properly chosen, their effect on the results of CFD fire simulations can be neglected.

Grid Sensitivity Analysis

Simulations with three meshes are performed for grid sensitivity analysis. The cell numbers within the three meshes are 522776, 447077, and 329928 respectively. The changes in these meshes are mainly the cell sizes in the cross section of the tunnel, in the vicinity of the fire source and in the immediate downstream region after the fire source. Three simulations with the three meshes are carried out with a constant ventilation velocity and a constant fire output. The constant ventilation velocity used is 3.1 m/s as suggested by Wang[8], the possible maximum critical velocity in spite of both the tunnel hydraulic height and fire heat release rate. The constant fire output is 100 MW, the assumed fire output of the Memorial test 621A.

All the three simulations reach steady state after 12 minutes of simulated time. The steady state results of the three simulations are compared in the grid sensitivity analysis. The maximum temperatures in the fire plume at the central vertical line 12 m after the fire source are 1340 K, 1350 K and 1310 K respectively for the three different meshes. The maximum velocities in the fire plume predicted by the three meshes are 10.77 m/s, 10.79 m/s and 10.50 m/s respectively. In particular, the temperature distributions within the fire plume produced by the first and second mesh are very similar as shown in Figure 1. Therefore, the second mesh with 447077 cells is chosen in this study as it is deemed that by increasing cell numbers there will be no significant improvement in simulation results.

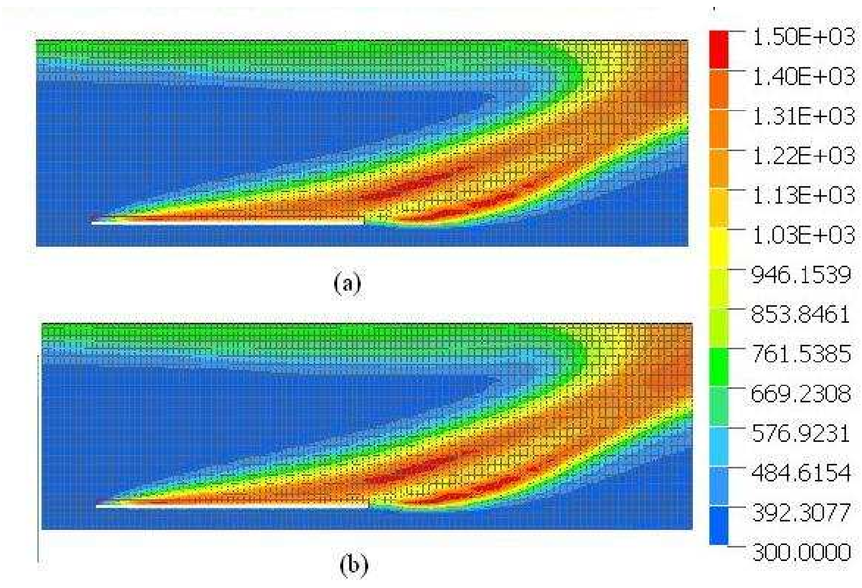
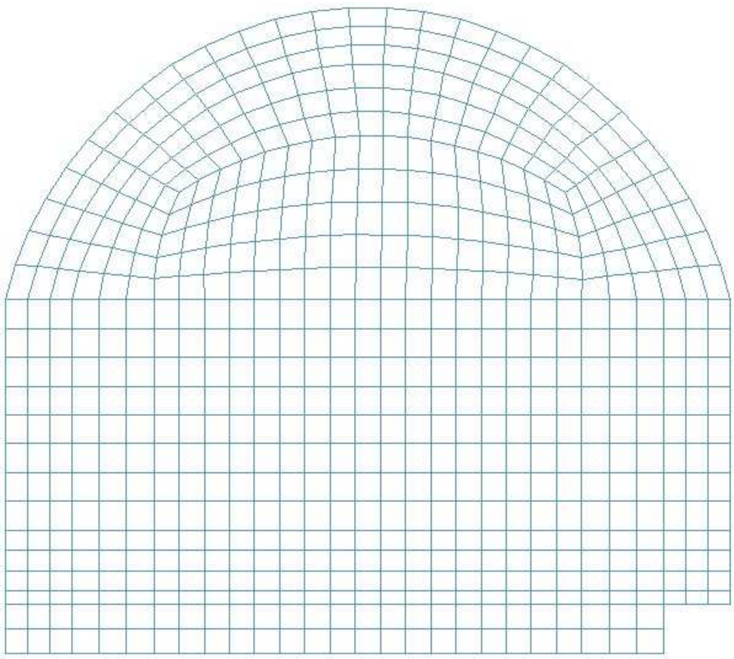
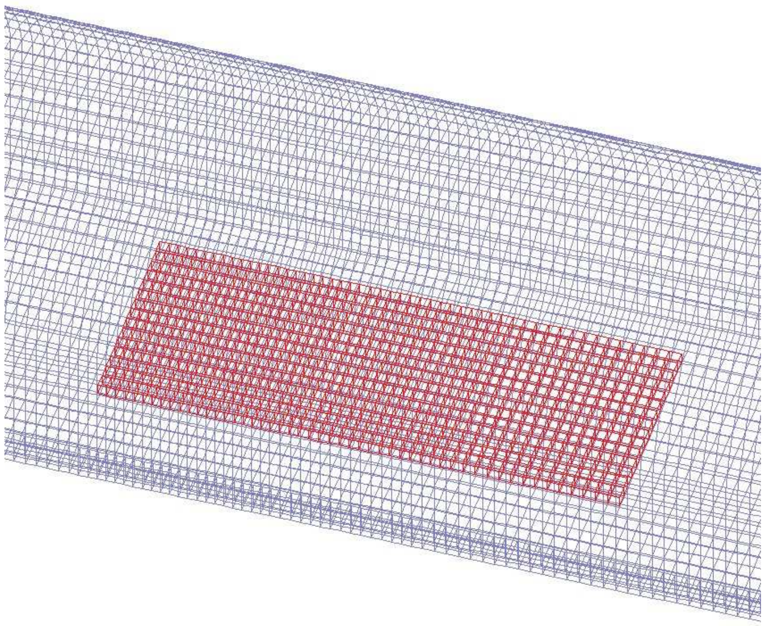


Figure 1: predicted temperature distributions by (a) the first mesh and (b) the second mesh. Unit: K

Depicted in Figure 2 is the computational mesh used in the simulations. The unstructured mesh through the cross-section and in the vicinity of the fire is depicted. Note that the curved shape of the tunnel ceiling is not represented as a stepped blocked mesh but is accurately represented in the SMARTFIRE model.



(a)



(b)

Figure 2: The mesh used in this study (a) in cross-section, (b) in the vicinity of the fire

RESULTS AND DISCUSSIONS

The timings of back-layering occurrence beyond 11 m upstream from the fire in the test and in the simulations are summarised in table 2.

Table 2: Occurrence of back-layering in the cases studied

	test	Base Case	Case I	Case II	Case III	Case IV	Case V	Case VI	Prediction From [9]
Onset(s)	696	600	660	648	676	660	660	632	396
Stop(s)	837	--	888	--	872	888	--	--	--

The Effect of the Ventilation Velocity On Back-Layering

Back-layering is extremely sensitive to the ventilation velocity. The Base Case shows the back-layering appears beyond 11 m upstream from the fire at approximately 600 s and is maintained throughout the simulation. However Case I, which is the same as Base Case with a 0.1 m/s increase in the ventilation velocity (also equivalent to the upper range of the experimental errors in the velocity measurement) shows that the back-layering occurs at approximately 660 s and disappears at approximately 888 s. However, as the ventilation velocity is reduced slightly after 879 s, the back-layering within the predicted results re-appears after 948 seconds – unlike in the experiment. This demonstrates how sensitive the occurrence of back-layering is to the imposed ventilation velocity. A series of simulations was conducted to determine at what level of ventilation velocity no reoccurrence of back-layering occurs. The numerical results show that 8% excess of the measured ventilation velocity is needed to stop the back-layering re-occurring.

In Woodburn and Britter's study[5], it is found that a 7.5% increase in ventilation velocity can result in a reduction of more than 70% in length of back-layering. In this study, during the period of the back-layering being stopped in Case I, smoke is just contained at approximately 11 m upstream from the fire. During the same period of time in the Base Case, smoke travels upstream as far as 25 m. During this period of time, the 0.1 m/s increase in the ventilation velocity of Case I is approximately 5% of deviation from that of Base Case in the lower part of the tunnel and 3% in the upper part of the tunnel respectively. The changes of the ventilation velocities cause a reduction of approximately 56% in length of the back-layering.

The Effect of Tunnel Surface Conditions on Back-layering

Tunnel surface conditions can affect the occurrence of back-layering. The only difference between Case I and II is the tunnel surface roughness. Case II has a surface roughness height of 1 mm, typical for concrete surfaces. In Case II, the back-layering occurs approximately 12 s earlier than it does in Case I and it remains throughout the simulation while in Case I, the back-layering stops at approximately 888 s. This finding is surprising because it is expected that the increase of the core flow velocities due to slowing down of the flow near the tunnel surface and the weaker buoyancy due to increased heat transfer near the tunnel surface caused by the rough surface in Case II will help prevent back-layering. However, with a stronger ventilation as shown in Case III and IV, the effect of surface roughness on the occurrence of back-layering is decreased. In both cases, back-layering is stopped at some stage.

Presented in Figure 3 and 4 are predicted velocities and temperatures within the back-layering of Case I and Case II at 900 s. Figure 3 shows that the velocities within the back-layering of the two cases are more or less the same. On the other hand, as shown in Figure 4, the back-layering of Case II with rough surfaces is cooler than the one of Case I with smooth surfaces, hence the air in the back-layering of Case II is heavier than that of Case I. This means that the back flow of Case II carries more momentum than the back flow of Case I. Therefore, the back flow of Case II is capable of travelling slightly further upstream.

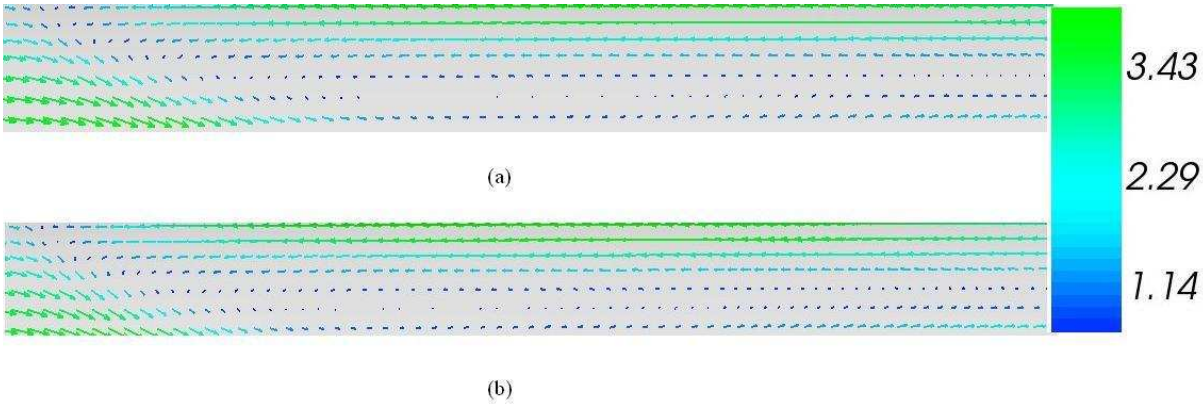


Figure 3: predicted velocities within the back-layering of (a) Case II (rough wall) and (b) Case I (smooth wall) at 900 s. Unit: m/s

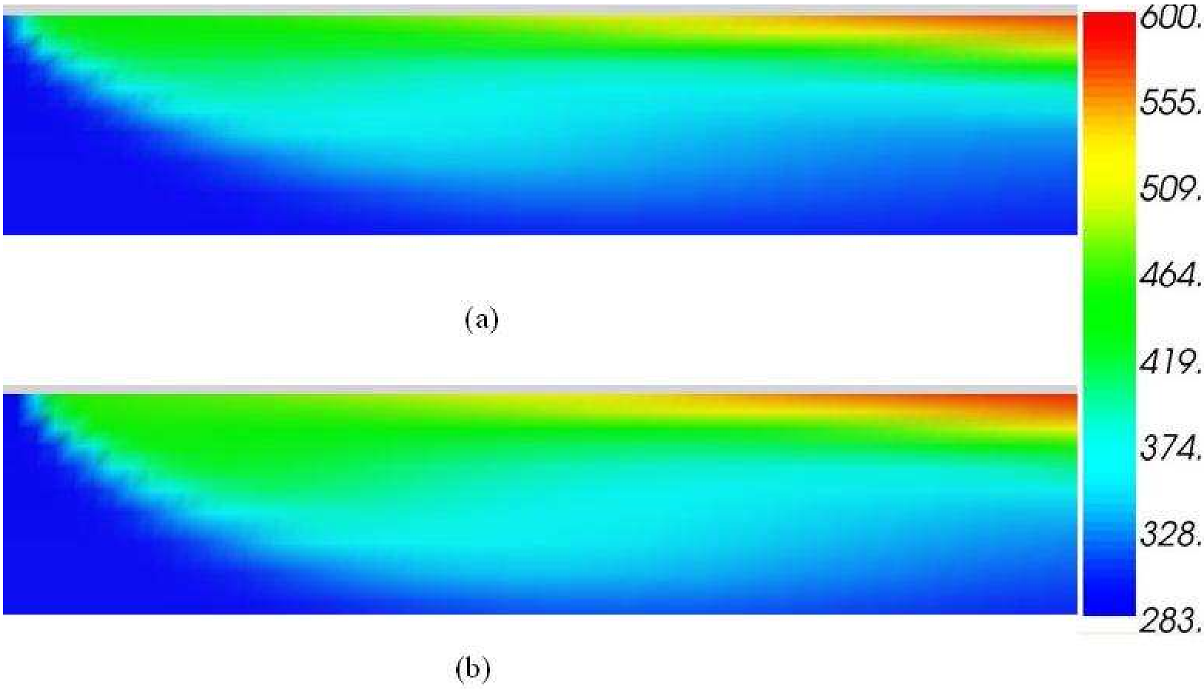


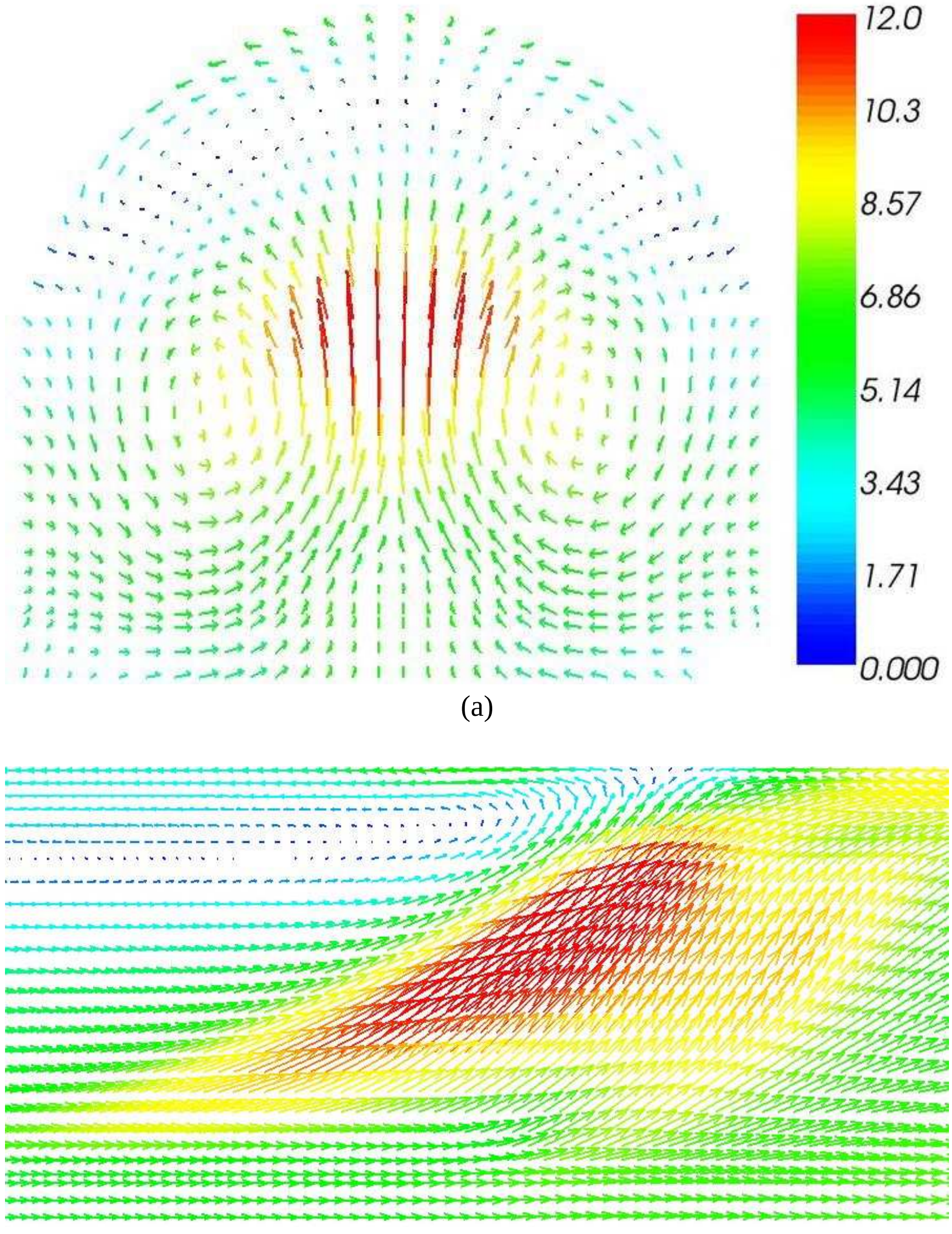
Figure 4: predicted temperatures within the back-layering (a) Case II (rough wall) (b) Case I (smooth wall) at 900 s. Unit: K

During the period of time in which the back-layering in Case I is stopped, smoke travels upstream as far as 13 m from the fire in Case II, in contrast with that smoke is contained at 11 m upstream in Case I. This indicates that while the tunnel surface roughness can affect the occurrence of back-layering, the length of the back-layering may still be effectively contained by the ventilation flow.

The Effect of the Obstacles In the Vicinity of the Fire

The obstacles in the vicinity of the fire affect the occurrence of back-layering. Case I and Case V are similar but in Case V, three columns (heavily insulated instrument trees) on the central axial line of the tunnel in the fire vicinity have been included. Unlike Case I, in Case V back-layering persists beyond 11 m upstream from the fire once it occurs. This is unexpected because the presence of the three columns is expected to cause an increase of the air velocity due to the reduction of the net free area and hence help prevent back-layering from travelling further upstream. The predicted velocities in Case V at the central line of the tunnel show that the air flow slows down slightly after the column approximately 11 m upstream from the fire. The slightly reduced momentum of the central flow may change to some extent the balance of the buoyancy force of the fire plume and the inertia force of ventilation.

The Effect of the Width of the Fire Source



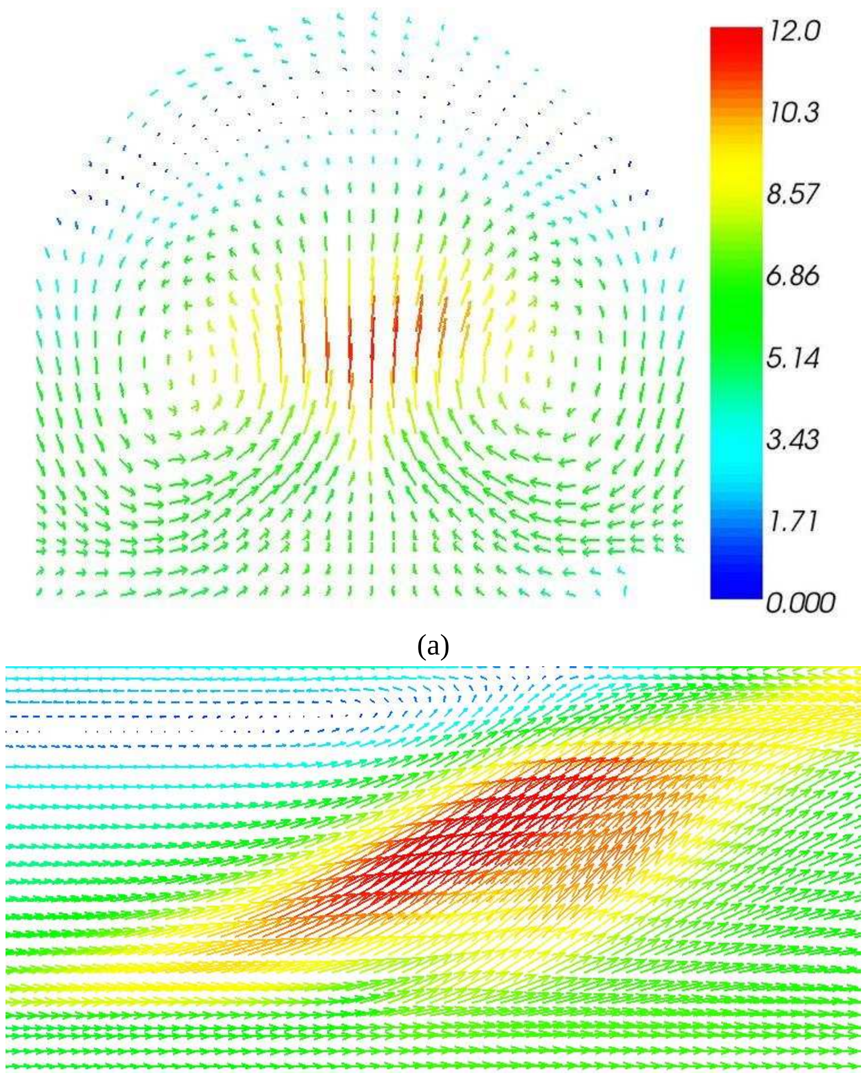
(a)

(b)

Figure 5: predicted velocity vectors in Case VI. (a): cross section 12 m downstream from the fire, (b) central longitudinal plane. Unit: m/s

The occurrence of back-layering is affected by the width of the fire source. In Case VI, the width of the fire is reduced to 2.6m while other model variables are identical to Case I. The back-layering appears beyond 11 m upstream from the fire at approximately 632 s, 28 s earlier than in Case I and persists for the rest of the simulation. An additional simulation of this scenario with 10% excess of the measured ventilation velocities is conducted and results show that back-layering persists once it occurs. This indicates that to prevent back-layering with a narrower fire source, stronger ventilation will be needed. This finding is consistent with the study in[7].

Figure 5 and 6 show the velocity vectors of Case VI and Case I respectively in the cross section 12 m downstream from the fire and the longitudinal central plane at 932 s. Case VI produces stronger vertical flow in the cross section than Case I does. In the longitudinal central plane, Case VI has a thicker fire plume than Case I does. This indicates that the fire plume of Case VI poses much stronger thermal resistance that makes it more difficult to be tilted downstream by the ventilation flow. In addition, Case VI has a much stronger circulation in the area where the back flow starts, which pumps more smoke into the back-layering. Therefore it is more difficult to contain back-layering in Case VI.



(a)

(b)

Figure 6: predicted velocity vectors in Case I. (a): cross section 12 m downstream from the fire, (b) central longitudinal plane. Unit: m/s

Concerns on the Inlet Representation of Jet Fan Flows

Almost all CFD tunnel fire simulations make use of inlet boundary conditions to represent the longitudinal ventilation flows generated by jet fans because it is notoriously difficult to model jet fan operation, in particular when a large scale tunnel fire is simulated. With such a boundary condition, a prescribed amount of air is either pumped into the tunnel at the entrance or some point upstream from the fire or taken out of the tunnel at the exit or some point downstream from the fire. It has been found in numerical studies[6,8] that ventilation flows accelerate towards the fire. This is caused by the combined effects of air entrainment into the fire plume and the thermal blockage of the fire plume for the ventilation air flow[6]. This study arrives at a similar finding. Presented in Figure 7 is the central vertical velocity profile at -11 m upstream from the fire at 960 s produced by Case III. Compared with the velocity profile at the inlet which is -107 m upstream from the fire, it is obvious that the air flow accelerates when approaching the fire.

However, the experimental data show a completely different velocity profile when the ventilation air flow approaches the fire. Presented in Figure 8 are the measured central velocity profiles at 969 s at -107 m and -11 m upstream from the fire. It is evident that there is no remarkable air flow acceleration towards the fire. In fact, the report of the Memorial tunnel tests points out that the presence of the fire actually reduces the tunnel airflow.

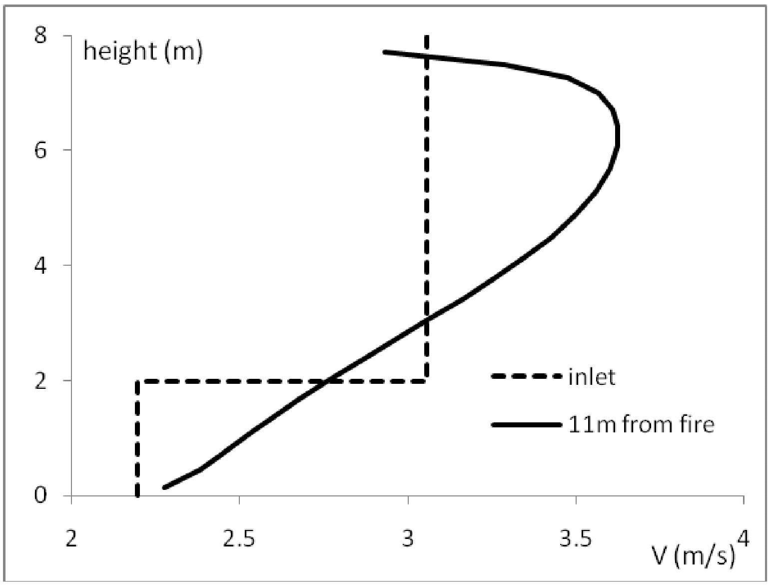


Figure 7: velocity profiles in Case III at inlet (-107 m) and -11 m from the fire.

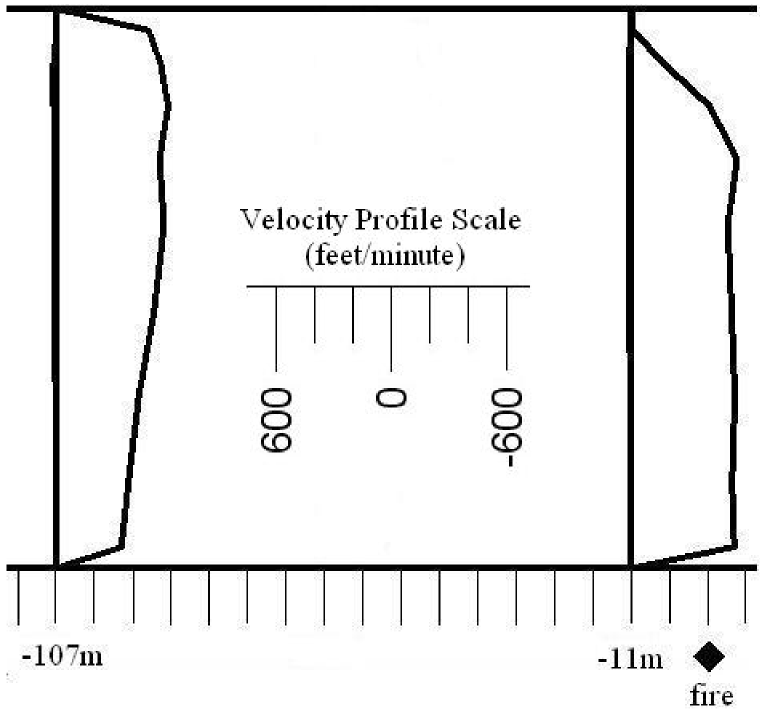


Figure 8: measured velocity profiles at -107m and -11m upstream from the fire at 969 s. (duplicated from [2])

The obvious difference in the behaviour of the simulated and measured ventilation airflow approaching the fire is probably mainly caused by the use of inlet boundary conditions in the CFD simulations to represent the ventilation air flow generated by the jet fans. The ventilation air flow generated by jet fans reacts with the local pressure conditions which can either augment or reduce the tunnel airflow, depending on the direction of ventilation. Therefore, the blockage of the fire plume to the ventilation flow path reduces the tunnel airflow upstream from the fire. However, with inlet boundary conditions, the velocity or mass flux at the boundary is prescribed and the prescribed amount of air is pumped into or taken out of the tunnel regardless the pressure conditions within the tunnel. Due to mass conservation, air flow accelerates towards the fire. For most large scale tunnel tests, the measured velocities were collected at the central line of the tunnel other than the whole cross section. This results in insufficient velocity data to represent the true air flow at the inlet. Therefore, even if measured velocities are used at the inlet, as has been done in this study, the prescribed velocities may not be a true replication of the reaction of the operation of the jet fans to the pressure conditions with the tunnel.

This raises a serious question over the use of inlet boundary conditions to represent the ventilation air flow generated by jet fans even if the velocities used at the inlet are measured values. As discussed previously, the occurrence of back-layering is very sensitive to ventilation velocity when it reaches the critical ventilation velocity. Since the simulated velocity profiles close to the fire are very different to the measured ones, it is very difficult for CFD simulations to produce satisfactory temperature distributions and velocity fields compared with the measure values. However, in spite of these difficulties, the predicted occurrence time of back-layering is in reasonably good agreement with the experimental one. Further study is required to determine a more accurate representation of the air flow in CFD tunnel fire simulations involving jet fans.

CONCLUSIONS

This study shows that at critical ventilation velocity, the occurrence of back-layering in tunnel fire simulations is extremely sensitive to several model setup variables. To a large extent, the success of simulating a tunnel fire test, such as the Memorial Tunnel test, relies on correct replication of the test conditions and setup. However, for most large scale tunnel tests, not all the information required to correctly specify a CFD fire simulation is available and hence sensitivity analyses over critical model setup parameters is essential to correctly interpret differences between predictions and experimental measurements.

This study also raises a serious question over the use of the inlet boundary condition to simulate the longitudinal ventilation air flow generated by jet fans. Further study is required to determine a more accurate representation of the air flow in CFD tunnel fire simulations involving jet fans.

REFERENCES

[1] Thomas PH, The movement of smoke in horizontal passages against an air flow, Fire research Station Note No. 723, Fire Research Station, UK, September 1968.

[2] Massachusetts Highway Department, Memorial tunnel fire ventilation test program, Comprehensive Test Report, Massachusetts, 1995.

[3] Bettis RJ, Jagger SF, and Macmillan A, Interim validation of tunnel fire consequence models: summary of phase 1 tests. The Health and Safety Laboratory Report IR/L/FR/94/2, The Health and Safety Executive, UK, 1994.

[4] Ingason H. and Lönnermark A, Heat release rates from heavy goods vehicle trailer fires in tunnels, Fire Safety J., vol. 40, pp. 646-668, 2005.

[5] Woodburn PJ and Britter RE, CFD simulations of a tunnel fire -- Part I, Fire Safety J., vol. 26, pp. 35-62, 1996.

[6] Wu Y and Bakar MZA, Control of smoke flow in tunnel fires using longitudinal ventilation systems -- a study of the critical velocity, Fire Safety J., vol. 35, pp. 363-390, 2005.

[7] Hwang CC and Edwards JC, The critical ventilation velocity in tunnel fires—a computer simulation, Fire Safety J., vol. 40, pp. 213-244, 2005.

[8] Wang HY, Prediction of soot and carbon monoxide production in a ventilated tunnel fire by using a computer simulation, Fire Safety J., vol. 44, pp. 394-406, 2009.

[9] Kim E, Woycheese JP and Dembsey NA, Fire dynamics simulator (version 4.0) simulation for tunnel fire scenarios with forced, transient, longitudinal ventilation flows, Fire Technology, vol. 44, pp. 137-166, 2008.

[10] Galea E. R., Wang Z., Veeraswamy A., Jia F., Lawrence P. J. and Ewer J., Coupled fire/evacuation analysis of station nightclub fire, Proc of 9th IAFSS Symp, Sep. 21-26, 2008, Karlsruhe, Germany, pp 465-476.

[11] Wang, Z., Jia, F., Galea, E. R., Patel, M. K. and Ewer, J., Simulating one of the CIB 14 round robin test cases using the SMARTFIRE fire field models. Fire Safety Journal, vol.36, pp. 661-677, 2001.

[12] Jia F., Patel M.K., Galea E.R., Grandison A. and Ewer J., CFD Fire Simulation of the Swissair Flight 111 In-flight Fire – Part II: Fire Spread within the Simulated Area, The Aeronautical Journal of the Royal Aeronautical Society, vol. 110, pp. 303-314, 2006.

[13] Tennekes H and Lumley JL, *A First Course in Turbulence*. MIT Press, Cambridge, Massachusetts, 1982.

[14] Woodburn PJ and Britter RE, CFD simulations of a tunnel fire -- Part II, Fire Safety J., vol. 26, pp. 63-90, 1996.

[15] Van Maele K and Merci B, Application of RANS and LES field simulations to predict the critical ventilation velocity in longitudinally ventilated horizontal tunnels, Fire Safety J., vol. 43, pp. 598-609, 2008.