FIRE SAFETY CASE STUDY OF A RAILWAY TUNNEL: SMOKE EVACUATION

by

Karim VAN MAELE and Bart MERCI

Original scientific paper
UDC: 656.085.5:614.72:519.876.5
BIBLID: 0354-9836, *11* (2007), 2, 207-222

When a fire occurs in a tunnel, it is of great importance to assure the safety of the occupants of the tunnel. This is achieved by creating smoke-free spaces in the tunnel through control of the smoke gases. In this paper, results are presented of a study concerning the fire safety in a real scale railway tunnel test case. Numerical simulations are performed in order to examine the possibility of natural ventilation of smoke in inclined tunnels. Several aspects are taken into account: the length of the simulated tunnel section, the slope of the tunnel and the possible effects of external wind at one portal of the tunnel. The Fire Dynamics Simulator of the National Institute of Standards and Technology, USA, is applied to perform the simulations. The simulations show that for the local behaviour of the smoke during the early stages of the fire, the slope of the tunnel is of little importance. Secondly, the results show that external wind and/or pressure conditions have a large effect on the smoke gases inside the tunnel. Finally, some idea for the value of the critical ventilation velocity is given. The study also shows that computational fluid dynamics calculations are a valuable tool for large scale, real life complex fire cases.

Key words: *tunnel fire safety, smoke control, simulation, fire dynamics simulator*

Introduction

If there is an emergency in a train inside a railway tunnel, a common practice is for the train to continue its way and to prohibit the train to stop inside the tunnel, even in the case of a fire, in order to increase the survival chances of the passengers. Although it is possible that the train is disconnected from the power supply in case of a calamity, in many cases it can be shown that the train has enough momentum to reach the tunnel exit, even if the train is travelling in the uphill direction of a moderately sloped tunnel. This assumption is certainly valid for nowadays high speed trains. However, in a worst case scenario, the possibility must be considered that a burning train is forced to stop inside a tunnel. Therefore, it is necessary already at the construction stage of a tunnel to provide all safety measures to ensure highest possible level of safety for the passengers in such an exceptional case.

DOI:10.2289/TSCI0702207M

In this example we consider a single bore sloped tunnel with double track. We also assume that there is also one emergency exit in tunnel (*e. g*. halfway the length of the tunnel), so that it is to be expected that passengers will have to move inside the tunnel while there is a burning train, producing smoke and heat. The smoke rises due to the buoyancy force and impinges onto the tunnel ceiling which deflects the smoke in two opposite directions, creating a wall-bounded plume. The only option to create a smoke-free route for the passengers and also for emergency purposes (*e. g.* for fire fighters), is to direct the smoke in one single direction. Consequently, it is of vital importance to understand the behaviour of smoke arising from the fire. In this paper, we apply computational fluid dynamics (CFD) techniques to investigate the natural behaviour of the smoke of a design fire in a sloped tunnel. Since hot smoke has the tendency of flowing in the uphill direction of a tunnel due to buoyancy, one might expect that a smoke-free path can be created in the downhill direction of the tunnel. However, several parameters may have an influence such as the possible chimney effect of the sloped tunnel in the absence of fire (due to temperature differences between the environment and inside the tunnel), wind effects at the tunnel portals, pressure differences between the two access openings, and the angle of inclination of the tunnel.

Computational fluid dynamics approach

The fluid flows that accompany a fire are driven by buoyancy forces, which are created by the density difference between the hot combustion products, *i. e*. smoke, and the ambient air. By considering the fundamental laws of conservation of mass, momentum, and energy, the governing equations describing these flows can be derived and are generally known as the Navier-Stokes equations. It is impossible to solve these equations analytically, and therefore we have to rely on numerical techniques. Typically for CFD, the finite volume method is used. This means that the physical space, in which we want to solve the flow field, is divided in small control volumes (the numerical grid or mesh) and the governing equations are integrated over each volume. The next step consists of transforming these integrated equations into a discretized form in order to arrive at a set of algebraic equations which can be solved numerically.

As for most flows encountered in nature, fire generated flows are turbulent. Everyone who has once observed a camp fire for example, knows that the form of the flames is highly irregular – every instant the shape of the fire and the flames changes. This is due to the turbulent nature of the fluid flows in the fire and indicates the chaotic and irregular, both in time and in space, behaviour of the gases. If we want to compute these flows with a high degree of accuracy, we have to apply a very fine mesh and small time steps. Consequently, the time and memory required for such calculations become tremendously large. Clearly, this is not a feasible approach for practical, day-to-day applications.

Hence, the effect of turbulence on the flow field has to be incorporated by some model. In this work, we used the Fire Dynamics Simulator, version 4, (hereafter FDS) of the National Institute of Standards and Technology (NIST), USA [1] which applies the Large-Eddy Simulation (hereafter LES) technique. In this technique, the large scale motions of the flow, which are the most energetic part and represent the large scale mixing processes, are resolved while the small scale motions are modelled. The underlying idea is that small scale motions contain only a small part of the turbulent kinetic energy and that the small scale eddies tend to have a more universal and isotropic character, independent of the geometry of the domain in which the flow takes place. The small scale motions are removed by filtering the Navier-Stokes equations. In FDS, the filter width is related to the size of the grid cells and the Smagorinsky model is applied to incorporate the effect of the sub-grid scale stresses.

Combustion modelling

When a fire takes place, chemical reactions occur between fuel and oxygen. These reactions result in the production of combustion products, such as H_2O and CO_2, and also in the release of heat. Due to the release of heat, the temperature of the combustion products increases and their density decreases, creating the buoyancy force which is the driving force for fire generated flows. Consequently, it is important and necessary to include the combustion process in simulations of fires. However, the chemical reactions taking place are very complicated and include a large amount of different species. Moreover, the reactions take place on time and length scales which are orders of magnitudes smaller than those typically for fluid flows. All this makes it impossible to calculate the combustion process from its first principles and again a model has to be used to incorporate the effect of combustion.

In FDS, two approaches are available for combustion: a global one-step, finite-rate chemical reaction and a mixture fraction-based model. The former model will not be discussed here as we did not apply it in our simulations. The latter model is based on the assumption that the combustion process is mixing-controlled. Therefore, all species can be described in terms of the mixture fraction variable. State relations give the connection between each species and the mass fraction. FDS applies piecewise linear functions as state relations. For the flame itself, a "flame sheet" model is used, representing the flame as a two-dimensional surface. By calculating the local consumption rate of oxygen at the flame sheet, applying the state relation for the oxygen mass fraction, the local heat release rate is computed. Hereby, the assumption is made that the heat release rate is directly proportional to the oxygen consumption rate, independent of the fuel involved.

The quality of this combustion model is limited by the resolution of the underlying mesh. If the grid is too coarse to accurately resolve the flames, the procedure to calculate the local heat release rate due to combustion does not work very adequately anymore. Therefore, in FDS two precautions are taken to overcome the problems related to coarse grids. The reader is referred to the Technical Reference Guide of FDS [2] for further details on this matter. Clearly, for practical applications where we have to cope with large physical dimensions and large fires, it is important to have such remedies since current day computer power puts an upper limit on the fineness of the underlying computational grid.

Radiation modelling

A portion of the energy that is released during the chemical reactions is transferred to nearby walls and the environment through radiative heat transfer. For sufficiently large fires, radiative heat transfer is the dominating mode of heat transfer. It is thus important to incorporate this process into fire simulations. In FDS, this is done by solving a radiation transport equation for a non-scattering grey gas. This equation is solved by the Finite volume method.

Implementation of a tunnel geometry and design fire

The example tunnel we examine in this paper has an arched cross-section with a nominal width of 10 m. The height of the tunnel at its centreline is 8 m. As FDS works with rectangular grids, curved geometries have to be approximated using rectangular obstructions ("stair stepping"). Figure 1 presents the actual geometry and the "stepped" geometry that was used in the simulations. The obstructions were chosen in such way that the area of the stepped cross-section is equal to the area of the cross-section of the original tunnel. Figure 2 gives an impression of what this geometry looks like in FDS. Also shown in fig. 2 is the applied computational grid. The grid cells are cubic with an edge of 0.5 m.

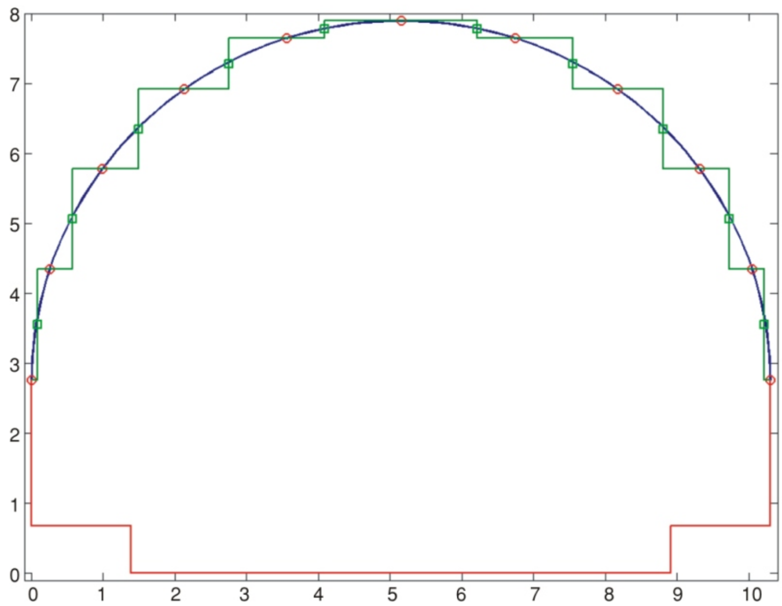


Figure 1. Stepped approximation of the cross-section of the tunnel

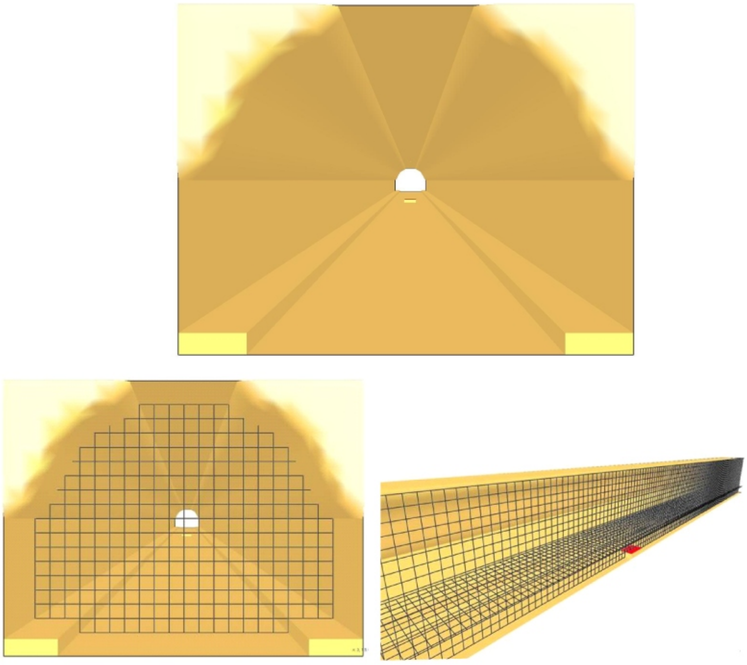


Figure 2. Tunnel geometry in FDS and computational grid

In this study, we consider a design fire with a constant power of 14 MW, which is a typical safe value for passenger train fires. In [3], the heat release rate from a full scale fire test of a German passenger train is presented. The measurement was done during the Eureka project [4]. For that full scale test, the heat release rate (HRR) was measured during almost 140 minutes with a peak HRR around 14 MW. This fire is modelled in FDS as a square burner with a dimension of 2 × 2 m. The surface of the burner is located at a distance of 0.5 m above the tunnel floor. In this study, no flame spread phenomena are considered. All simulations are performed with this constant HRR fire output.

Results and discussion

Influence of the length of the simulated tunnel section

Tunnels in real life can be very long, in the order of several kilometres. If we want to perform numerical simulations for the total length of such a tunnel, this would require several millions of grid cells. This would require an enormous amount of memory and it would take a long time until simulations come to an end. Clearly, this is not affordable for practical applications and some simplification of the problem has to be made.

In this section we focus on the local smoke behaviour in a horizontal tunnel. If the tunnel has a slope, there will be a global chimney effect due to the heat released by the fire source. This global effect should then be superposed onto the local behaviour we discuss here. We do not discuss the global chimney effect in this paper.

Since here we restrict ourselves to the local behaviour of the smoke produced by the fire, we are able to simulate a reduced section of the tunnel instead of the whole length. In order to examine the influence of the length of the simulated section, we performed simulations for a section of 100, 200, and 300 m. The tunnels in this case have no slope. For all simulations a fire of 14 MW is applied with the burner located centrally in the tunnel section. The temperature is monitored at a point 10 m away from the burner and at a height of 7.8 m. The monitor points are located in the symmetry plane of the tunnel. The results for the three sections are presented in fig. 3. The results show that the instantaneous values of the temperature for the three lengths can differ strongly. However, the mean temperature (calculated from $t = 40$ s to $t = 200$ s) is for the three cases around 290 °C. Thus, although the length of the simulated tunnel has an influence on the instantaneous temperature values, there is hardly any impact on the mean flow field. According to this, the error made by reducing the length of the simulated tunnel is negligible, with respect to the behaviour of smoke in the immediate vicinity of the fire.

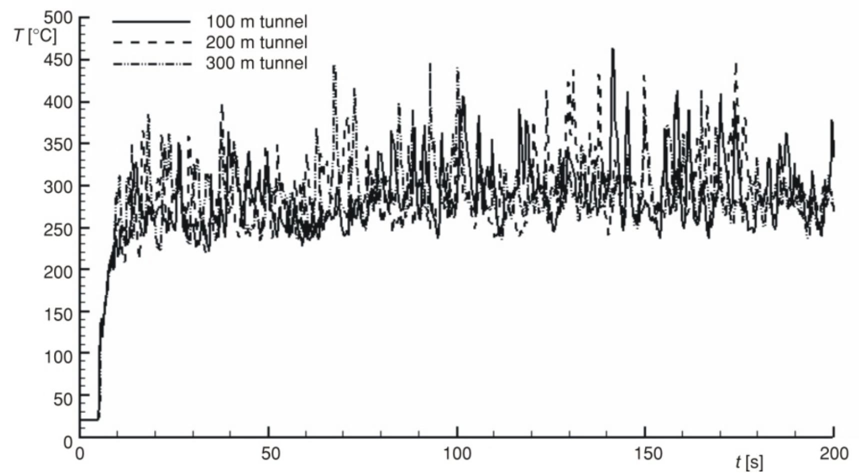


Figure 3. Comparison of thermocouple temperatures for the tunnels with different length

Influence of the tunnel slope

As a fire in a long sloped tunnel (in the order of kilometres) evolves, hot combustion products are produced and the tunnel fills up with hot and light gases. If we consider the entire tunnel globally, this resembles a large chimney, as previously mentioned. Due to the temperature difference between the gases inside the tunnel and the temperature outside the tunnel, the hot gases tend to move in the uphill direction of the tunnel. However, it takes some time to arrive at this situation. Simple analytical calculations show that for a tunnel with a length of 10 km and with a slope of 2%, it takes approximately 10 minutes for the chimney effect to become effective. At this stage, we can expect that the smoke moves in one single direction, implying there is a smoke-free path created, in the downhill direction, in a natural way.

However, considering the safety of the passengers in case of an emergency, the early stages of a fire have to be regarded. Therefore, for these first minutes, the large scale chimney effect can be decoupled from the local behaviour of the smoke. It is then meaningful to perform the simulations on a reduced length section of the tunnel. If it is necessary to incorporate the large scale chimney effect, after approximately 10 minutes, the entire length of the tunnel has to be incorporated in the simulations, which demands large computational power as already mentioned. In order to evaluate the effect of the slope of the tunnel on the local behaviour of the smoke during the first instants of a fire, we performed simulations of a tunnel section of 100 m with three different slopes (0, 2, and 10%). The effect of the slope is incorporated in FDS by changing the angle between the gravity vector and the axis of the tunnel. The results are presented in figs. 4-6. For the results presented, the tunnel is inclined such that the uphill direction is from the left to the right of the figure. The results show that for the local behaviour of the smoke in the first instants of the fire (say, the first 10 seconds), the effect of the slope is negligible. Then

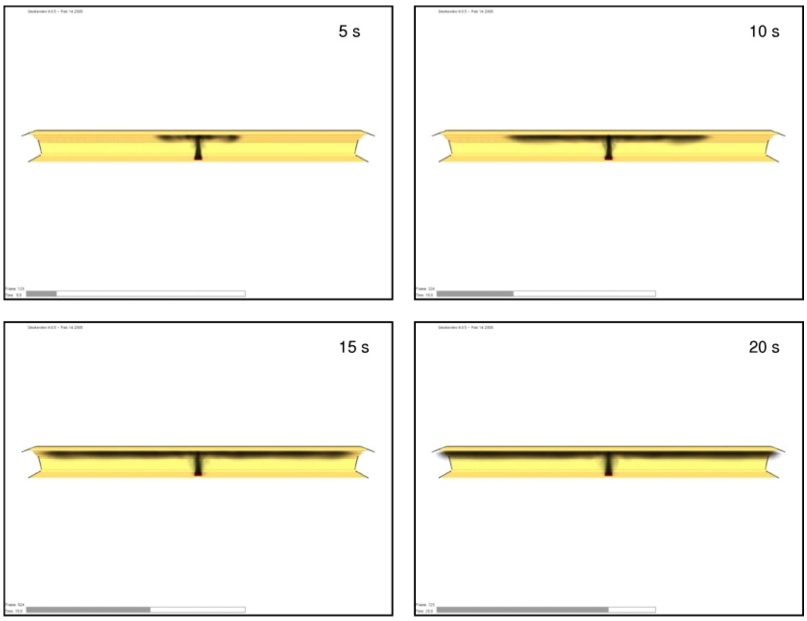


Figure 4. Results for the simulation of the tunnel without slope

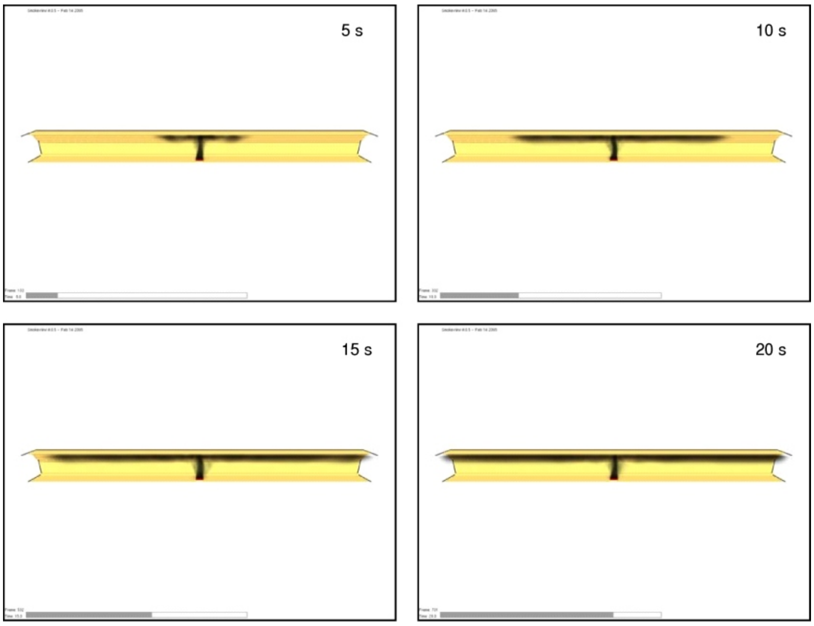


Figure 5. Results for the simulation of a tunnel with a slope of 2%

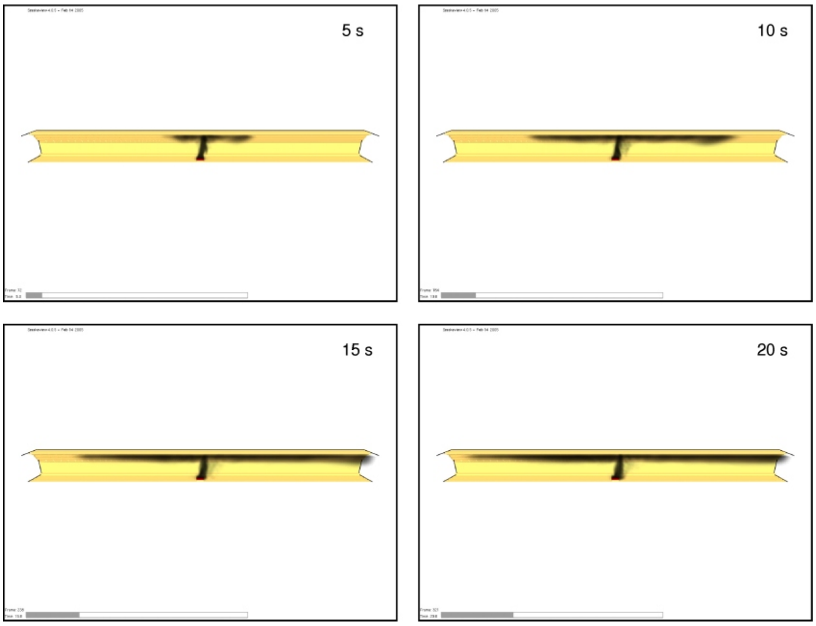


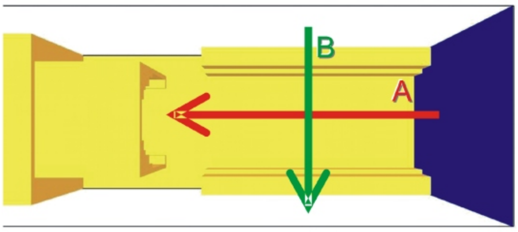
Figure 6. Results for the simulation of a tunnel with a slope of 10%

some asymmetry in the smoke motion becomes visible, but even for the slope of 10%, the smoke travels in both opposite directions under the tunnel ceiling. Consequently, in this stage of the fire, the effect of the slope is not sufficient to create a smoke-free path for the passengers.

The effect of external wind

The behaviour of the smoke can also be influenced by external wind. Two extreme situations are considered in the simulations: an external wind parallel with the tunnel axis (A) and a wind perpendicular to the tunnel axis (B), see fig.7. In both situations A and B the wind is parallel to the tunnel floor. To examine the effect of these winds, a possible set-up for the surrounding area at one portal of the tunnel is modelled. In this paper, some blockings created by two hills that run along the two railway tracks are modelled. It is stressed out that this set-up just serves as an example of what is possible for a real life tunnel.

Figure 7. Schematic representation of the external wind conditions applied at one side of the tunnel



Results for the wind parallel to the tunnel axis

Two wind velocities are considered: 5 m/s (18 km/h) and 10 m/s (36 km/h). The results are presented in figs. 8 and 9. The results show that for the wind velocity of 5 m/s, the effect of the wind is not strong enough to blow all the smoke in one single direction. The simulation shows that a stable layer of smoke is formed in the opposite direction of the wind. This layer becomes stable after approximately 150 s.

For the case in which a wind velocity of 10 m/s is applied, the simulation shows that the wind is strong enough to force all the smoke in a single direction so that a smoke-free route is created towards the tunnel exit. After 40 s, the back-layer of smoke disappears.

We note that these values for the wind velocity are not directly related to the critical ventilation velocity. As the wind approaches the tunnel portal, a large portion of the air flows above the tunnel portal instead of entering the inside of the tunnel. This is due to the flow resistance that is created by the tunnel. Consequently, only a small portion of the air mass put in motion by the wind will enter the tunnel and therefore the air velocity inside the tunnel is lower than the velocity of the wind outside the tunnel. It is shown further in the paper that the critical ventilation velocity for a 14 MW fire in this tunnel is around 2 m/s.

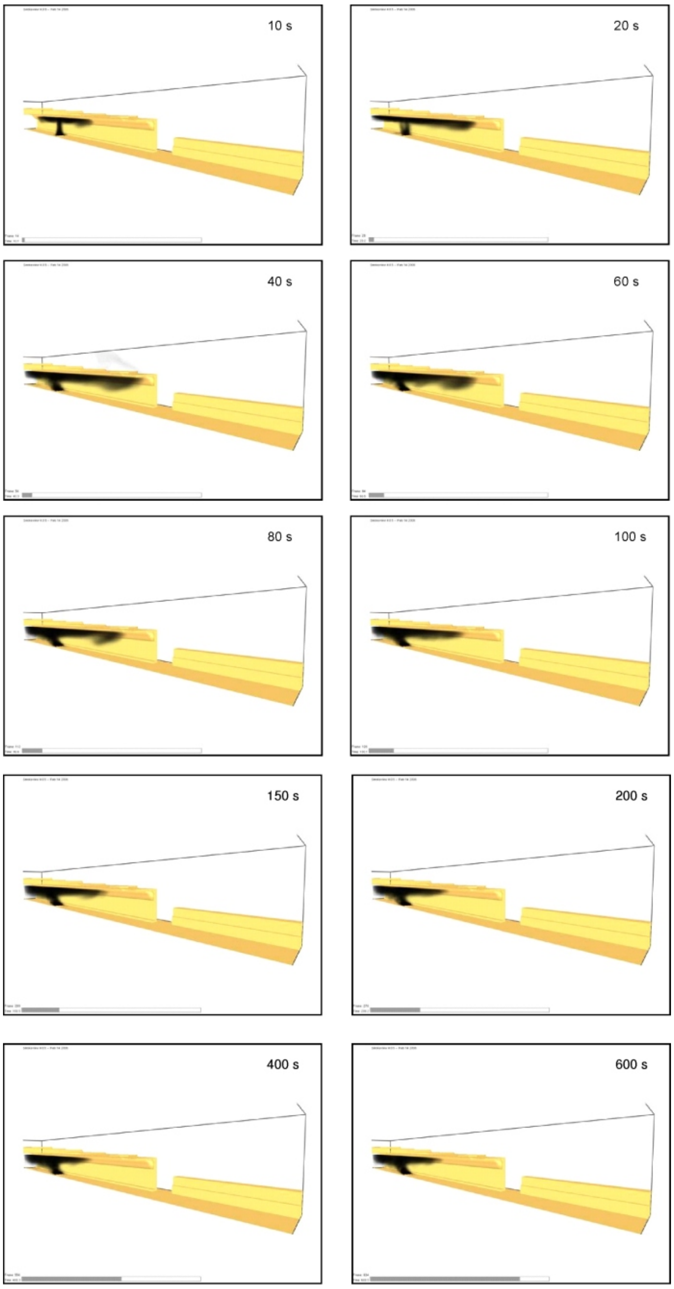


Figure 8. Results for the simulation with an external wind of 5 m/s (parallel with the tunnel axis)

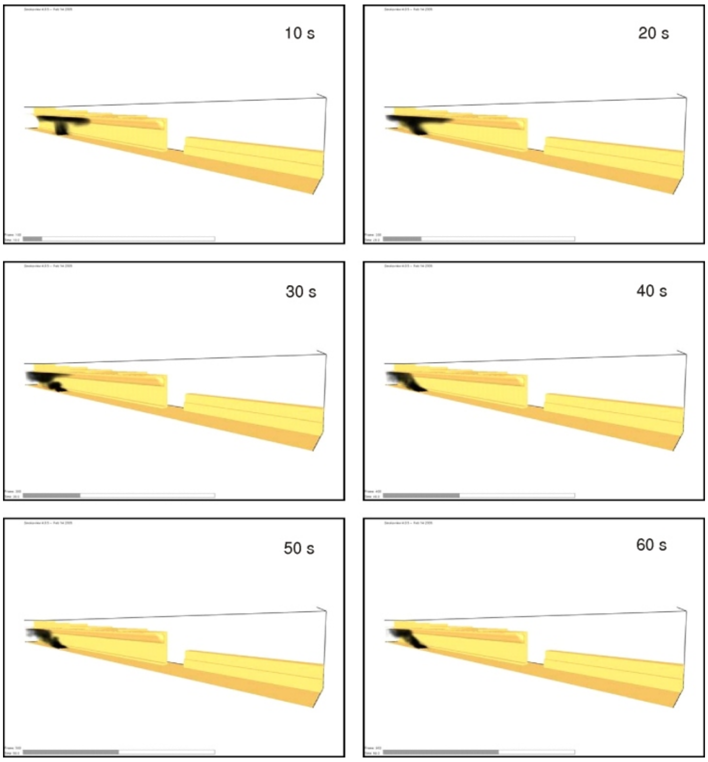


Figure 9. Results for the simulation with an external wind of 10 m/s (parallel with the tunnel axis)

Results for the wind perpendicular to the tunnel axis

Two wind velocities are considered in this case: 10 m/s (36 km/h) and 20 m/s (72 km/h). The results are presented in figs. 10 and 11.

When the external wind blows perpendicular to the tunnel axis, a low pressure region is created at the tunnel exit due to the curvature of the flow streamlines. This low pressure sucks the smoke out of tunnel. This effect can be seen in figs. 10 and 11. However, even for the wind velocity of 20 m/s, which is already a high value and considered as stormy conditions, the action of the wind is not strong enough to force all the smoke out of the tunnel in order to create a smoke-free path. It is clear from the simulations that the effect of wind blowing perpendicular to the tunnel axis has a much smaller effect than wind blowing directly towards the tunnel portal. Where the parallel wind creates an impulse effect inside tunnel (and consequently there is a transfer of momentum between the air entering the tunnel and the smoke gases, by which the gases can be forced in one single direction), the perpendicular wind only generates a pressure drop at the tunnel portal.

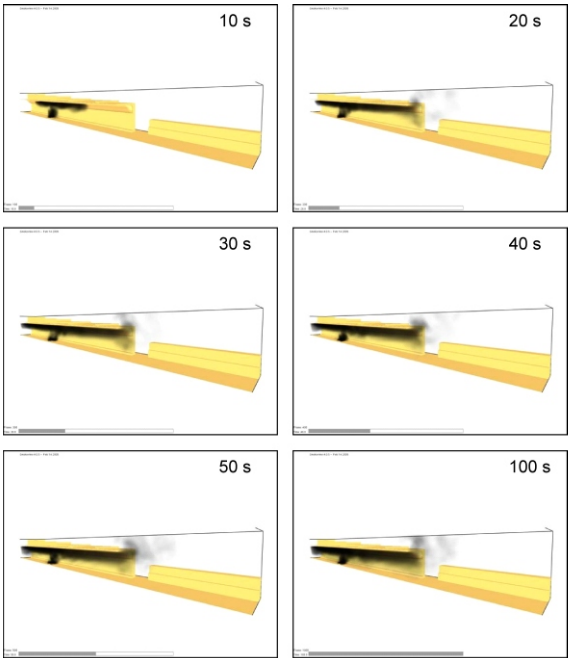


Figure 10. Results for the simulation with an external wind of 10 m/s (perpendicular to the tunnel axis)

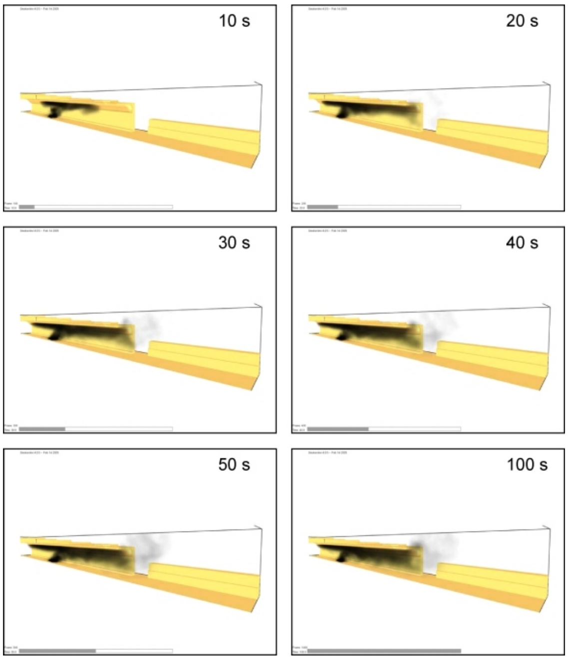


Figure 11. Results for the simulation with an external wind of 20 m/s (perpendicular to the tunnel axis)

In the first place, these results indicate, as could be expected, that external winds have a non-negligible influence on the behavior of the smoke inside the tunnel. Secondly, depending on the direction and the velocity of the wind, a totally different effect on the smoke movement can be expected. In case of an emergency, the wind conditions may be favorable to create a smoke-free path in the proper direction for the passengers to escape from the fire. The ideal condition is when the wind acts in the same direction as the chimney effect but it is not guaranteed that this situation will take place. Nonetheless, in most practical circumstances the wind is unable to extract the smoke or blow the smoke in one single direction.

Influence of the naturally created air current inside the tunnel

As already mentioned, we consider a sloped, long railway tunnel in this work. We take a tunnel of 10 km with a slope of 2%. For this example, there is a height difference of 200 m between the two portals of the tunnel. As pressure decreases with height, a static pressure difference of about 14 hPa exists between the portals. This pressure difference does not induce any flow of air inside the tunnel. However, for long tunnels where the air inside has a lot of time to transfer heat to tunnel walls or receive heat from the walls, a temperature difference may occur between the air inside the tunnel and the ambient air outside. As the tunnel is sloped, this creates the aforementioned chimney effect, *i. e.* a current of air is induced inside the tunnel, in the absence of a fire. In wintertime conditions, it is to be expected that cold ambient air is heated up inside tunnel resulting in warm air inside, so that the density becomes lower compared to the ambient air. Consequently, air is flowing in the uphill direction of the tunnel. In summertime, the opposite is to be expected: warm ambient air is cooled down inside tunnel resulting in air inside with a higher density compared to the ambient air. Consequently, air is flowing in the downhill direction of the tunnel. Here, we consider a temperature difference of 10 °C between the inside of the tunnel and the outside, with the warmer air inside the tunnel. Simple analytical calculations based on the Darcy-Weisbach equation for pressure loss in ducts and pipes and on the Colebrook equation to determine the friction factor, show that an air flow of 2 m/s inside the tunnel is a realistic value. The effect of this airflow on the local behavior of smoke gases during the early stages of a fire, is modeled as an aspirating fan at one end of the tunnel and a ventilation velocity of 2 m/s is applied. The result of the simulation is presented in fig.12. The arrow indicates the direction of the flow induced by the fan.

The results show that by applying a ventilation velocity of 2 m/s nearly all smoke gases are extracted from the tunnel. This creates a smoke-free path in the direction opposite to the direction of ventilation. However, we notice on the figures above there is still some part of smoke which travels against the ventilation direction, possibly creating harmful conditions for escaping occupants of the tunnel. This indicates that 2 m/s is somewhat below the critical ventilation velocity in this c ase. The critical ventilation velocity is the velocity at which there is no back-layering of the smoke. It is an important parameter when designing a mechanical ventilation system in tunnels. The simulation

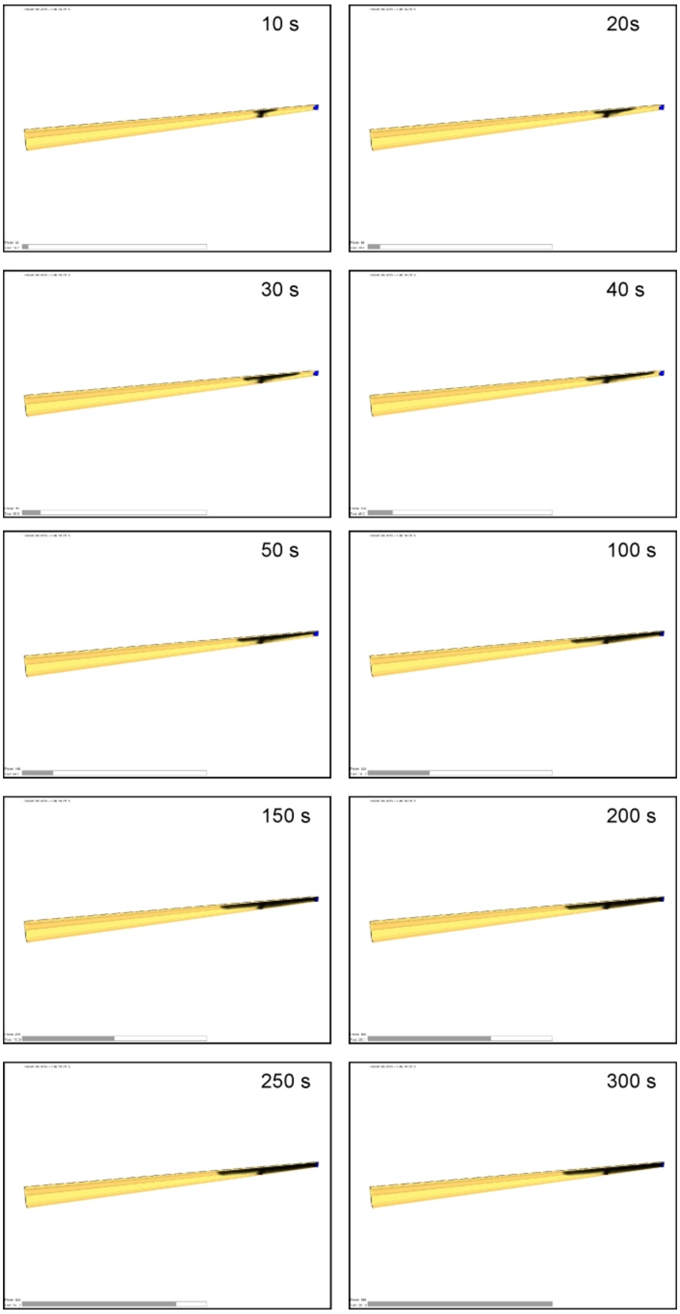


Figure 12. Simulation result for a 14 MW fire with a ventilation velocity of 2 m/s

performed with a ventilation velocity of 2 m/s indicates that after approximately 150 s a stable layer of smoke is established in the opposite direction of the ventilation.

The simulation also reveals that, under certain circumstances, (super)critical ventilation conditions can be achieved by the naturally created chimney effect due to a temperature difference between the air inside and outside the tunnel. However, this effect should not be relied on to force the smoke out of the tunnel since some aspects are uncertain: the temperature difference may be too small to create a sufficient air flow in the tunnel; the direction of the airflow may change depending on the weather conditions (warm inside the tunnel and cold outside *versus* cold inside the tunnel and warm outside); the fire itself can alter the density difference between the gases inside and outside of the tunnel, hence altering the chimney effect, by releasing heat inside the tunnel.

Conclusions

Depending on the position of the burning train with respect to the emergency exits, a certain ventilation strategy can be followed. The simulations presented in this paper primarily focus on the possibility of natural ventilation in sloped tunnels. Different aspects on the natural behavior of smoke have been considered using the CFD technique. All simulations are performed applying FDS.

It was shown that the length of the simulated tunnel section has a negligible influence on the results with respect to the local smoke movement during the first stages of a fire. Secondly, we presented results concerning the effect of the slope on the local smoke movement, illustrating that the slope effect can be neglected.

Next, the influence of external winds at one portal of the tunnel was examined. The simulations showed that the wind conditions can be favourable to create a smoke-free path but it is as well possible that the wind conditions have an undesired influence. The simulations also showed that there is large difference between the effect created by a wind parallel to the tunnel axis and that of a perpendicular wind. The former influences the smoke behavior due to an impulse effect, while the latter sucks the smoke out of the tunnel due to a low pressure created at the tunnel portal. Overall, the direction in which the smoke will travel is unknown in case of an emergency and the flow of combustion gases can not be controlled in a suitable way. The same is true for the chimney effect created by the temperature difference between the air inside and outside the tunnel in the absence of a fire. Furthermore, the release of heat during a fire will alter this effect. Finally, this study also shows that CFD simulations are a valuable tool for real life, real scale fire problems. CFD simulations can indeed provide quantitative information for the design of mechanical smoke extraction systems.

Acknowledgments

The authors are grateful for the financial support from BOF project 011/013/04 (Bijzonder Onderzoeksfonds; Research Fund of Ghent University).

References

[1] http://fire.nist.gov/fds
[2] McGrattan, K., Fire Dynamics Simulator (Version 4) – Technical Reference Guide, NIST Special Publication 1018, 2005
[3] Johnson, P., Faller, G., Hoffmann, N., Luo, M., Coles, A., Fire Safety for Passenger Rail Cars – A Global Perspective on Infrastructure Design, *Proceedings*, Conference on Fire Safety in Terrestrial Passenger Transportation, Santander, Spain, 2005, pp. 61-72
[4] EUREKA. Fires in Transport Tunnels: Report on Full-scale Tests. EUREKA-Project EU499: FIRETUN, Studiengesellschaft Stahlanwendung e.V., D-40213 Düsseldorf, Germany, 1995

Authors' address:

K. Van Maele
B. Merci
Postdoctoral fellow of the Fund for Scientific Research, Flanders, Belgium (F.W.O.-Vlaanderen)
Ghent University – UGent, Faculy of Engineering, Department of Flow, Heat and Combustion Mechanics
41, Sint-Pietersnieuwstraat, B-9000 Ghent, Belgium

Corresponding author K. Van Maele
E-mail: VanMaele@UGent.be

Paper submitted: March 30, 2006
Paper revised: April 30, 2006
Paper accepted: September 1, 2006