

Modelling Fire in Tunnels: A Large Scale Validated Two-Step Modelling Method

Benjamin Truchot

INERIS - Verneuil en Halatte, Rue Jacques Taffanel, 60550 Verneuil-en-Halatte, France

Abstract: Fire is a quite common phenomenon in tunnel and being able to model its consequences with a good precision is crucial to design adapted safety measures. Modelling the fire behaviour in tunnel is quite challenging. Managing large-scale experiment for all the possible configurations is however economically unrealistic. This paper presents an experimental real scale fire test that was used not only for demonstrating the fire behaviour but also for evaluating the capabilities of the FDS (fire dynamics simulator) fire code to model fire consequences too. It enables highlighting the importance of wall and inlet boundary condition treatment. Keeping in mind that predicting fire development using a CFD (computational fluid dynamics) code is quite impossible, a two-level approach is discussed with an analytical model to predict the fire curve and a CFD model for predicting smoke propagation, temperature and toxic gases distribution inside the tunnel. The comparisons show a good agreement between experimental fire test and CFD modelling but also let appear requirements when using CFD.

Key words: CFD modelling, real fire experiment, tunnel.

1. Introduction

Fire in tunnels can generate dramatic consequences as shown in the past. Following the “Mont Blanc” fire in 1999, a new regulation was created in France with different prescriptions for safety measures. Those measures are of course hardly applicable to existing tunnel and must be adapted to the real risk.

The “tunnel du Roux”, in Ardeche, a region in the middle of France, is a typical case where a compromise has to be found. This tunnel is located in a mountainous region invaded by snow during long periods in winter. This tunnel is then crucial for the local inhabitants. This tunnel is however not provided with safety equipment and the question of consequences in case of fire was asked. To demonstrate the smoke propagation and fire consequences inside the tunnel in case of a car fire, a real test was managed with different measures.

Then, to explore the different possibilities in terms of fire, and considering that it was unrealistic to manage several fire tests, for obvious economic reasons a

coupled approach was validated on the experiments and then used for predicting consequences in other cases. This coupled approach includes a CFD (computational fluid dynamics) model for predicting fire consequences associated with an algebraic model to predict the fire curve.

This paper presents the result of the experimental test and the comparison with numerical predictions.

2. Two-Step Modelling Strategy

Fire consequences modelling in tunnels requires predicting temperature raise inside the tunnel, smoke behaviour and toxic gases distribution inside the infrastructure. One of the key points in such a modelling consists in the definition of the evolution of the fire heat release rate with time used as an input in the 3D model. Regarding car fire, several tests were realised [1, 2] and lead to normalized fire curve for such vehicles [3]. Such a curve does not however enable to predict real car fire considering these curves give just an estimation of the maximum power and duration

linked with linear curve. Furthermore those curves are only provided for one “small vehicle” and one “large vehicle” while there are differences between vehicles. To make a better prediction of the heat release rate from a vehicle fire, a specific model was developed. The fire curve predicted using analytical model was then introduced in the FDS (fire dynamics simulator) CFD fire code to evaluate the ability of this code to predict the fire consequences. Having validated the numerical approach, it was then used to model some other scenarios.

2.1 Mathematical Modelling of a Car Fire

Considering total heat release rate for a vehicle is governed by the heat release of the different components, this tool enables to compute the fire curve by summing the individual heat release, considering also a propagation time. The car is split into 5 parts, namely: wheels, engine block, interior, trunk and fuel tank. The average combustion velocity and heat of combustion is then computed for each part considering the distribution between following materials (Table 1).

Table 1 Average combustion velocity and heat.

Material	H_c (MJ/kg)	v (g/m ² s)
(1) Polymers	35.0	20.0
(2) Elastomers	35.0	20.0
(3) Oil	40.0	30.0
(4) Battery	35.0	25.0
(5) Tires	30.0	20.0
(6) Fuel	42.0	55.0

Following relation is used for averaging:

$$\bar{\varphi} = \frac{\sum_i m_i \varphi_i}{\sum_i m_i}$$

m_i is the mass of i th combustible and the interested quantity.

The non-combustible materials are considered as energy well. Each component fire curve follows three phases: the fire growth, a steady state and a linear

decrease phase. The contribution of each element is then summed.

2.2 CFD Fire Modelling

As mentioned previously, the temperature and smoke distribution along the tunnel was modelled with a CFD code. The FDS fire code [4] was used. It is clear that predicting the fire growth and evolution with a CFD code is quite difficult [5] but this code was largely validated for smoke propagation prediction in tunnel [6, 7], this paper provides a validation for a real car fire in naturally ventilated tunnel. It also highlights the importance of the boundary conditions. It was then chosen for the present modelling. This code enables solving the fluid mechanics equations, continuity, momentum and energy in the computational domain. It also permits to manage largely parallel computations with an acceptable speed-up.

One important point to be mentioned is that turbulence modelling is based on an LES approach. This of course influences the mesh generation because of the constraints imposed by such a modelling approach. For such an LES modelling approach, the cutting scale, defined by the cell size, must be placed in the inertial zone of the turbulence spectrum. This will be highlighted in the numerical description of the paper. It is also important to have in mind when using an LES approach for smoke propagation in tunnel with natural ventilation that the turbulent fluctuations due to external wind perturbations must be considered. The turbulence intensity has to be modelled through velocity fluctuations on tunnel portal. This was done using a sinusoidal variation along time with measurement-based frequencies.

It was also demonstrated that predicting the correct smoke behaviour, including the backlayering phenomena requires to model tunnel walls characteristics with a great precision [6]. This is to have a good prediction of the energy transfer from the smoke to the walls. This energy transfer is also crucial when considering the stratification of the smoke. This stratification

results from the equilibrium between density gradient due to thermal effect that generates stratification and turbulent mixing due to the flow that destroys the stratification [8].

3. Description of the Experimental Campaign

3.1 Tunnel Description

The “Tunnel du Roux”, is a 3,325 m long French tunnel located in Ardeche, in the middle of France, at 950 m of altitude. The tunnel section is about 41 m² (Fig. 1), and it has the particularity to be strictly linear. Safety measures in this tunnel are limited and it is not equipped with ventilation system. Consequently, smoke propagation in the tunnel is governed by pressure difference between tunnel portals, except in winter when one of the two portals is closed to prevent ice formation; there is then no flow inside. However, this closing door is equipped with an automatic system to let vehicle go through the tunnel.



Fig. 1 Section of the “Tunnel du Roux”.



Fig. 2 Images of car used for the experiment.

3.2 Introduction of the Fire Source

The fire in the tunnel was produced using a real car, a Ford Fiesta, from which the main polluting materials as battery and oil were removed (Fig. 2). The fuel tank was filled with 10 L of gas-oil and the fire was ignited on the passenger seat after having made a cut in its envelope to uncover the foam, several centilitres of heptanes were used for ignition.

Because of the natural ventilation of the tunnel and of the complex wind profile in the surrounding, it was impossible to predict the flow direction for the morning of the test. It was decided to locate the car at 275 m of the South portal of the tunnel.

The tunnel was protected from thermal effects along about 20 m with mineral wool to avoid damages during the test. Such a protection is of course to be considered when modeling the fire because it has an influence on thermal exchanges between smoke and tunnel walls.

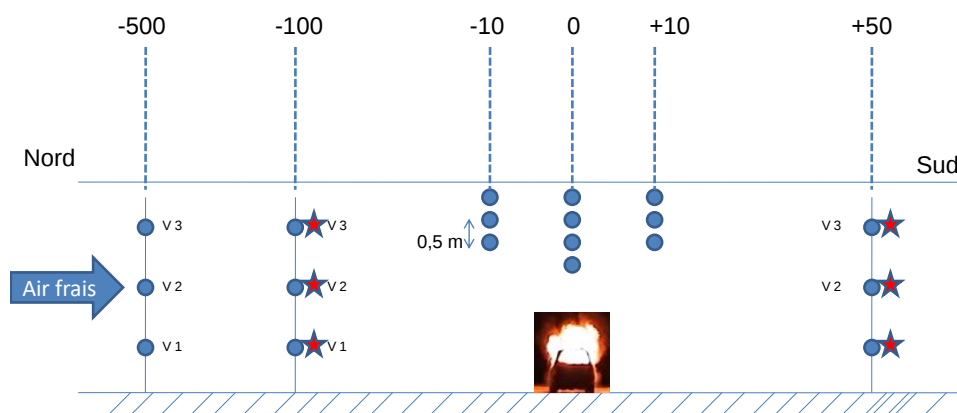


Fig. 3 Position of sensors: thermocouples (circles) and Mac Caffrey probes (stars).

3.3 Metrology

Several sensors were distributed along the tunnel to measure temperature, velocity and carbon dioxide concentration.

First of all, velocity (Mac Caffrey probes) and temperature were measured along the cross-section on each side of the fire. This enables to measure the convective power produced by the fire. This quantity is of course crucial for comparison between numerical modelling and experimental measurements. Then thermocouples were positioned in the fire area to follow the temperature evolution all along the fire near the tunnel roof (Fig. 3).

On top of those measurements, gas analysers were distributed along the tunnel with, mainly, one located 100 m downstream the fire.

4. Main Experimental Results

It first must be noticed that the wind was from North to South when fire test was made. The velocity of the air flow in the tunnel before the fire ignition was around 2.5 m/s, this means a natural air flow rate of 102.5 m³/s.

4.1 Evaluation of the Power Release

The fire curve developed by the ignited car was computed from the experimental data based on the energy balance between fresh air and smoke. This consists in applying the first principle of thermodynamics on an open system.

Considering ϕ is an energy flux, the conservation of energy is then written:

$$\phi_{s1} + \phi_{s2} + \phi_{\chi} = 0$$

This means that outflow energy flux, ϕ_{s2} , equals the sum of entrance energy, ϕ_{s1} , and the energy produced by the fire, ϕ_{χ} . The power release by the fire was obtained considering both chemical, kinetic and sensible energy (Fig. 4). This yields the convective heat release rate. The total heat release rate was then obtained assuming a radiative fraction of 0.3 [3].

This curve shows a slow inflammation phase during the first minutes after ignition, then a quick increase of heat release rate due to the inflammation of all the combustible materials inside the car and finally a quite steady state before extinction. The maximum heat release rate reached by this fire is around 3 MW, 10 min after ignition.

This curve also shows the ability of the analytical model to predict the car fire curve including both the maximum heat release and the combustion duration. Both those values are over estimated using the standard fire curves. One improvement in this model should be the consideration of the incubation time to improve the fire growth prediction.

4.2 Temperature Distribution

Because of the low power developed by the fire in relation with the air flow, no backlayering was observed, the maximum temperature 10 m upstream the

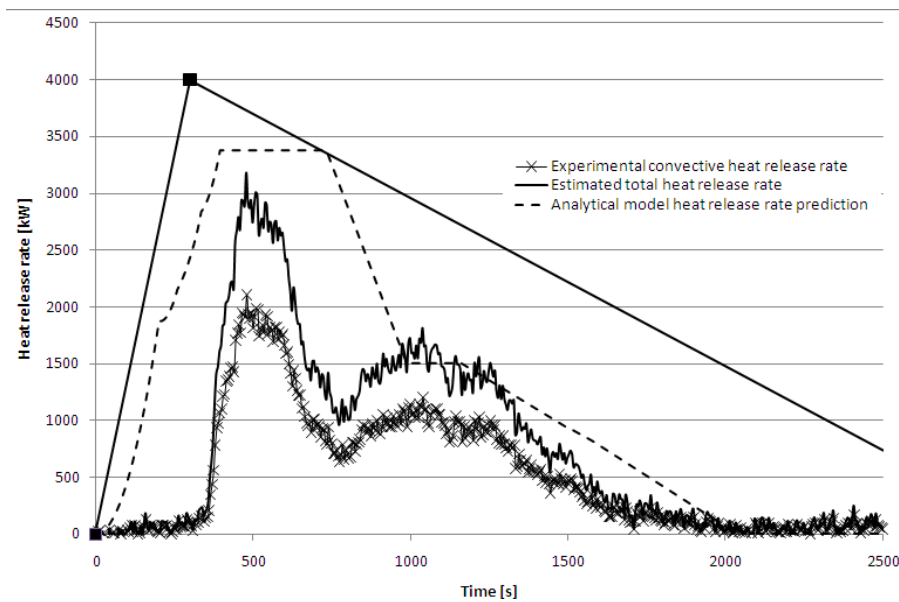


Fig. 4 Fire curve determined using energy balance.

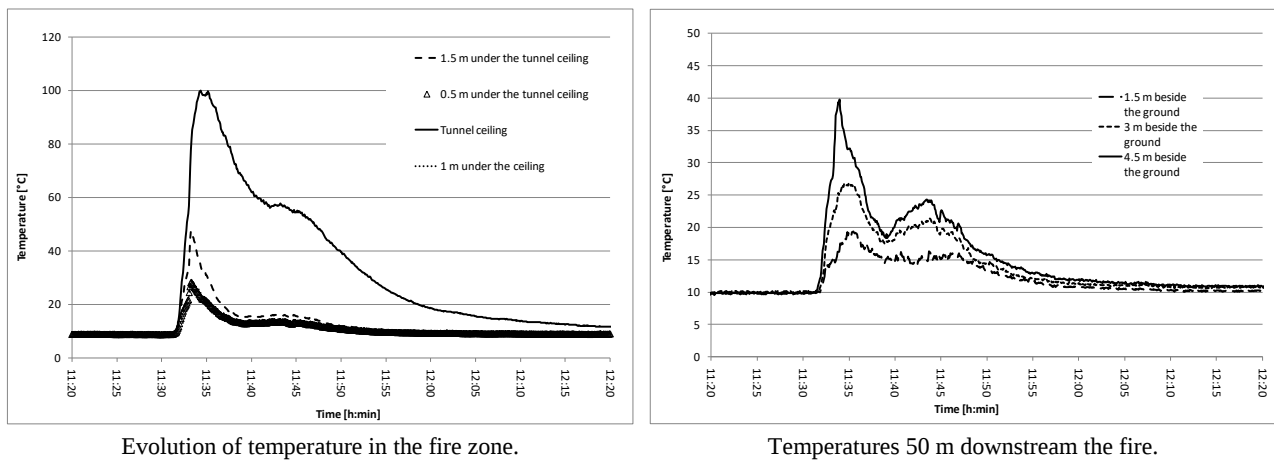


Fig. 5 Experimentally measured temperatures on fire zone and 50 m downstream the fire.

fire on tunnel ceiling was not modified from ambient, and fire influenced temperatures are located in the fire area and downstream the fire. The maximum temperature reached under ceiling in the fire zone is about 100 °C. This is also the case for temperatures measured 50 m downstream the fire (Fig. 5).

4.3 Visibility

Visibility measures inside the tunnel were done using a video camera and a calibrated source. One of the main results is the quick diminution of this visibility and the important smoke cloud generated by the fire.

5. Comparison between Numerical Modelling and Experiments

5.1 Requirements for Modelling

Following constraints mentioned in the FDS specific description, the mesh was built putting a minimum of 20 cells to capture the integral scale. For the validation part, a 1,300 m long part of the tunnel was modelled including the entire tunnel downstream the fire and around 1,000 m upstream to ensure an inlet independent flow. The total number of cells used for that case is 1,900,800 cells with a characteristic size of 20 cm in the fire zone.

The fire was modelled using the fire curve measured during the test considering a ramp based on the measurement. The fire was started 1,200 s, 2.5 convective times, after the beginning of the simulation to let time for the flow to be established. To take into account the wind velocity fluctuations that occur outside and of course generate turbulence inside the tunnel, a sinusoidal signal was used [9]. This enables to introduce some turbulence inside the domain as in the real configuration. These fluctuations were constructed on the basis of velocity measurement inside the tunnel.

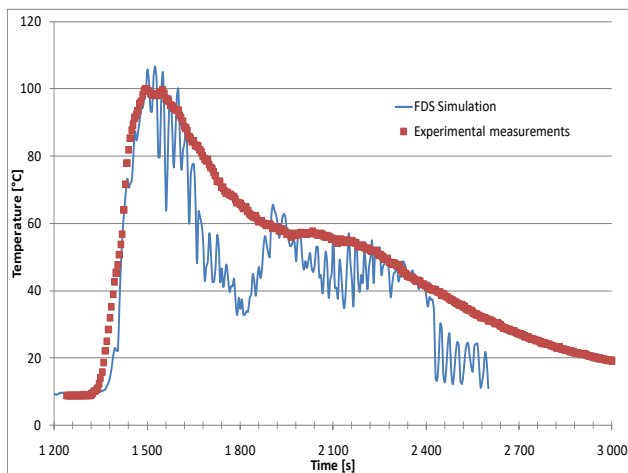
Finally, the tunnel walls were considered as rock with a thickness of 2 m with a conductivity of 3.5 W/m/K, a density of 2,600 kg/m³ and a specific heat of

1,000 kJ/kg/K. In the fire region, the product used for the tunnel thermal protection was modelled with its real characteristics including the evolution of the conductivity with temperature.

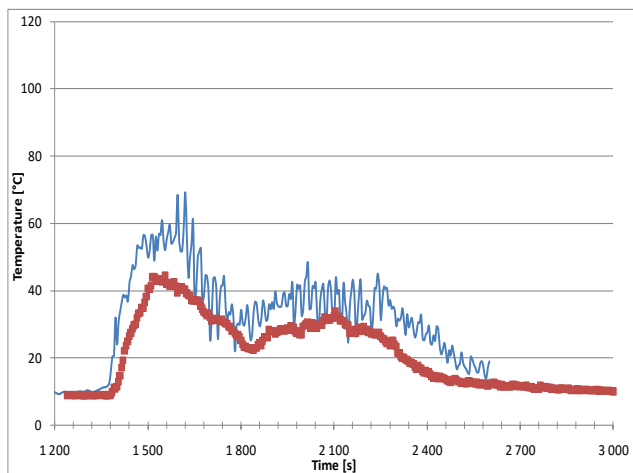
5.2 Comparisons

Numerically computed temperatures were first compared with experiment and show a good agreement, (Fig. 6). The fluctuations on the numerical curves are the consequences of the LES turbulence model used.

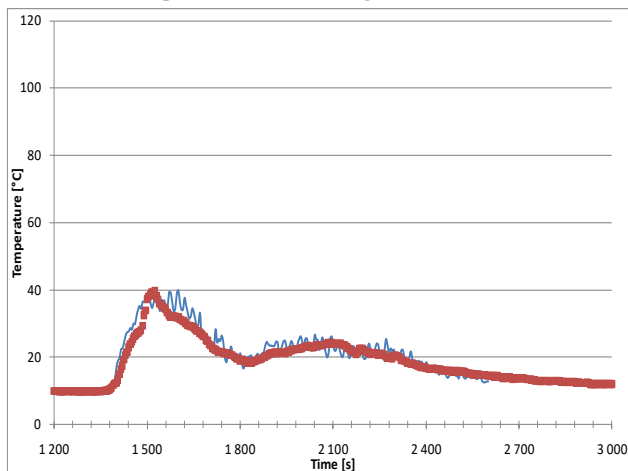
Carbon monoxide concentrations 100 m downstream the fire and 1.5 m above ground were also compared between experiment and numerical simulation and confirm the ability of the CFD code to give a quite good overview of the fire consequences (Fig. 7).



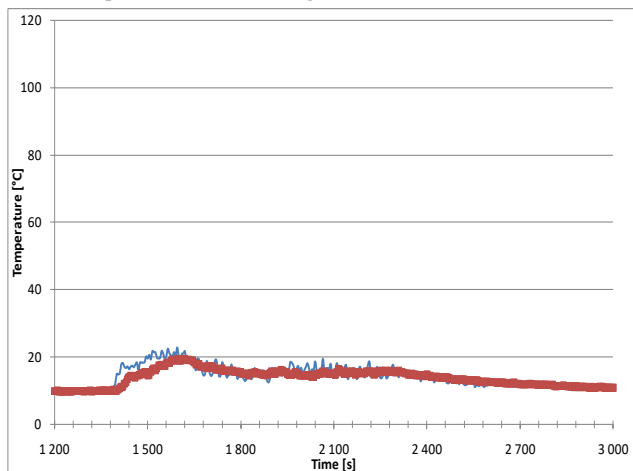
Temperature under ceiling beside the fire.



Temperature under ceiling, 10 m downstream the fire.



Temperature 4.5 m above ground, 50 m downstream the fire.



Temperature 1.5 m above ground, 50 m downstream the fire.

Fig. 1 Temperature comparisons, legend is identical for all graphs.

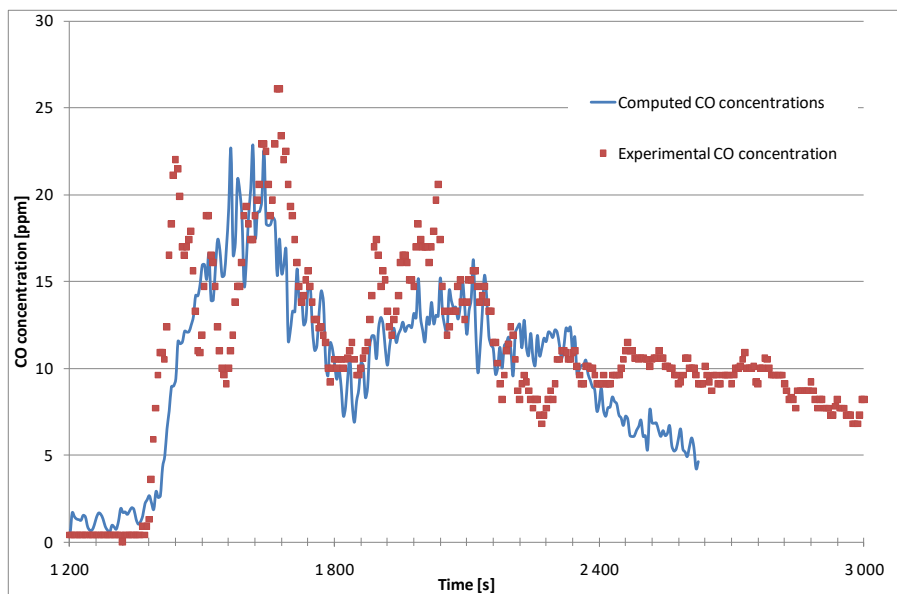


Fig. 2 CO concentration comparisons.

These comparisons show the good agreement that can be obtained between experiments and simulations on some important parameters: the energy exchanges between smoke and walls and correct turbulence introduction in the numerical domain. This first simulation case was used to validate the numerical approach and the use of CFD to reproduce other configurations such as more powerful fires or other wind directions considering that all cases cannot be explored experimentally.

6. Conclusions

The aim of this paper was to present the experimental results and comparisons between experiments and numerical simulation. This paper shows that the analytical model developed for predicting the heat release by a fire of vehicle is able to predict the fire curve with quite a good precision including mainly the maximum value for heat release rate and the combustion duration. This is useful considering CFD codes encountered difficulties in such a prediction.

This paper also showed that, when using correct hypothesis, a CFD code is able to give good predictions of temperature, toxic gas concentration and visibility, i.e. all fire consequences, in the tunnel. The different simulations achieved in this study let appear 2 major

elements that have to be accounted for to obtain quite a good prediction. The first is the turbulence in the entrance of the domain, if this is quite simple with a RANS approach, using LES introduces a difficulty and the velocity fluctuation must be considered. The second crucial point is the thermal effect of walls. The exchange between cold walls and hot smokes is fundamental for smoke behaviour modelling and mainly stratification that is based on the thermal gradient. In the present case, the various wall properties were considered.

References

- [1] Okamoto, K., Watanabe, N., Hagimoto, Y., Chigira, T., Masano, R., Miura, H., Ochiai, S., Satoh, H., Tamura, Y., Hayano, K., Maeda, Y., and Suzuki, J. 2009. "Burning Behavior of Sedan Passenger Cars." *Fire Safety Journal* 44: 301-10.
- [2] Lönnemark, A., and Blomqvist, P. 2006. "Emissions from an Automobile Fire." *Chemosphere* 62: 1043-56.
- [3] CETU. 2003. *Guide to Road Tunnels Safety Documentation, Booklet 4, Specific Hazard Investigations*. LOMÉ: CETU.
- [4] McGrattan, K., Klein, B., Hostikka, S., and Floyd, J. 2008. *Fire Dynamics Simulator (Version 5) User's Guide*. Gaithersburg: National Institute of Standards and Technology.
- [5] Sanchez, J. G. 2009. "Fundamentals for the Development and Application of a Design Train Fire Scenario." In *Proceedings of the 13th Int. Symposium on Aerodynamics*

- and Ventilation of Vehicle Tunnels*, New Brunswick. ISBN: 978-1-85598-107-2.
- [6] Truchot, B., Boehm, M., and Waymel, F. 2009. "Numerical Analysis of Smoke Layer Stability." In *Proceedings of the 13th Int. Symposium on Aerodynamics and Ventilation of Vehicle Tunnels*, New Brunswick. ISBN: 978-1-85598-107-2.
- [7] Hwang, C. C., and Edwards, J. C. 2005. "The Critical Ventilation Velocity in Tunnel Fires—A Computer Simulation." *Fire Safety Journal* 40: 213-44.
- [8] Boehm, M., Truchot, B., Fournier, L., and Biollay, H. 2010. "Smoke Stratification Stability: Results of Experiments, Tunnel Safety and Ventilation." In *Proceeding of the 4th Symposium Tunnel safety and Ventilation*, Graz.
- [9] Mouilleau, Y., and Champassith, A. 2009. "CFD Simulations of Atmospheric Gas Dispersion Using Fire Dynamics Simulator (FDS)." *Journal of Loss Prevention in the Process Industries* 22: 316-23.