Numerical analysis of smoke layer stability

B Truchot [(1)], M Boehm [(2)(3)], F Waymel [(2)]
[(1)] Ineris, France
[(2)] Egis Tunnel, France
[(3)] LMFA Lyon University, France

ABSTRACT

The EGSISTES project is a global reflection about risk and dangerous phenomena relative to underground infrastructures. One category of risk identified for such an infrastructure is the fire and its consequences in terms of temperature and smoke propagation. In some situations, smoke stratification is used to ensure safety of people located inside the tunnel. In such a case, it must be ensure that smoke stay stratified even in the case of an aerodynamic perturbation such as a jet fan or vehicles presence.

Two ways enable the improvement of the understanding of smoke behaviour in underground infrastructure: experiments and numerical approach. Both strategies are used complementary during the project. Experiments are achieved in the INERIS fire gallery while two CFD codes, FDS and Phoenics, based on two different approaches for turbulence modelling, are used.

The first step consists in a comparison between experimental and numerical results on a configuration given as a reference. This reference case was chosen as the backlayering smoke layer establishment and stability. The numerical objective was to reproduce the length and thickness of this layer.

After having shown that both codes should predict with a quite good accuracy the backlayering length, those two codes are used to study the influence of perturbation on the stratification stability.

This study shows firstly that a jet located upstream the backlayering smoke layer tends to modify the smoke layer front but influences slightly the smoke layer near the fire.

Secondly, in case of the presence of vehicles downstream the fire in a congested tunnel, the stratification is not altered just above vehicles but can be altered downstream these obstacles.

TABLE OF NOTATIONS

Symbol	Meaning
Ri	Richardson number
ϕ	Physical variable
$\underline{\phi}$	Vector variable
V	Velocity
H	Enthalpy
h	Height of the tunnel
k	Turbulent energy
ε	Turbulent energy dissipation rate
ϕ_v	Volume flow rate
ϕ_m	Mass flow rate
ρ	Density
Γ_ϕ	Generalised diffusion coefficient
S_ϕ	Source term

1. INTRODUCTION

This study is a part of the EGSISTES research project that aims to improve the understanding of dangerous phenomena in underground infrastructures. For the present case, it deals with fire in tunnels.

Recent large fires in tunnels have induced a modification of the regulation relative to the driver's protection. Mainly, smoke management in case of fire was define on the base of the smoke stratification for bidirectional tunnels and for congested unidirectional ones. For both cases, smoke generated by the fire has to be pushed out of the tunnel without invading the whole section because of people presence. As an illustration of this strategy, the French regulation (1) imposes for congested unidirectional tunnel, a maximum velocity of 2 m/s in order to keep the stratification during the evacuation procedure. Such a velocity will generated backlayering flow that must also keep stratified.

As that appears clearly in this above description, stratification must be strictly ensured during the evacuation period, not only for the downstream flow but also for the backlayering layer too. It must however be kept in mind that the stratification of the flow is based on an equilibrium between buoyancy effect and sheer stress.

In above described tunnels, two main disturbance sources were identified: jet fans located upstream the fire, which are used to control the air flow velocity, and turbulence induced by vehicles.

The aim of this paper is to study the influence of those two kinds of perturbations on the stratification.

The whole study, in the context of the EGSISTES project is based on both experimental and numerical works.

The experiments, achieved in the INERIS fire gallery are used to improve physics understanding. Moreover, numerical approach enables to go further and reveals some details that can hardly be enlighten by the experiment.

The aim of this paper is mainly to describe the numerical work based on two Computational Fluid Dynamics codes: the Fire Dynamics Simulator (FDS) and Phoenics.

First of all, the numerical approach is briefly described comparatively for the two codes. Then, the experimental geometry, which was retained as the numerical one, is described.

Next, before describing comparatively the results of those two codes, a comparison with available experimental result is achieved. This comparison concerns the backlayering flow and its properties.

At the end, the two disturbance source effects were tracked. Results obtained with both codes are detailed and the influence of disturbance on the stratification phenomena is discussed.

2. NUMERICAL APPROACH

2.1 Conservation equation

The smoke behavior in tunnel in case of fire is governed by the fluid mechanic equations. These equations are the mass, momentum, and energy conservation that can be written in the conservative general form as on [1].

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho V \phi) = \nabla \cdot (\Gamma_\phi \nabla \phi) + S_\phi \qquad [1]$$

These equations cannot be solved analytically and require a numerical approach. The complexity of the geometric real domain leads to introduce in this numerical approach, some physical models.

In fire modeling, main physical phenomena that have to be modeled are turbulence and fire heat release.

2.2 Numerical solvers

PHOENICS is based on the finite volume method which solves the equations on a staggered grid. The solutions of the resulting coupled differential equations are obtained in an iterative manner using the SIMPLEST algorithm (2).

FDS is based on a second order finite difference scheme for the spatial discretization. The temporal resolution uses predictive corrective scheme (3).

2.3 Turbulence modelling

The two codes used in this paper are based on two different turbulence models.

To account for turbulence in Phoenics code, the Chen-Kim (4) modified RANS k-ε model (5) (6) is used. In this model, turbulence is represented by the turbulent kinetic energy k and its dissipation rate ε. Both of these variables require solving two equations of the generic form presented in [1].The effects of gravity on the turbulence are modelled by the introduction of buoyancy source terms into the equations for k and ε.

In FDS, the turbulence closure is based on the LES Smagorinsky static approach (7). In such a model, the largest scales are directly solved while phenomena smaller than the cut scale are modeled.

Before going any further in the code results comparison, it must be notice that LES provide instantaneous values while RANS results gives temporal, which can be considered as statistical, means quantities. So, it is important, for the results comparisons, to average LES results along the steady phase. This is done only for comparison purposes, underlying the fact that providing instantaneous results, in the case of an LES model enables to capture some details that cannot be trapped using a RANS approach. Instantaneous temperature pick, for example, will not be enlightened by the averaged method while LES will be able to show it. Secondly, characteristics frequencies of the flow, such as vortex formation, can also be revealing by the LES approach.

2.4 Fire Representation

Physically, a fire could be modeled as a series of complex chemical reactions that induced species transformation and the release of energy, only the thermal effect is taken into account in this study, species production is not discussed. It can be numerically represented in many different ways.

A classical approach to represent fire in tunnel (2) consists in imposing a source term in the energy equation associated with a kinetic evolution. This approach was, for example, retained by the French ministry of transport (8). The species source terms is defined proportionally to the heat source term.

As discussed above, the way to represent the fire is to define a source term in the energy equation to represent the heat release rate of the fire.
The energy equation that is solved is given on [2];

$$\frac{\partial \rho H}{\partial t} + \nabla \cdot (\rho V H) = \nabla \cdot (\Gamma_H \nabla H) + S_H \qquad [2]$$

In Phoenics, the energy source term S_H is the convective heat release rate of the fire per unit of volume as the fire is modeled by a volumetric source of heat; the radiation part is not modeled. In simulations presented in this paper, the heat release rate density retained is 1 MW/m^3, which is the value advocate for this code.

In FDS the fire is represented by a surface of emitted combustible products. The heat released is next distributed in different cells using a simple combustion model (3). This method enables to represent exactly the experimental fire source. Radiant heat transfer is modeled in FDS using a 1-flux approach.

2.5 Boundary conditions

Walls are considered as adiabatic and rough. Standard wall-functions are used with the Phoenics *k-ε* model. A 3 mm roughness scale is imposed, this corresponds to the concret roughness. In the LES approach, using FDS, there is no particular model for the wall. The smoke layers are not as long as the roughness should have an impact on the results. Furthermore, in the experimental part, the higher probe is not installed in the boundary layer.

At the inlet, the atmospheric pressure and the ambient temperature are imposed. This boundary condition allows the smoke to go out if the backlayering predicted length is too important. The fan is represented by a flow rate imposed on the chimney base.

3. EXPERIMENTAL DOMAIN

This experimental study is carrying out using the INERIS fire gallery: a 1/3 scale tunnel with a maximum clearance cross section of 5.4 m^2 and with a total length of 50 m.

The experimental set up is schemed on Figure 1. The upper part is a smoke duct which permits to ventilate using a transversal ventilation system, system that is not used in the present paper.

The ground is not regular (approx. ±1%) that was modeled in the different simulations.

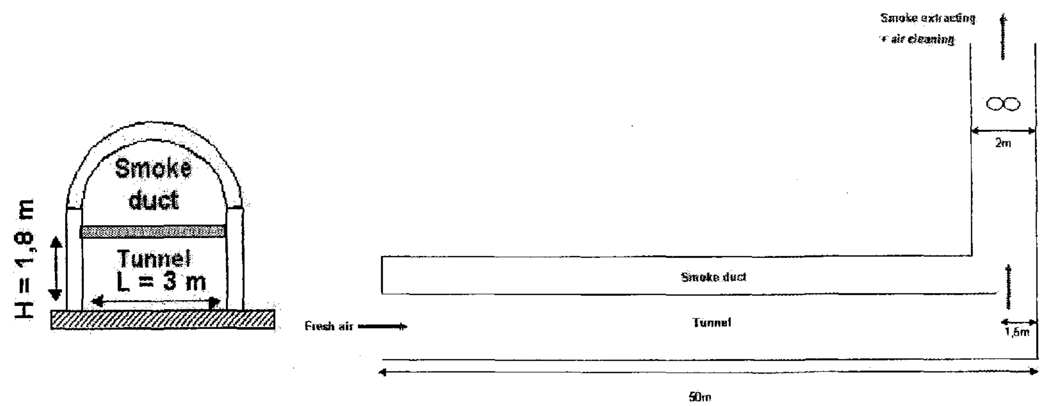

Figure 1: Scheme of the experimental set-up.

All parameters of this study are calculated using Froude similarity to take in account of the reduced scale.

During experiments, the fire is generated by an heptane pool fire. The experimental heat release rate of the fire is computed from the mass consumption measurement and reaches 352kW in reduced scale, this corresponds to a heat release rate of 5,5 MW in full scale. The radiative fraction was also computed on the basis of the experimental result. To do that, velocity and temperature profiles were measured downstream the fire, this gives the convected part. Knowing both total and convective energy, the radiative fraction was evaluate to 0.39, value in accordance with standard value for 0,25 m² heptane pool fire (9).

The ventilation is supplied by the fan installed in the chimney. It is important to note that the fan is far from the fire and downwind an air washer. This means that the temperature at the fan is constant during one experiment. The fresh air would arrived by the open extremity on the left on the scheme.

4. NUMERICAL REPRESENTATION OF THE DOMAIN

Because of numerical constraints, the geometrical model is based on rectangular cross sections. In such a model, the lower part of the fire gallery is correctly represented because its section is a square of 3 m width by 1.8 m height.

After having checked that the chimney does not affect simulations and in order to reduce the cells number, i.e. the CPU time for each computation, this chimney was not modeled. Then, a study of the influence of the grid sensibility was achieved and the independence of the results was checked, focused on the backlayering length. Following this study, for both FDS and Phoenics, the same number of cells was used: 250 cells in the length, 60 in the width and 40 in the height, which gives a total number of 600,000 cells. This gives characteristic size for the cell of: 20 cm in the length, 5 cm in the width and 4.5 cm in the height.

Two phenomena were studied, upstream and downstream stratification, so two fire positions were tested:

- To obtain an upstream smoke layer in the gallery (backlayering), the fire was placed at 10 m of the chimney;
- To evaluate the downstream layer stratification, the fire was located 10 m from the inlet.

At first, the stratification of both upstream and downstream smoke layers are characterized.

Then, the influence of the perturbation on the stratification is analyzed. For the backlayering, the perturbation corresponds to a jet fan. For the downstream smoke layer, the perturbation is due to the presence of vehicles blocked by the traffic jam downstream the fire. The vehicles are represented by cube to avoid cutting cells. The vehicles position studied represents the traffic of an unidirectional tunnel.

5. STRATIFICATION QUANTIFICATION

To discuss about stratification, Newman (13) has introduced two stratification parameters:

$$\begin{cases} S_1 = \dfrac{T_c - T_f}{\Delta T_{avg}} \\ S_2 = \dfrac{T_c - T_f}{T_h - T_0} \end{cases} \qquad [4]$$

Where T_c is the temperature at the height of 0.88h, T_f the temperature at 0.12h and T_h is the temperature in the smoke layer. ΔT_{avg} is the gas temperature rise relative to ambient condition and is defined in (13).

These two parameters inform on the amplitude of the temperature differences, and on the relative thickness of the hot and cold layers.

The study of the stratification using those two parameters can be done for both upstream and downstream smoke layers.

For the downstream study, the function between those two parameters defines two regions that depend on the stratification characteristics (14).

In the region II (Figure 2), for $S_1 \leq 1.7$, buoyancy dominated the temperature stratification. This leads to a destruction of stratification. If S_2 approaches values of the order of 0.1, the smoke layer is completely destratified.

In the region I, for $S_1 > 1.7$, S2 is roughly equal to 1. This implies that the gas at the floor is at the fresh air temperature.

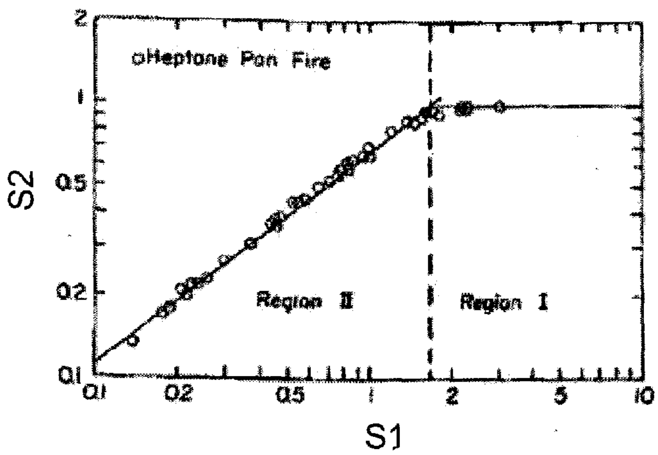

Figure 2: Evolution of the two stratification parameters, reproduced from (14).

For the upstream flow, the evolution of the function is supposed to be uniform following a power low (15).

6. COMPARISON BETWEEN NUMERICAL AND EXPERIMENTAL RESULTS FOR THE BACKLAYERING

Before going any further in the physical and numerical analysis, it is important to ensure that the numerical results are in good agreement with experimental values.

This comparison is based on the experimental measurement of the backlayering. The experiment use a fire located at 10 m of the base of the chimney and the ventilation flow rate was chosen to respect French regulation for a longitudinally ventilated congested tunnel (1). The air flow velocity at reduced scale is 0.93 m/s, this corresponds to one of 1.6 m/s in full scale.

6.1 Experimental and numerical results comparison

Representative quantities for the backlayering phenomena are temperature and velocity. First of all, maximum temperature reached under the roof above the fire was compared because the backlayering effect directly depends on this parameter.

The experimental value of 285°C was quite correctly reproduced by the two CFD codes (245°C for FDS and 240°C for Phoenics).

Two temperature profiles were measured: 10 m and 17 m upstream the fire and one velocity profile was measured 17 m upstream the fire. Those three profiles were compared with the numerical prediction. Figure 3 shows the temperature profile comparison between both FDS and Phoenics prediction and the experimental one.

This graph enables to compare the thickness of the thermal layer. The interface between the hot smoke and the fresh air layers height is calculated by the N-rule method (12). These interface heights are represented on the figure by stars. It is shown on this figure that the thickness of the backlayering layer is over predicted by the two CFD codes.

It is also shown on this figure that the temperature is under predicted with FDS in the upper layer. Whereas, the maximum temperature of the backlayering layer is well predict by Phoenics. Nevertheless, the temperature of the upper part of the smoke layer is not well predicted by Phoenics. This can be partially explained by the ceiling model in Phoenics simulations that do not take in account the heat transfer between the ceiling and the smoke layer.

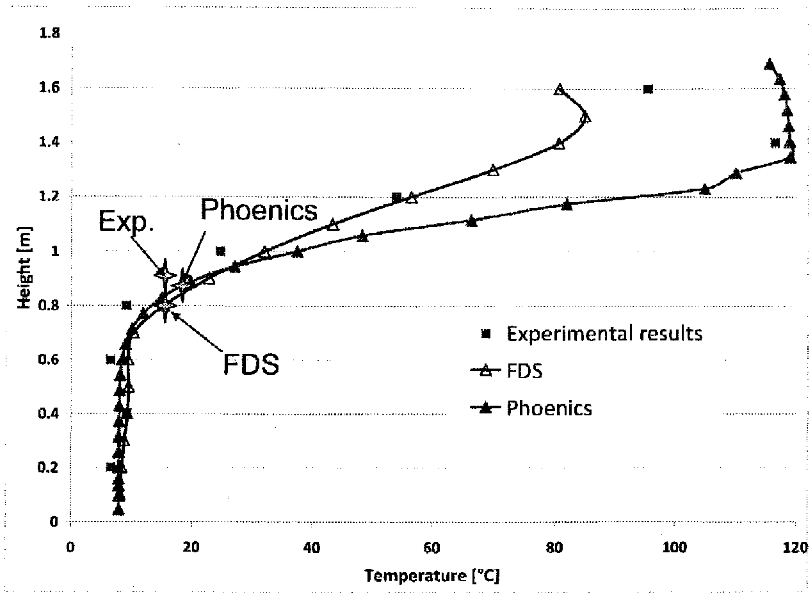

Figure 3: Temperature profile comparison at 10 m upstream the fire.

Next, as discussed earlier, an experimental profile of velocity is available 17 m upstream the fire, Figure 4 shows the comparison between numerical predictions and experimental measurements for the temperature and the velocity profiles.

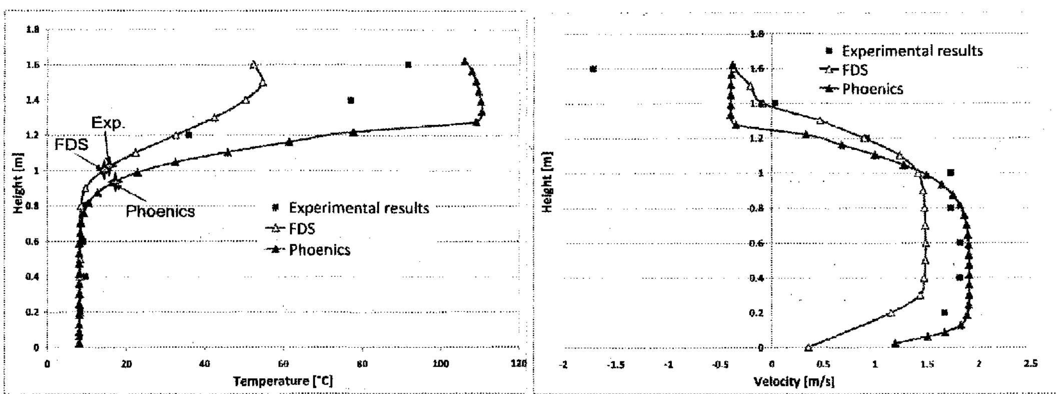

Figure 4: Temperature and velocity profiles comparison at 17 m upstream the fire.

The interface heights on the temperature profile are represented by stars.

This figure shows that the layer thickness is well predicted by FDS farther from the fire while for Phoenics the thickness is over predicted. This figure confirms the under prediction of the temperature of the layer by FDS.

The temperature of the backlayering layer is over predicted by Phoenics, this could be explained by the over prediction of the upper part of the smoke layer nearer the fire.

Before concluding, the measured length of the backlayering layer is about 25 m, which is in great accordance with the correlation given by (12).

The predicted lengths using the two CFD codes are compared to this experimental result. The two results are reproduced on Figure 5. FDS simulation predicts a backlayering length of 21 m upstream the fire and Phoenics a length of 26 m, lengths in good agreement with the experimental measurements.

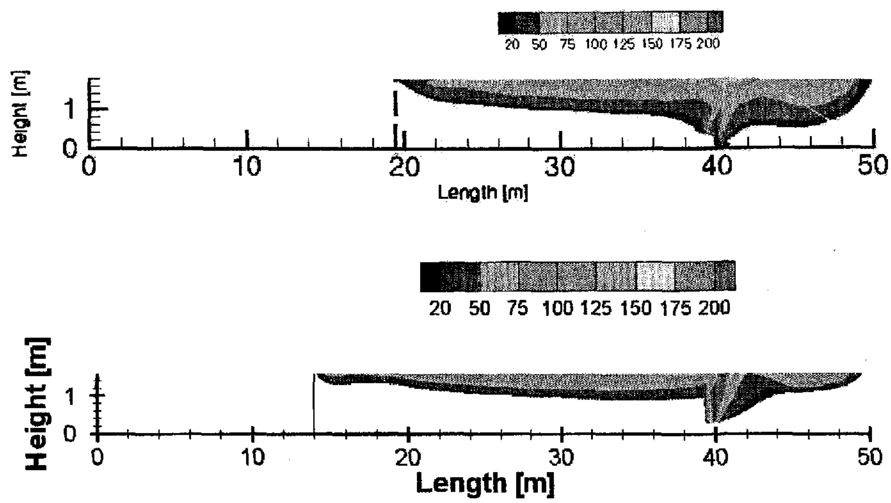

Figure 5: Visualization of the backlayering length obtained with FDS (up) and Phoenics (down).

6.2 Conclusion of the experimental confrontation

This first comparison permits to conclude that the *k-ε* Chen-Kim model is probably inappropriate to predict the smoke layer thickness whereas the LES model permit to obtain a result in better accordance with the experimental measurements.

The smoke temperatures obtained with Phoenics are over predicted. Whereas, FDS under predicts the hot smoke temperature. In FDS, the radiant heat transfer between hot smoke layer and the ceiling is probably over predicted. In Phoenics, the fact that the heat exchanges at the ceiling (adiabatic boundary condition) are not modeled can explain the over prediction of hot layer temperatures. Whereas, the sides model would not affect the result as the smoke layer has negligible heat exchange with the sides. In fact, the contact surface between the smoke layer and the sides is small. Moreover, the sides are made of refractory concrete.

For next experiments, it will be important to refine the temperature profile in the smoke layer and to determine the evolution of the ceiling temperature during experience to check the level of heat exchange between the hot layer and the ceiling.

Next sections are based only on numerical simulations. Experimental results for following configurations are not yet available and will be published later.

7. COMPARISON OF THE TWO CODE RESULTS FOR THE PERTURBATION INDUCED BY A JET FAN

In longitudinally ventilated tunnels, jet fans, located under the ceiling, are used to generate the air flow. The momentum source generated by such an apparatus can have an influence on the stratified backlayering layer.

The aim of this part of the study is to characterize this influence. To prevent for complex coupled physical effects, the jet was designed to prevent from rotational effect in the experimental situation.

The same approach, i.e. a plane jet without rotational effect, was used in the numerical approach. This simplified approach enables to represent the vertical gradient of velocity that propagates into the tunnel to the backlayering layer.

The jet will be generated at the entrance of the fire gallery using a ventilation box. The output velocity of the ventilation box was set to 3 m/s.

The characteristics of the backlayering layer are then compared with and without jet in terms of temperature distribution, velocity profile and backlayering length. In order to compare the hot smoke layer characteristics, it is necessary to conserve an air flow velocity of 0.93m/s for the cases with and without jet.

First, the vertical temperature distribution is plotted on Figure 6 for numerical results obtained with FDS and Phoenics.

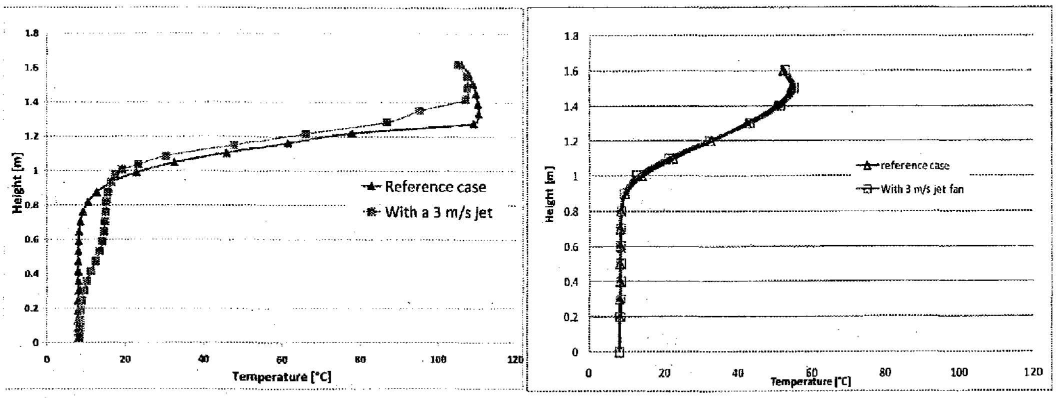

Figure 6: Vertical profile of temperature with and without jet fans at 17 m upstream of the fire obtained with Phoenics (left) and FDS (right).

These results show that the backlayering layer is slightly affected by the jet near the fire.

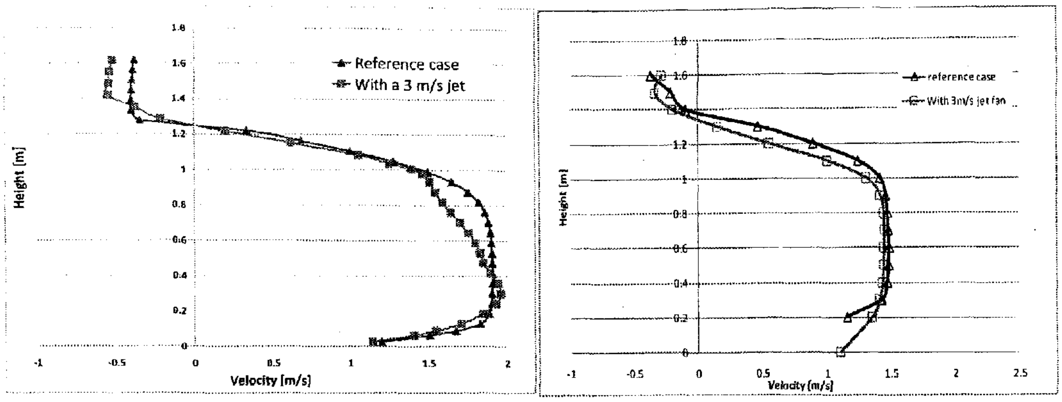

Figure 7: Velocity profile 17 m upstream the fire source obtained with Phoenics (left) and FDS (right).

The jet influences greatly the part of the smoke layer near the front. The comparison of the Figure 5 and Figure 8 shows the modification of the backlayering front due to the jet.

The hot smoke layer in this part is thicker with the jet than without. This implies that the backlayering layer is longer with the jet than without.

Moreover, the front layer temperature distribution is perturbed by the jet. The smoke with temperature between 20 and 50°C takes a thicker layer on the front with the jet than without.

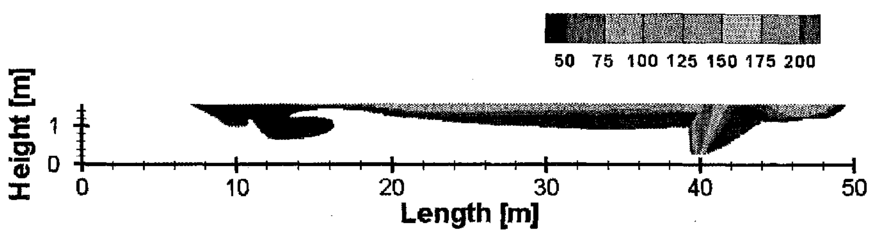

Figure 8: The backlayering layer front obtained with Phoenics

The FDS code predicts a lower influence of the fan on the jet with a small impact on the head of the backlayering layer. This lower influence should be explain by the fact that the length of the layer compute with FDS is smaller than the one obtained with Phoenics and, consequently, the distance between the backlayering front and the fan is more important with FDS, so the effect of the fan is diminished. Moreover, the turbulence models used with Phoenics and FDS are not the same and should affect the influence of jet on backlayering layer.

Experiences will be achieved to explain this influence of the jet located just near the backlayering front.

8. COMPARISON OF THE TWO CODE RESULTS FOR THE PERTURBATION INDUCED BY THE VEHICLE PRESENCE

The two previous sections dealt with the backlayering layer.

The present part objective is to evaluate the stratification of the downstream smoke layer and the influence of vehicles on its stratification. This point is particularly important in case of longitudinally ventilated congested tunnel. In such a case, vehicles can be blocked under the smoke layer and their influence could be important as shown on some previous studies (10) and (16). The aim of this part of the study is to evaluate this importance and the ability of numerical CFD code to predict this influence. Those results will be compared with future experimental work. The numerical simulations are released in the gallery with an air flow velocity of 0.93 m/s.

For convenient reasons, vehicles are based on cubic forms for both the numerical and the experimental approach. The vehicle distribution includes 13 cars and 2 lorries. The vehicles are assumed to be blocked downstream the fire in a congested tunnel. They are distributed as reproduced on Figure 9.

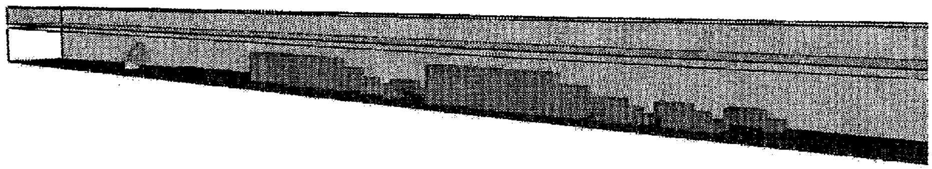

Figure 9: Vehicles distribution in the tunnel.

The comparison between the two configurations, with and without vehicles, is based on the two stratification parameters, S_1 and S_2, defined earlier.

The evolution of $S_2=f(S_1)$ was plotted along the gallery for both codes in the two configurations (Figure 10).

The results show that, according (14), the flow is stratified for both configurations but with two different zones:

- Above vehicles, the stratification is improve by a diminution of the layer height;
- Downstream of the vehicles, the stratification stability is altered because of turbulence generation.

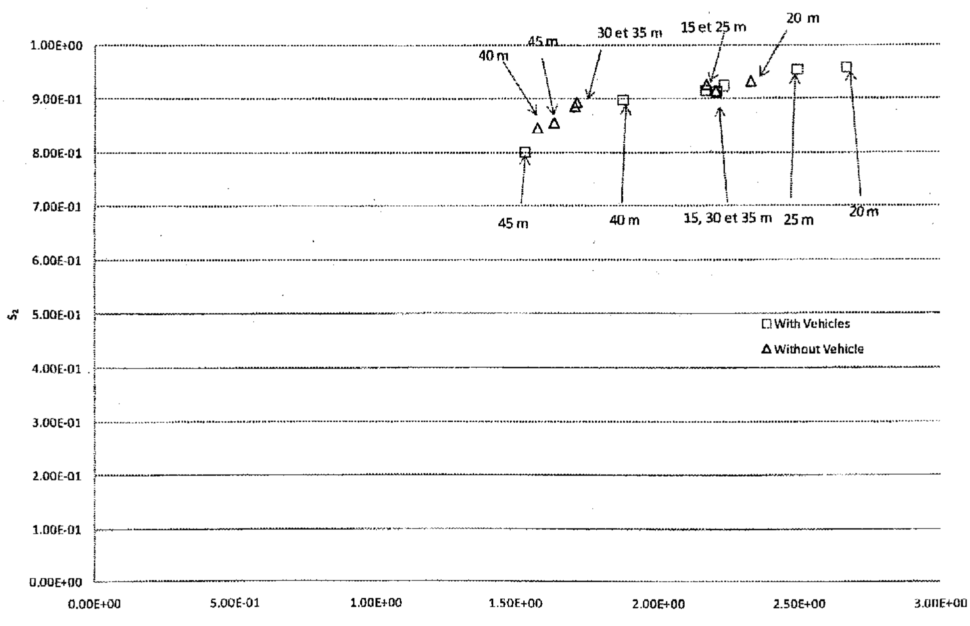

Figure 10: Plot of S2=f(S1).

The temperature profile will give more detailed information about the effect of vehicles on the stratification.

The temperature profile for both codes in the two configurations is plotted hereafter on Figure 11 in the centreline section behind the first lorry.

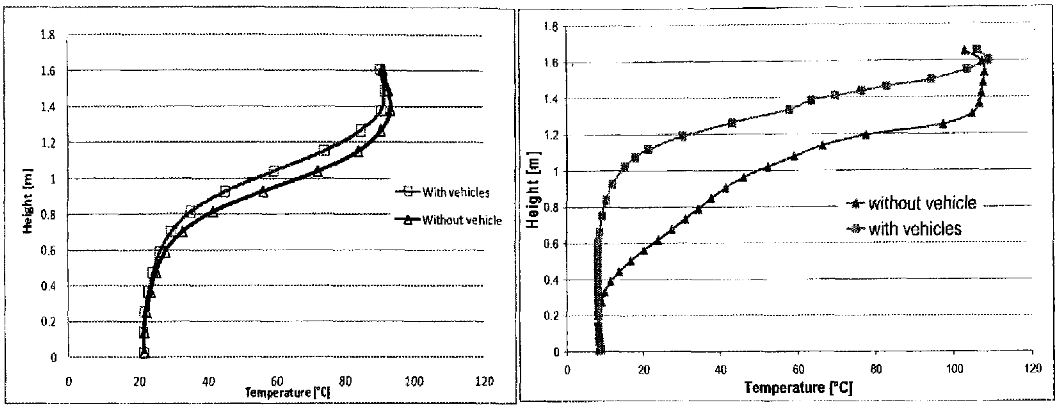

Figure 11: Vertical profile of temperature with and without vehicle obtained with FDS (left) and Phoenics (Right).

Those two figures lead to the same conclusion concerning the stratification evolution in case of vehicle presence in the tunnel. Even if the maximum temperatures reached by the two codes are not equals, each code predicts a reduction of the smoke layer thickness. This diminishing can be explained by the corridor effect on the roof of vehicle and is illustrated by the velocity evolution in presence of vehicle Figure 12.

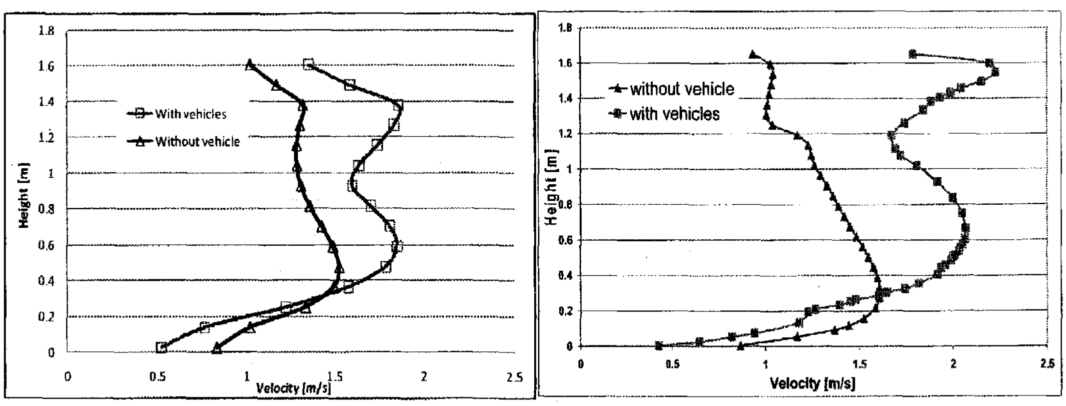

Figure 12: Evolution of the velocity profile in the centerline section, FDS on the left and Phoenics on the right.

The turbulence induced by vehicles, and mainly the recirculation downstream a lorry, was well capture by the two CFD codes Figure 13. The intensity of those structures is not important enough to affect greatly the stratification.

This conclusion is probably also due to the reduce power of the fire (352 kW), in case of larger smoke volume production, this effect may not be observed.

This will be checked by future numerical simulations. Finally, it was observed that the reduction of the smoke layer thickness decrease while increasing the distance from the fire and, consequently, an effect on the stratification stability downstream the vehicles.

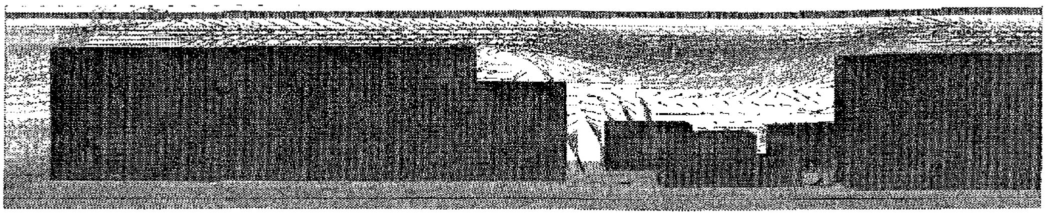

Figure 13: Detail zoom on the recirculation behind a lorry.

9. CONCLUSIONS AND PERSPECTIVES

This article is the beginning of a larger study that aims to improve the understanding of the physical phenomena responsible of the smoke propagation in tunnels. To reach this objective, an experimental campaign is achieved in accordance with a numerical one.

The first part of this study aims to quantify, using two CFD codes, the impact of a perturbation on the stratification phenomena. This also enables to compare the ability of the two codes to predict smoke behaviour in the context of a fire in tunnel.

After having briefly described the numerical models and the geometry used for the experimental campaign, the Newman parameters that can be used for describing the stratification are presented (14). Those parameters are used to evaluate the influence of obstacles on the smoke layer behaviour and stability.

Before going any further, the confrontation between experiment and numerical modelling have shown that both codes are able to predict with a quite good accuracy the length of the backlayering smoke layer. However, the two CFD codes with the model used in the simulations do not permit to represent with good accuracy the temperature distribution in the smoke layer. More experiences are required to determine more accurately the temperature smoke layer profile and the evolution of the temperature on ceiling.

Finally, the influence of two types of external perturbation was numerically studied.

The first kind of perturbation is an air flow coming, in real situation; from jet fans located upstream the fire. To prevent for complex interaction and enable to understand the influence of a velocity gradient coming from such a fan, the jet was generated in order to remove all rotational effects. The numerical approach has shown that a plane jet has no effect on the backlayering layer near the fire. Nevertheless, the jet influences the smoke layer front: the hot layer is thicker than the one without jet and the front starts to destratify. Thus, the backlayering layer tends to be longer than the one without jet.

The second type of perturbation consists in vehicles blocked in the traffic jam downstream the fire. The numerical comparison between the reference case, that is to say without vehicle, and the smoke propagation above the vehicle has shown that just above vehicle, the stratification is not altered by cars and lorries but can be altered downstream these vehicles.

As indicate above, this part is the beginning of a larger study. The next step of it is to achieve the experiment which corresponds to the modelled situation. Those experiments will not only enable to validate the two codes but also provide a complete experimental work concerning the smoke propagation in tunnel.

Next, the numerical study will be extended to real size tunnels. Compiling all the results will permit to have a great improvement of the understanding of the physical phenomena involved in the smoke propagation in tunnels and then, to propose a simple model to reproduce this phenomena.

ACKNOWLEDGMENT

The authors thanks the ANR, "Agence Nationale pour la Recherche" for funding the projects and all the other partners of the projects: CEA, LME and RATP for their involvement in the EGSISTES project.

REFERENCES

(1) Circulaire interministérielle n°2000-63 du 25 aout 2000 relative à la sécurité dans les tunnels routiers du réseau national.
(2) Numerical heat transfer and fluid flow; S. PATANKAR, Hemisphere publishing corporation, 1980.
(3) Fire Dynamics Simulator (Version 5) Technical reference Guide. 20 October 2008.
(4) Computation of turbulent flows using an extended k-e turbulence closure model; Y. CHEN and S. KIM, report NASA-CR-179204, 1987.
(5) Turbulence Models and Their Application in Hydraulics; W. RODI, Journal of Fluid Mechanics Digital Archive (1983), 131 : 471-471
(6) The prediction of laminarization with a two-equation model of turbulence. JONES, W.P. AND LAUNDER, B.E (1972), International Journal of Heat and Mass Transfer, 15: 301-14
(7) General circulation experiments with the Primitive equations I. The basic experiment. Monthly weather review, SMAGORINSKY, 91(3):99-164 March 1963.
(8) Guide des dossiers de sécurité des tunnels routiers, Fascicule 4 : Les études spécifiques de dangers ; CETU, Septembre 2003.
(9) Scale Dependency of Radiation and Smoke Emission from Large Pool Fires, KOSEKI ; Second Symposium on Scale Modelling (Inter.) pp87-96,1997
(10) Etude numérique de l'influence des véhicules sur la déstratification des fumes d'un incendie en tunnel ; F. WAYMEL, L. FOURNIER and E. RUFFIN, Congrès AFTES – Octobre 2005.
(11) An Experimental Study of Upper Hot Layer Stratification in Full-Scale Multiroom Fire Scenarios, L.Y. COOPER, M. HARKLEROAD, J. QUINTIERE, W.RINKINEN, ASME (1981)
(12) Two-layer theory applied to the phenomenon of backlayering-Ministry of Public Worksand Water Management, Center for Tunnel Safety, B. RIGER.
(13) Flame propagation in Ducts; J. NEWMAN and A. TEWARSON, Combustion and Flame 51:347 (1983).
(14) Experimental evaluation of Fire-Induced stratification J. NEWMAN, Combustion and Flame 57:33 (1984).
(15) Characterisation of fire induced smoke stratification in a longitudinally ventilated tunnel, S. GAILLOT, D. BLAY, J-P. VANTELON AND P. DEBERTEIX, BHR Group 2006.
(16) Influence of stationary vehicles on backlayering characteristics of fire plume in a large cross section tunnel, Y. KUNIKANE, N. KAWABATA, S. OKUKAWA, K. OKUBO and A. SHIMODA, Safety Road and Rail tunnels, 2003.