CHT-08: THREE-DIMENSIONAL NUMERICAL MODELLING OF FIRES IN ROAD AND RAILWAY TUNNELS USING THE PHOENICS CODE

Hervé Biollay[*,§], Luc Fournier[*], Mélanie Boehm[*] and Jalil OUAZZANI[**]
[*]Egis Tunnels, PRINGY, France
[**] ArcoFluid, Pessac, France
[§]Correspondence author. Fax: +33 4 50 27 39 40   Email: herve.biollay@egis.fr

ABSTRACT   Recent change of European country regulation about safety conditions in tunnels implies an evaluation of the tunnel global safety level. Such type of evaluation needs knowledge of the smoke movements. This is often done using simple 1D model, but for complex situations where smoke behavior is mainly multidimensional, 3D CFD simulations are required. The current paper presents four model developments specially adapted to such context:

- the first one is the representation of the fire. This has been done in two ways considering the phenomenological aspect of each vehicle (truck or train);
- the second one is an approximate outlet boundary condition to account for density stratification du to the not yet fully developed thermal field. This outlet boundary condition, derived directly from mass conservation, is particularly useful for long tunnel, which cannot be entirely modeled;
- the third one is gravitation force varying in space in order to numerically represent successive tunnel slope;
- the last one is post treatment module which provides an estimation of the radiation due to the hot smoke layer.

The use of these functions will be illustrated by one particularly example: an urban 2x4 lane road tunnel.

INTRODUCTION

This work can be considered as a demonstration of the safety procedure and equipment effectiveness. Tunnel ventilation system is usually considered as one of the main safety equipment. Due to recent evolution of the French regulation concerning the safety in tunnel, for each tunnel longer than 300 m a global safety analysis has to be provided.

The main objectives of ventilation systems are usually:
- providing satisfactory evacuation or auto-evacuation conditions;
- maintaining access for fire fighting.

These two aims are often scheduled at different time. Consequently, the ventilation can be based on a two stage approach. In term of safety, the auto-evacuation phase is in principle the most important one. Its numerical evaluation is often performed with 1D simulations based on pressure loss computation principle.

However, there is a need to characterize smoke stratification as a multidimensional phenomenon which implies to perform 3D simulations. These simulations have to satisfy a full coherency with possible common 1D hypothesis and provide a sufficient level of reliability.

In the current paper we present a set of numerical developments added to the PHOENICS CFD code in order to achieve satisfying safety studies.

We introduce first a general problem definition and then outline three distinct aspects:
- the outlet boundary condition for tunnels ;
- tunnels slope variation ;
- fire representation.

Then we describe a way to represent radiation through post calculations deduced from the CFD results. After discussing the assessments of these developments, we describe and show a real case scenario of a fire in a tunnel.

PROBLEM DEFINITION

In order to do the modeling, we have adopted the finite volume [Patankar 1972] code PHOENICS, [Pratap 1976].

In fire modeling besides the 3D transitory phenomena some important physical phenomena must be taken into account:
- turbulence modeling
- buoyancy phenomena generated by the fire in momentum and turbulence;
- Heat and smoke release at the fire.

In addition, to simulate the flow correctly and thus to be able to estimate the evolution of the smoke as a function of time, it is essential to model the exact geometry as well as all of the infrastructure and obstacles of the work area. The equations solved are the momentum, mass, energy and species conservation equations written in the conservative form as follows:

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot \left(\rho V \phi \right) = \nabla \cdot \left(\Gamma_\phi \nabla \phi \right) + S_\phi \tag{1}$$

where ϕ is the dependent variable considered, ρ, V are respectively the fluid density and the velocity, Γ_ϕ the generalized diffusion coefficient and S_ϕ the source term.

To account for turbulence, we have used the Chen-Kim [Chen 1987] modified k-ε model [Rodi 1984, Launder 1972]. In this model, turbulence is characterized by the turbulence kinetic energy k and its rate of dissipation ε. Both of these variables are governed by differential equations in the form of Eq. (1).

The solutions of the resulting coupled differential equations and their respective boundary conditions are obtained in an iterative manner using the SIMPLEST algorithm on a staggered grid. This algorithm is similar to the SIMPLER or SIMPLEC algorithm [Patankar 1980].

BOUNDARY CONDITION

In the following we give the boundary conditions used in our model for the walls, the inlets and the outlets.

The walls are considered as isotherms with a fixed surface temperature and no-slip boundary conditions. Standard wall-functions relations related to the k-ε Chen-Kim model are used.

At the inlet a given fixed velocity of 1.5m/s with fixed ambient temperature and zero smoke concentration are prescribed

Tunnels are often very long and it is impracticable to model the entire tunnel. Thus, we have to cut the tunnel at a specific location and apply an outlet boundary condition. At first, we can simply represent the outlet boundary condition with an additional area of the tunnel created at the exit and an additional area of atmosphere (figure 1). In this case, the outlet boundary condition is not directly fixed at the tunnel exit [Biollay 1997] [Woodburn 1996]. However, the smoke movement is not correctly determined and extra computational volumes are needed.

If the outlet boundary condition is applied directly on the model limit (figure 2), a numerical problem appears when we have hot smoke exiting the tunnel. For this boundary condition, a special treatment is needed as explained below.
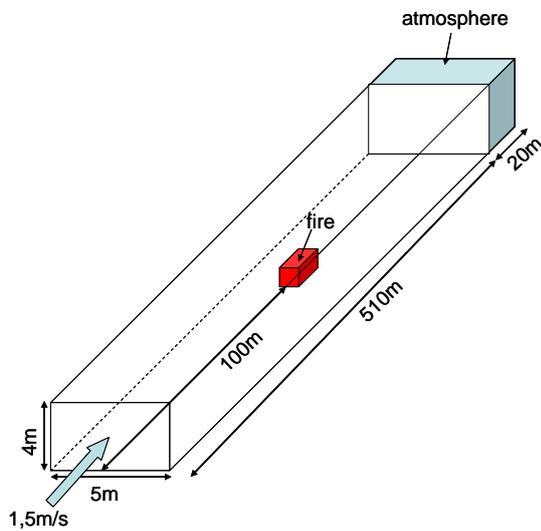

Figure 1: Representation of the tunnel numerical model on case 1
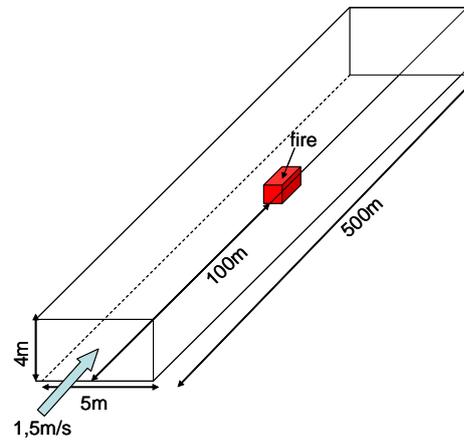
Figure 2: Representation of the tunnel numerical model on case 2

In many fire simulations, the flow will exhibit density and concentration stratification due to the temperature gradients.

A quick overview of the momentum equation shows that a fixed pressure deviation boundary condition when imposed with a constant value will lead to erroneous results.

This can be illustrated through this simple example:
Let's assume that the flow is fully developed and that all axial velocities derivatives are negligible. Assuming moreover that the gravity vector is oriented towards one of the axis, in occurrence the Y-axis, then we obtain:

$$\frac{\partial p_\infty}{\partial y} = -(\rho - \rho_0)g \tag{2}$$

where $P\infty$ is the dynamic pressure at infinity, ρ the density, ρ_0 a reference density at y = 0 and g the gravity amplitude. The integration of eq. 2 gives us the following profile for dynamic pressure:

$$p_\infty = -(\rho - \rho_0)gy \qquad (3)$$

In order to find the pressure boundary condition at the outlet, we will suppose that we have a linear interpolation between the value at infinity eq. 3 and the one at the centre of the last cell [Kobayashi 1993].

We have therefore the pressure boundary condition which is:

$$p_b = f_1 p_P + (1 - f_1)p_\infty \qquad (4)$$

with $f_1 = \left(\dfrac{L_\infty}{L_\infty + \dfrac{\Delta z}{2}} \right)$ and $L\infty$ is an arbitrary long length problem dependent, Δz the depth of the last cell in the axial direction. The subscript P is for values at the centre of the cell and b at the boundary. The length $L\infty$ can be determined through classical Poiseuille fully developed flows.

In Phoenics, the boundary conditions can be imposed through the prescription of a source term in the appropriate equation as:

$$S_\Phi = C_o(Val - \Phi_P)*type \qquad (5)$$

The source term eq. 5 is given in terms of a coefficient "C_o", a value "Val" and a type. In here, the coefficient will be a large value (>to 1.E+00), the value will be the value calculated through eq. 4 and type will be unity.

TUNNEL SLOPE VARIATION

Some tunnels are subject to inner slope variations. Road tunnel slopes are in general varying between -5% to +5%. With such low slopes, it is not advantageous to reflect these changes in the geometry but rather in the gravity vector.

The study of evacuation conditions of a tunnel cannot be achieved without representing the slope. In fact, the evacuation conditions are modified by the smoke movement which is slope dependent.

We assume that the change in slope is such that the geometrical effects are negligible at the singular point of change in first derivatives which is the case in most tunnels.

Let's suppose that the slope is doing an angle alpha with the horizontal, the gravity vector can be decomposed in two components g_y and g_z as:

- in the Y direction parallel to the gravity vector:

$$g_y = g \cos(alpha) \qquad (6)$$

- in the Z direction normal to the gravity vector in the axial direction:

$$g_z = g \sin(alpha). \qquad (7)$$

FIRE REPRESENTATION

A fire can be numerically represented in many different ways. In case of real fire in tunnels, the nature of the combustible is complex and difficult to determine precisely.

As an example, a car is mainly a compound of plastic based materials and more particularly polypropylene, rubber (tires) and fuel. The proportion and the global mass of combustible vary with the car model. Such complex framework is out of reach of the computational power available to now and hence cannot be fully numerically modeled.

The most simple combustion models are one equation based on mixture fraction model as in FDS [FDS 2005], SRC (Simple reacting scheme) or Eddy Break up models. But such models are based on the knowledge of an average global combustible which is quite difficult to determine in a general predictive manner.

The French ministry of transport [CETU 2003] suggested a way to represent fire in tunnels for 1D safety studies. This consists in imposing a source term in the energy equation associated to a kinetic of evolution. To represent the smoke and the combustion products, a passive transport of the scalar variable is added to the solved equation. The source term of this scalar is defined proportionally to the heat source term. In the current work, this 1D technique is extended to a 3D scenario.

To model the fire, it is necessary to define a source term in the energy equation to represent the heat release rate of the fire, and a source term in the transport equation of the scalar C_1 chosen to represent combustion products and more specifically the CO_2 mass fraction.

The energy equation solved is:

$$\frac{\partial \rho H}{\partial t} + \nabla \cdot (\rho V H) = \nabla \cdot \left(\Gamma_H \nabla H \right) + S_H \tag{8}$$

where H is the enthalpy, ρ, V are respectively the fluid density and the velocity , Γ_H is the diffusion coefficient and S_H the source term.
The source term of the energy equation (S_H) is the heat release rate of the fire per unit of volume as the fire is modeled by a volumetric source of heat.
For example, the max car fire power is 8MW whereas the max truck fire power is 30MW (these values are indicated in [CETU 2003]).

In our simulations, the heat release rate density retained is $1MW/m^3$. The fire volume corresponds to the fire modeled (car, truck or train)

Pollutant emission of the fire is known as a mass flow rate per MW (0.1kg/s/MW). However, the pollutant is considered as a marker: it can be considered as having the same physico -chemical properties as the ones of the domain fluid. The pollutant distribution is determined by the resolution of the transport equation of C_1 in kg per kg of fluid:

$$\frac{\partial \rho C_1}{\partial t} + \nabla \cdot (\rho V C_1) = \nabla \cdot \left(\Gamma_{C_1} \nabla C_1 \right) + S_{C_1} \tag{9}$$

where C_1 is the mass fraction of CO_2, ρ, V are respectively the fluid density and the velocity , Γ_{C1} is the diffusion coefficient and S_{C1} the source term.
Therefore, the source term to model the CO_2 fire emission (S_{C1}) is the mass flow rate of CO_2 per unit of volume as for heat.
For example with the CETU recommendations, the max car fire CO_2 emission is 0,8kg/s whereas the max truck fire CO_2 emission is 3kg/s.

RADIATION

The post treatment module, which provides an estimation of the radiation due to the hot smoke layer, is based on a simplification of smoke stratification. In fact, hot smoke and cool air are considered as two distinct layers with uniform temperature. Several techniques exist to calculate uniform layer temperatures and interface between the two layers height.

For calculus, T_u represents the uniform upper layer temperature, T_l the uniform lower one, z_i the interface height and H the tunnel height.
Cooper et al. [Janssens 1992] proposed the N% rule method: the interface height is placed where the ambient temperature (T_∞) rise equals N% of the temperature rise at the tunnel ceiling.

$$\frac{T(z_i,t)-T_\infty}{T(H,t)-T_\infty} = \frac{N}{100} \tag{10}$$

This equation permits to obtain the interface height z_i.
Quintiere et al. [Janssens 1992] obtained the two layers temperatures from an arithmetic average.

Another method, named "integral method", is based on the resolution of two equations. The first equation is an expression of the continuity equation.

$$\int_0^H \frac{1}{T(z)}dz = (H-z_i)\frac{1}{T_u} + z_i\frac{1}{T_l} \tag{11}$$

The second describes a mathematical averaging procedure, but has no physical meaning.

$$\int_0^H T(z)dz = (H-z_i)T_u + z_iT_l \tag{12}$$

Emmons [Janssens 1992] proposed to use only the first equation. T_l is defined as the temperature at the bottom and z_i is the point where the slope of temperature goes through a maximum.

To calculate the radiation of the smoke layer on users, the post treatment module uses the second method. T_l is the temperature in the lowest grid cell. The two integrals are estimated by the use of the trapezium method.

$$\int_{Z_1}^{Z_2} F(Z)dz' = (F[Z_1]+F[Z_2])\cdot\frac{(Z_2-Z_1)}{2} \tag{13}$$

These calculus permit to obtain the interface height and the temperature at this interface height.

Therefore, it's possible to determine the radiative flux (P) by the smoke layer with the Stefan-Boltzmann law.

$$P = \sigma AT^4 \tag{14}$$

$\sigma = 5.67 \times 10^{-8}$ W m^{-2} K^{-4} (Stefan's Constant)

A = Surface area of body (m²)
T = Temperature of body (K)
Using these equations (11, 12, 13, 14), we are able to calculate the radiative flux of the smoke considering as a black body to overestimate the flux.

MODEL VERIFICATIONS

In order to use these developments, it's necessary to verify if they permit to obtain correct results for tunnel simulation.

About slope variation In order to validate, the inclusion in Phoenics of a varying gravity with slope changes, two simulations are achieved with a unique slope. One is using the built-in option of a gravity given in 3 space direction and the other is programmed by the user.
The simulation model is represented on figure 3. The fire power is 20MW.
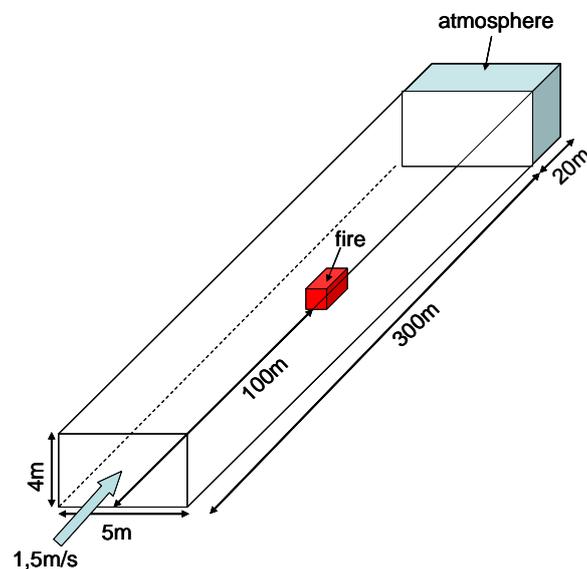
Figure 3: Representation of numerical model

The temperature results are compared for both slope of -3% and of +3%: these results are obtained 10 minutes after the fire starts. The results are almost identical and enable us to use the same development at each axial distance but with different g_y and g_z.

About fire representation It is necessary to verify that the mass of the CO_2 theoretically injected in the tunnel equals the mass of the CO_2 produced by the modeled fire.

In order to check this, a fire tunnel simulation was done. The tunnel modeled is a 20m² cross section with a length of 110m (figure 4). The fire is placed at 50m of the tunnel entry; the heat release rate of the fire is 20MW. The fire volume is 20m³. At the inlet, the air velocity is about 2m/s. At the tunnel exit, a small area of the outside atmosphere of the tunnel is simulated (10m in blue).
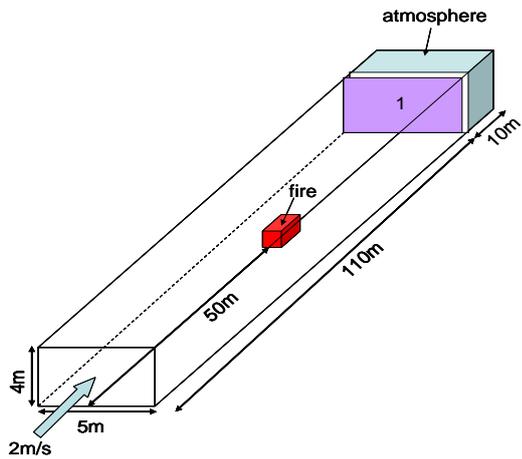
Figure 4: Representation of the tunnel numerical model
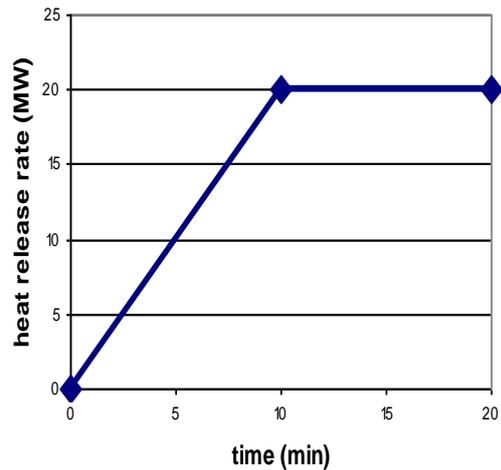
Figure 5: Fire kinetic

The fire kinetic is plotted on figure 5. Therefore, the theoretical CO_2 mass produced during the fire is 1800 kg.

The mass passed by the plane 1 is calculated. The calculated mass is about 3% less than the theoretical mass. This is due to the larger time step.

About outlet boundary condition We want to analyze the impact of the outlet boundary condition on the smoke movement. For doing this, two different boundary conditions are imposed at the modeled piece of tunnel exit represented on figure 6. The modeled fire power is 10MW.
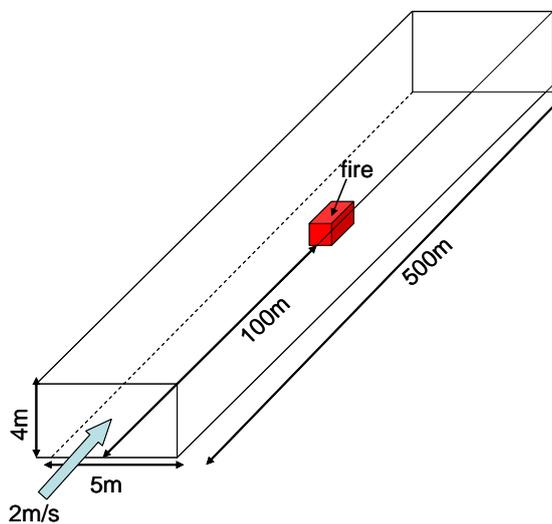
Figure 6: Representation of the tunnel numerical model

In the first case, the atmospheric pressure is imposed at the piece of tunnel exit. Whereas the air movement is well determined when there is no fire, the exiting smoke movement is not correctly determined. As presented on figure 7, the smoke layer falls down at the piece of tunnel exit.
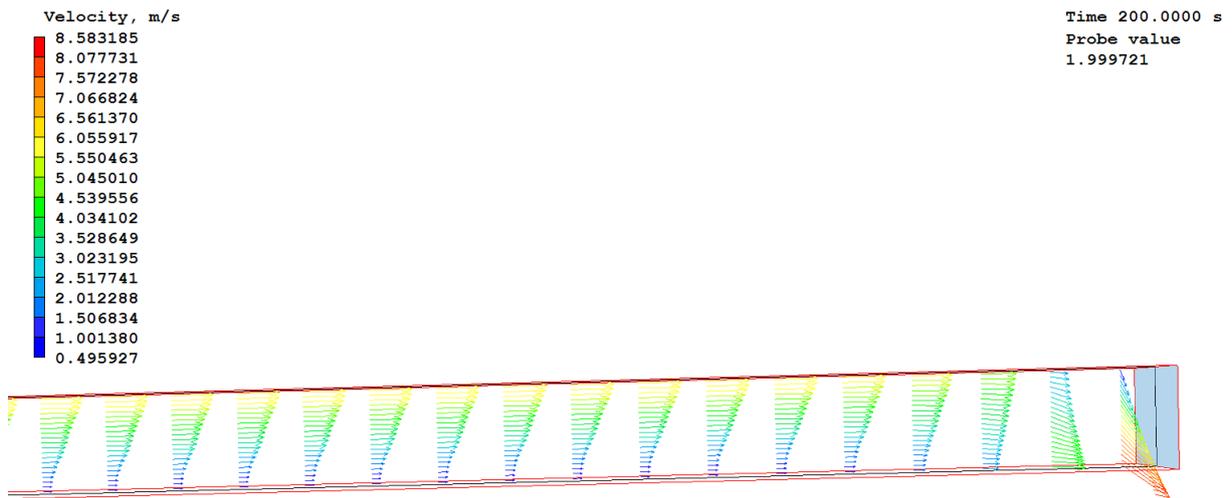
Figure 7: Velocity repartition on the 50 last meters of the piece of tunnel in the first case

In the second case, we imposed the outlet boundary condition developed above. At the piece of tunnel exit, the smoke movement is not influenced by the outlet boundary condition (figure 8).
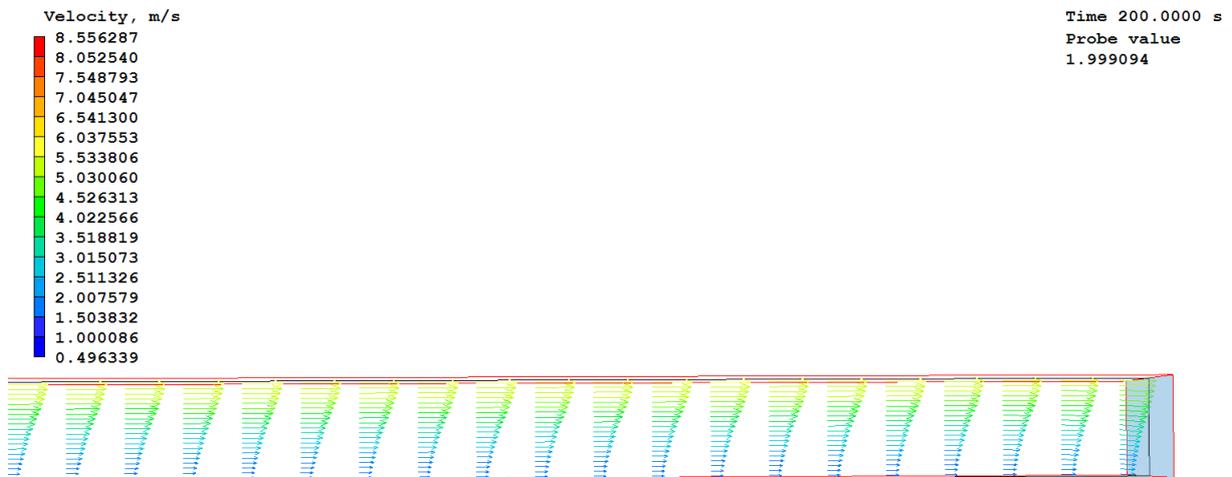

Figure 8: Velocity repartition on the50 last meters of the piece of tunnel in the second case

USE OF THESE DEVELOPMENTS

A simulation using the variation of slope, the fire representation and the radiation calculus was made to illustrate these developments. The modeled tunnel is represented on figure 9. The outlet boundary condition developed above was not used as the modeled tunnel is short.
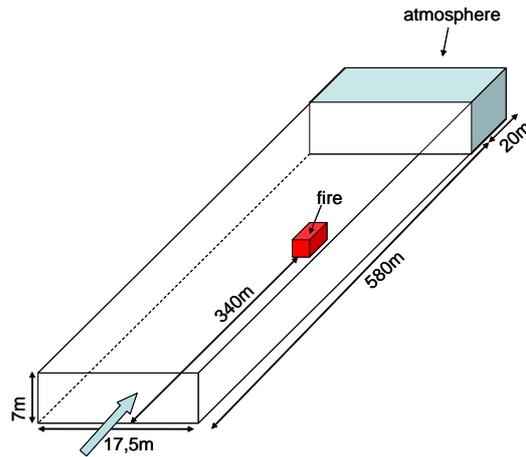
Figure 9: Representation of the tunnel numerical model

The fire maximum power is 20MW and the fire kinetic is represented on figure 5. The results presented below are obtained 17 minutes after the fire start.

About slope variation The variation of the slope has an impact on the smoke movement as presented on figure 11.
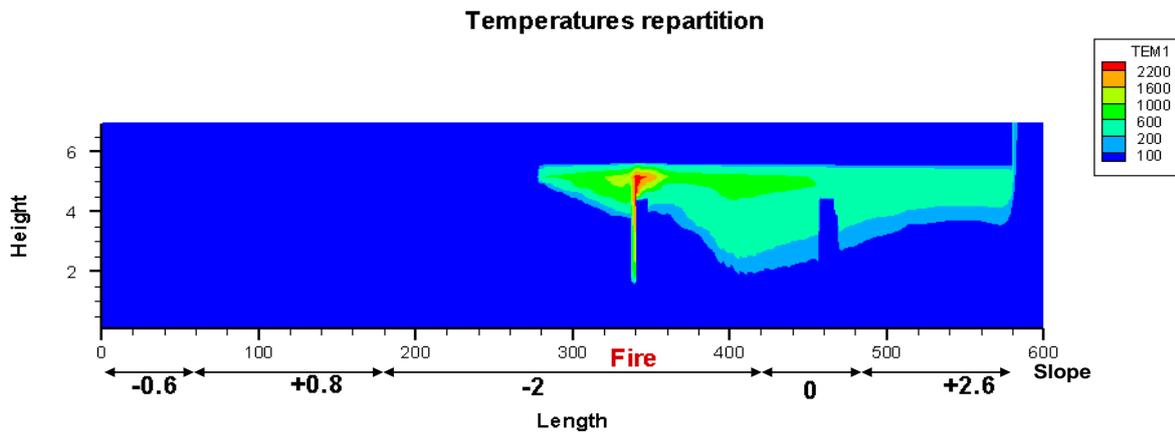

Figure 11: Temperatures repartition for the various slope tunnel

About fire representation The repartition of the concentration of CO (figure 12) permits to know if there is a risk of asphyxia for users in case of fire.
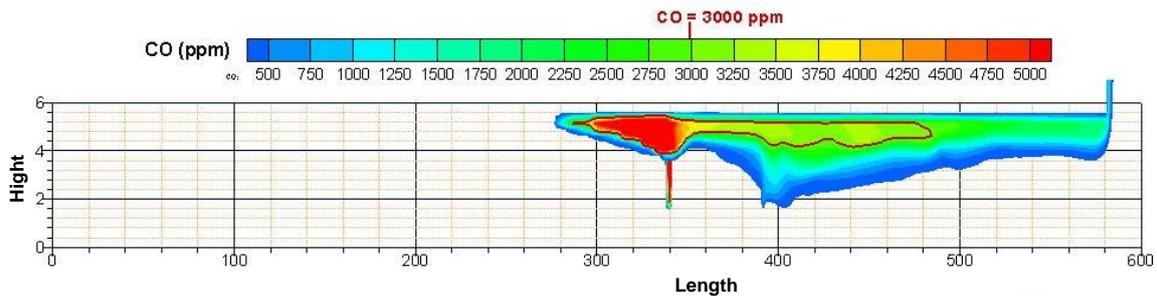

Figure 11: Concentration of CO repartition

About radiation The calculus of the interface radiative flux on users using eq.11, 12, 13, 14 (on purple on figure 9) permits to determine if the evacuation conditions are suitable for users.

The effect of radiation on the users depends of both the exposition time (t) and the radiative flux (P). Two levels of burn are fixed by Baker and al. [Baker 1983]: the first level for irreversible damages is passed when $P^{4/3}t = 600$; the second level for instantaneous death is passed when $P^{4/3}t = 1000$.
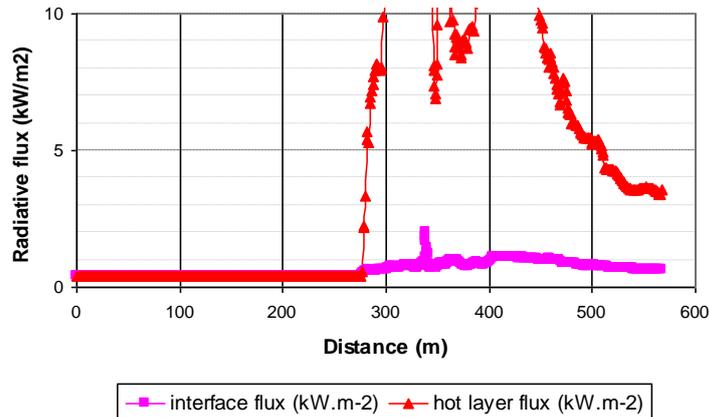

Figure 9: Radiative flux of the smoke layer

The radiative flux on the users of the interface is lower than the radiative flux of the upper layer.

Consequently, the "irreversible damages" level exposition time at the radiative flux by the interface is about 10 minutes whereas for the radiative flux by the hot layer it is about 1 minute.
Therefore, it is not convenient to calculate the radiative flux without taking in account the upper layer radiation.

CONCLUSION

The ever-increasing influence of security concept notably in French tunnel (since the fire in the Mont Blanc tunnel in 1999), often imposes the use of 3D CFD model to validate the relevance of evacuation scenario in case of fire. In this context, different physical and mechanical phenomena must be simulated in order to obtain realistic and directly usable results in a security study. The knowledge about these phenomena must be continually improved and integrated in the CFD model.

In the present article we have described some important phenomena to account for in simulating the flow behavior during a fire in tunnel:
  - the different possible slopes in the tunnel (buoyancy effect),
  - the representation of the fire which is fundamental to obtain a good estimation of pollutant repartition (passive scalar) according to the heat source,
  - the evaluation of radiation especially due to the hot smoke layer,
  - the improvement of the outlet boundary condition to simulate correctly the exit of hot smoke.

The use of CFD model in a tunnel requires a good knowledge of the models and phenomenology related to fire in confined spaces. It's fundamental to estimate the development of the smoke layer and of the temperature gradient, in order to determine if the evacuation conditions are suitable for users in case of fire. The use of a three-dimensional transient CFD model appears to be appropriate

when the tunnel geometry is complex, with slope variation, with traffic blocked on both sides the fire, etc. However more development are necessary in order to improve and obtain realistic simulations of a fire in tunnels.

REFERENCES

Baker W.E. et al. [1983]: Explosion Hazards and Evaluation, *Fundamental studies in engineering n°5*

Biollay H. [1997]: Contribution à la simulation numérique d'un incendie en tunnel par un modèle de champ, *Phd Thesis*, Claude Bernard University, Lyon France

CETU [2003]: Guide des dossiers de sécurité des tunnels routiers

FDS [2005]: Technical Reference Guide

Janssens M., Tran H.C. [1992]: Data Reduction of Room Tests for Zone Model Validation, *Journal of fire science*, Vol 10 n°6 , pp 528-555

Kobayashi M.H. et al. [1993]: Comparison of Several Open Boundary Numerical treatments for Laminar Recirculating Flows, *Int. J. Num Meth in fluids*, Vol 16 n°5 , pp 403-420

Launder B.E., Spalding D.B. [1972]: *Mathematical models of turbulence*, Academic Press London,

Patankar S.V, Spalding D.B. [1972]: *International Journal of Heat and Mass Transfer*, 15 1787

Patankar S.V [1980]: *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation

Pratap V.S., Spalding D.B. [1976]: *International Journal of Heat and Mass Transfer*, 19 1183

Rodi W. [1984]: *Turbulence models and their application in hydraulics – a state of the art review*, IAHR – Section, Second revised edition

Chen Y.S., Kim S.W. [1987]: Computation of turbulent flows using an extended k-e turbulence closure model, *NASA CR-179204*

Woodburn P.J., Britter R.E. [1996]: CFD simulations of a tunnel Fire – Part I, II, *Fire safety Journal* 35-62