Calculations of fire smokes behaviour in long rail tunnels

S. Daeron, Emmanuel Ruffin

To cite this version:

S. Daeron, Emmanuel Ruffin. Calculations of fire smokes behaviour in long rail tunnels. 1. International Conference "Tunnels and Underground Station Fires", May 2000, Hong Kong, China. <ineris-00972196>

HAL Id: ineris-00972196

https://hal-ineris.ccsd.cnrs.fr/ineris-00972196

Submitted on 3 Apr 2014

CALCULATIONS OF FIRE SMOKE BEHAVIOUR IN LONG RAIL TUNNELS

S. DAËRON, E. RUFFIN
INERIS
Parc Technologique ALATA- BP N°2
60550 Verneuil-en-Halatte
France.

Abstract:

In order to simulate fire consequences in complex underground networks, we want to implement a coupling between a 1D ventilation code and a CFD model or a zone model. The project consists in 3 main steps: the development of a 1D ventilation code whose programming structure will support a coupling with another code, the definition of exchange of boundary conditions between the 2 codes and the validation of this exchange. In this paper we present our new 1D code developed in this framework. A case study shows the global reaction of the flow to a fire and proves the interest of keeping into account the whole network instead limiting the calculation domain to a zone closed to the accident.

INTRODUCTION

Accurate knowledge of accidental phenomena in long rail tunnel such as fire and release of toxic or inflammable gases constitutes an important goal for safety management. This knowledge will, on the one hand, provide for better planning of the operation and the calculation of the dimensions of ventilation systems and other safety measures while on the other hand can be used to define procedures to limit the consequences of an accident. The various techniques presently used to predict the effect caused by fires or accidental gas leaks can only be applied to the simplest ventilation systems or network geometry. Regarding modelling, the CPU calculation time needed by a field code to model a fire in a whole network is prohibitive. Thus generally the calculation domain is limited to a small zone close to the accident for fixed boundary conditions. As a result, overall effects of an accident in a complex tunnel (e.g. Channel Tunnel) or in a network of suburban tunnels (e. g. underground) are practically impossible to predict with a satisfying accuracy. This is mainly due to the fact that one cannot simultaneously take into account the global and environmental parameters of a ventilation network and the local physical effects of an accident.

The INERIS project aims at carrying out a numerical simulation tool designed to predict the global behaviour of the heat or smoke in a tunnel. There are different families of numerical codes available. There are mono-dimensional codes, which allow the calculation of ventilation systems in complex networks. It's possible to implement in those codes models for heat and mass transfer. However this kind of model cannot take into account some physical effects like stratification or backlayering. Another family is the field model family (Computational Fluid Dynamics), which allows a rather good modelling of various physical process if appropriate models are used (turbulence, buoyancy, combustion...), see for example [1]. But this kind of code cannot be used for the calculation of a complete network because of the CPU calculation time, as said above.

So, in order to use the complementarity of these families we want to implement a coupling between a ventilation code (network model) and a CFD model. Each code will use the boundary conditions provided by the other one. The transfer of boundary conditions will be validated through theory, well known case study and numerical comparison as well as experiments. Experiments will be carried out by means of a cold mock-up facility, which has been purpose-built in INERIS.

1D CODE BRIEF DESCRIPTION

Main features

A new version of our existing ventilation code VENDIS (VENtilation DISpersion), including new major capabilities, has been already achieved [2]. This program now includes a mesh builder (definition of network geometry), a tool which helps the user to define various accidental scenarios, a new calculation model of the transient network flow taking into account density variations and a post-processor for results' display (cf. Figure 1).

The mesh builder allows the user to define the network in terms of several geometry elements, which represent the whole network of galleries. A particular effort focused on the automatic calculation of connections existing between the geometry elements. This allows the user to easily delete or add an element or to modify the network geometry.

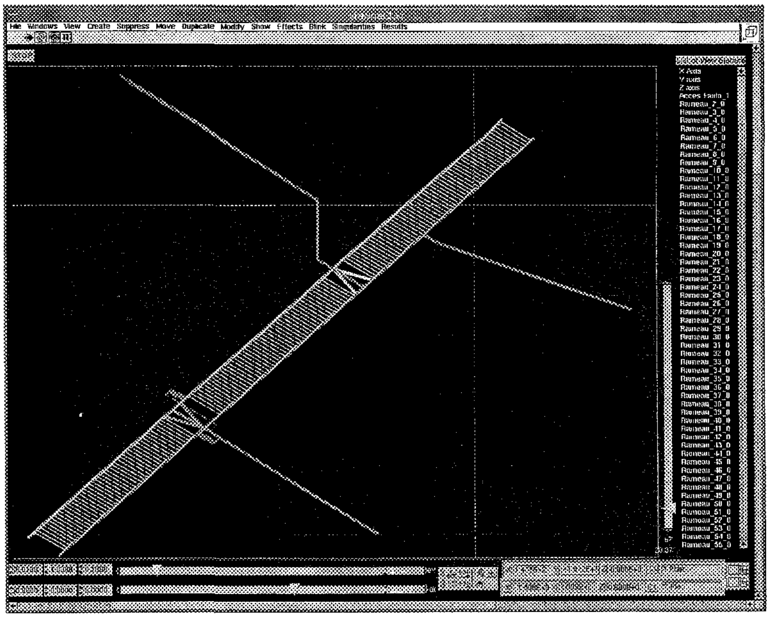


Figure 1: A ventilation network example, a twin rail tunnel.

Modelling the air/smoke/gas flow in an underground network is complex because many environmental parameters have to be taken into account during an accidental situation. Each phenomenon in a gallery has an effect on the overall flow, which is translated into equations. Among all effects, which can occur in underground networks, let's notice:

- natural and mechanical ventilation (fans, jet fans)
- piston effect
- singularities of the network
- fire, gaseous emissions
- buoyancy

A particular attention has devoted to modular program structure in order to make easier further developments like implementation of the different phenomena. Each transient phenomenon is bound with a so-called scenario-file, which groups all the parameters necessary for the modelling and of course the scenario describing phenomenon evolution.

Calculations

The Hardy-Cross Algorithm (HCA) [3] gives the different flow rates and the pressure losses for all the branches. This algorithm is based on two kinds of equations. First it uses the Kirchoff's law applied to the flow rates and pressure losses. Then the Ventilation Network Theory makes it possible to choose independent equations among them in order to calculate the unknown variables. The detailed method is described in [2]. Secondly, HCA uses equations characterising the branches of the network. Accidental and environmental effects are also modelled at this step and linked with their pressure effects in the network.

A good taking into account of phenomena requires necessarily a fine description of the effect in the branch. Consequently, we can refine the concerned branch by cutting it into several branches. This can strongly increase the total number of branch in the HCA. The calculation time will then also increase. To avoid the increase of number of branches while preserving a good description of the effects, we developed a near field model. Our idea was to include internal points in the different branches that need a local refinement. These points will never define any intersection but just segments of the branch, on which local equations are solved to estimate the parameters on each segment. As a result the so defined portions contribute numerically to the overall evaluation of the branch but don't appear in the resolution of the HCA. Of course, according to the branch we can use or not this near field model.

So we developed a first 1D/1D-coupled model where the 1D-coupled model gives a refined description of the accidental source (heat and mass transfer). This is the first step to a 1D/3D coupling since we hope that various kind of existing field models can be coupled in the same way in the future.

Particular models used in our code

Our aim is essentially the modelling of transient heat and mass transfer phenomena. Thus, it's necessary to take into account the inertia of air mass and to model heat and mass transfer. All effects are translated in pressure terms and included in the characteristic equations of the branches. For example to model the resistance of the galleries we use the well-known RQ^2 model.

- **Inertia model**

Let's denote:				
	Q^t	the flow-rate at the time t	L	the length
	S	the section of the branch	ρ	the density
	Δt	the time step	γ	the acceleration of the flow

We wrote a model for pressure losses induced by inertia $\Delta P_{inertia}$, which is inspired from the fundamental principle of mechanic. If we applied this principle to the air mass contained in a branch, we can write that this inertia force $F_{inertia}$ equals the product of $\Delta P_{inertia}$ with the section of the branch:

$$F_{inertia} = m\gamma = \Delta P_{inertia} .S$$

In this way we obtain the model representing inertia of air mass.

$$\Delta P_{inertia} = \rho L \frac{Q^{t+1} - Q^{t}}{S \, \Delta t}$$

We made a first validation with on the basis of tests carried out in the Norwegian Fodness tunnel (cf. [5]). Because of missing data, we made some simplifying hypothesis. The data taken into consideration were:

Length =6565 m	Natural ventilation velocity = 1 m/s
Density = 1.30 kg/m^3	Section=52 m^2

With these hypotheses, the whole tunnel contains about 400 t of air. This mass is accelerated through groups of jet fans at different sections, until a velocity of 4.5 m/s is reached. The ventilation stops after 360 s. This simulation needs transient scenarios to simulate the fans. Figure 2 shows the air velocity variation in the tunnel. We see that in our calculations the stable airflow is not completely reached. This could be explained by our simplifying hypotheses and/or by bad adjustments of parameters (density, fan characteristics...). The come back to the initial flow seems to be well predicted by our model.

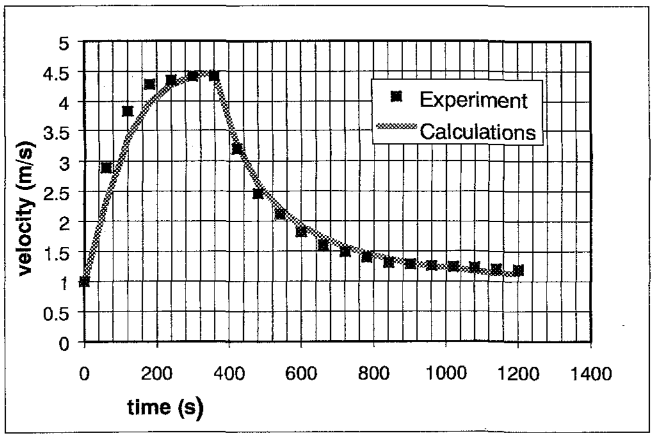


Figure 2: Time evolution of the velocity in the tunnel

- **Heat transfer model**

The model is based on the fact that time step is small enough to ensure that a quantity of heat cannot “jump” a complete branch of the network. We suppose that the heat transfer is homogeneous in a branch. The algorithm treats successively all intersections of the network. At an intersection, writing the heat flux conservation, we calculate a mixture temperature while supposing that the mixture is instantaneously achieved. The model convects this temperature in the concerned branches respecting convection length. The remaining part of the branch is gradually treated in the same manner.

The mass transfer is modelled in the same way and won’t be further treated here. Note that the smoke is associated with the heat front and propagates with the same velocity.

A SIMPLE RAIL TUNNEL CASE STUDY

Presentation

The following figure shows the geometry of the case study. The geometry points and the internal points necessary for a good description of the accidental effect are represented on *Figure 3.* All sections of the branches are supposed to be circular. Table 1 gives the other parameters.

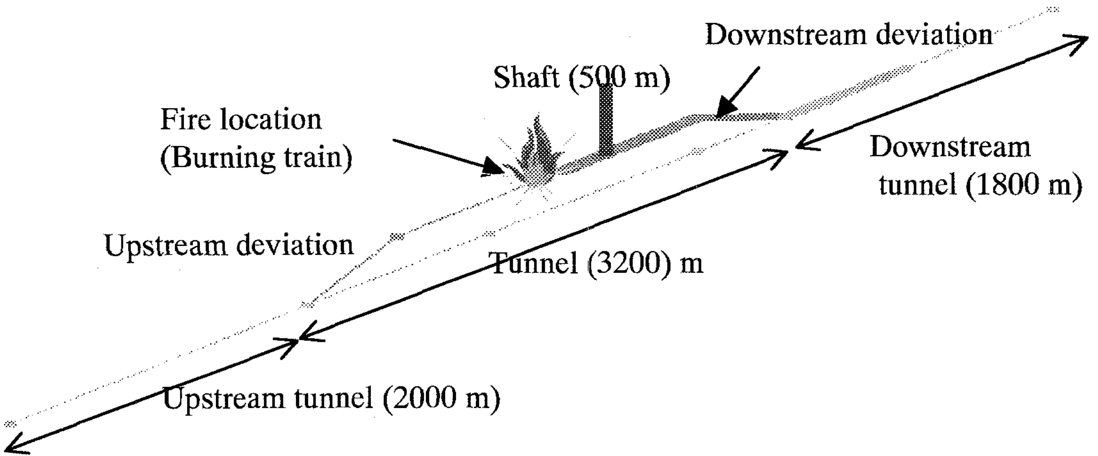


Figure 3: case configuration

Tunnel and deviation	Shaft
$friction\ \ coefficient = 0.009$	$friction\ \ coefficient = 0.005$ (a closed shaft is modelled by an infinite coefficient)
$Hydraulic\ \ diameter = 8\ \ m$	$Hydraulic\ \ diameter = 3.5\ \ m$
$heat\ \ transfer\ \ coefficient = 7\ \ W/K/m^2$	$heat\ \ transfer\ \ coefficient = 7\ \ W/K/m^2$

Table 1: Parameters for the different branches

We supposed that a piston effect (due to a train) causes a fixed flow rate equal to either 400 kg/s or 600 kg/s in the upstream tunnel. We supposed that a burning train is stopped in an underground station (the deviation) 300 m before an exhaust shaft. The convected firepower is fixed constant to 20 MW. The refinement points were added all ten meters in the whole deviation, after the fire location, in the shaft and in the first half of the downstream tunnel. This refinement procures a good description of the heat transfer. Note that in all cases we supposed that the wall and the ambient temperature are equal to 273 K.

The aim of this case study is to point out the global reaction rather than a precise study of the temperature field. This example shows that only a fire can strongly disturb an established flow. Figure 4 shows the different points where we follow the transient effects. These points are used to represent the time variations of temperature and mass flow-rates at different locations.

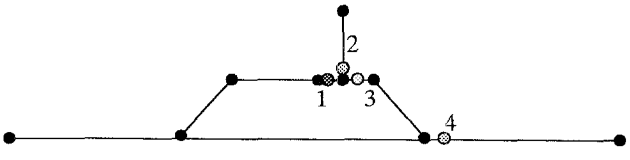


Figure 4 : points of observation of transient effects

Transient accidental situations

- **Fire with closed exhaust shaft**

In this case we suppose that the shaft is closed. The fire starts 20 s after the beginning of simulation. A steady situation for the temperature field is reached after about 8 min (cf. Figure 5). In this case the heat transfer does practically not affect the flow rate. Downstream from the fire, in the deviation, the temperature rapidly exceeds the maximum temperature (about 60°C) that people can support without protection (cf. Figure 7). Even rescue service with protection cannot intervene in the deviation downstream the fire (cf. Figure 6). The smoke progresses with a velocity of about 3 m/s. Figure 8 shows that after 10 min half of the downstream tunnel is invaded by the heat and that after about 20 min all the downstream tunnel is concerned.

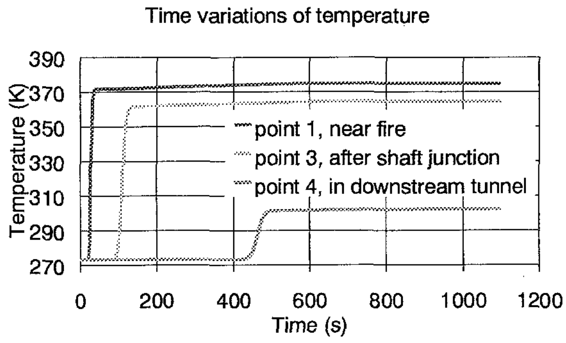


Figure 5

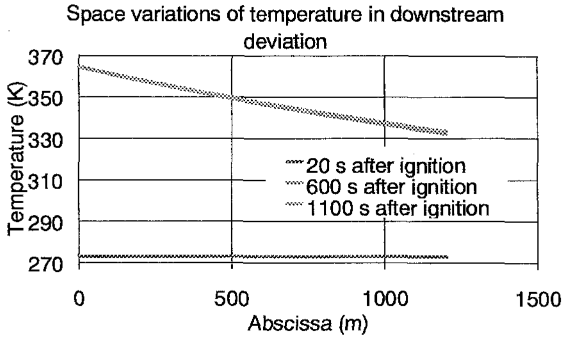


Figure 7

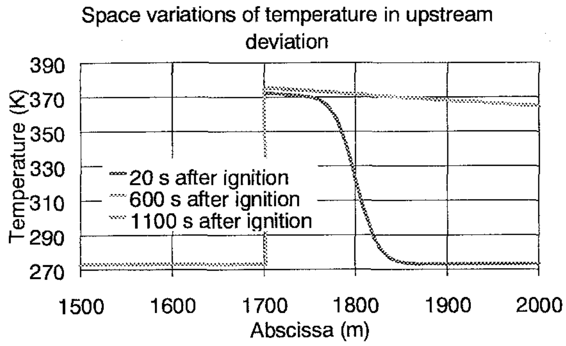


Figure 6

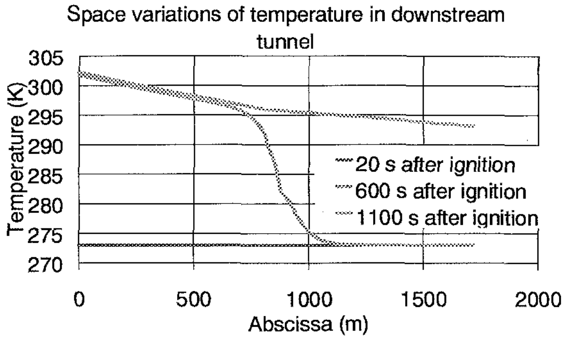


Figure 8

- **Fire with exhaust shaft and a main flow rate equal to 400 kg/s**

This case is the same as above but with the shaft opened. We let the flow establish by itself. We can see (cf. Figure 9) rather important variations in the flow rates after the ignition of fire. In the shaft (point 2) the flow rate increases (+200 kg/s) whereas in the downstream deviation (point 4), we note an important decreasing (- 160 kg/s) that causes an inversion of the flow direction. The inversion of the flow occurs after 174 s. That's early enough to prevent the apparition of smoke in the downstream tunnel. At this time the smoke occupies the whole shaft and the temperature are quite high (cf. Figure 12). 200 m of the downstream deviation is touched by a smoke front with a high temperature gradient (cf. Figure 13). If we suppose that 60°C is the limit that passengers can support, this limit is overstepped along 350 m in the downstream deviation. This situation is transient (cf. Figure 10, point 3) and because of the inversion of the flow at the time 174 s in the downstream deviation, all the smoke is evacuated in the shaft. The portion of deviation between the fire and the shaft stays as for it inaccessible for rescue services. The stable situation is reached after about 300 s for the flow rate (cf. Figure 9) and 350 s for the temperature (cf. Figure 10). In the upstream deviation (cf. Figure 9, point 1), we note small variations of the flow rate that causes small variations in the maximum temperature of the fire(cf. Figure 11, abscissa 1700) whereas the power of the fire heat release rate is fixed constant.

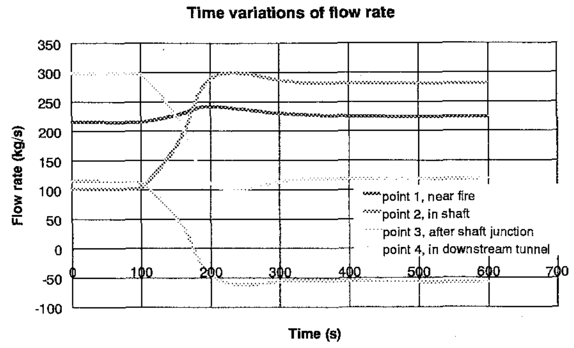


Figure 9

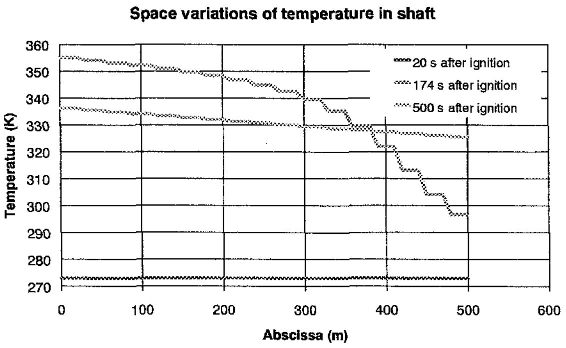


Figure 12

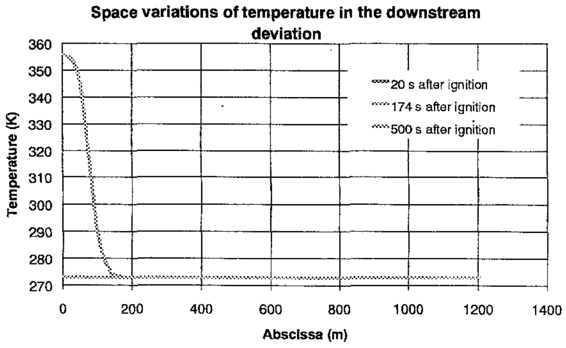


Figure 13

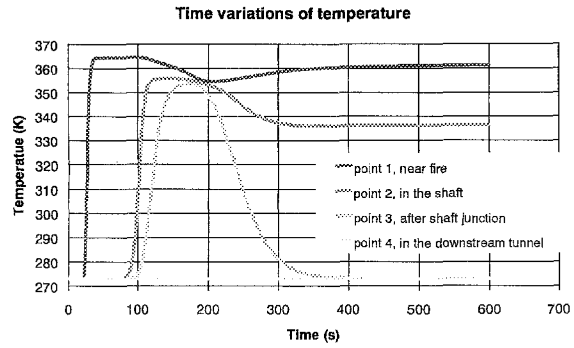


Figure 10

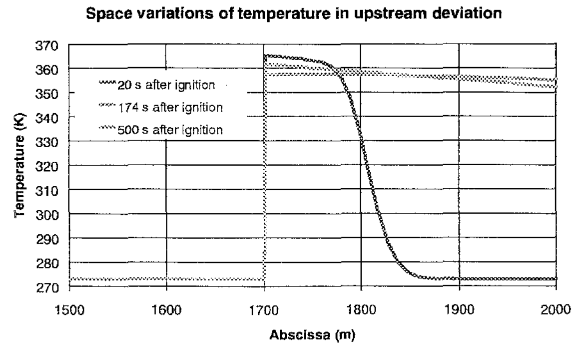


Figure 11

- **Fire with exhaust shaft and a main flow rate equal to 600 kg/s**

The value of the main flow rate upstream the fire plays an important role. We suppose now that the piston effect in the upstream tunnel causes a mass flow rate of 600 kg/s that corresponds to a velocity equal to 9 m/s. Figure 14 shows the decreasing of value of flow-rate

(-120 kg/s) in the downstream deviation and the augmentation of the flow-rate in the shaft (+120 kg/s) due to the chimney force. This flow rate variation is sufficient to prevent the inversion of the flow in the downstream deviation observed in the previous case. The smoke is therefore distributed in the shaft and in the downstream deviation (cf. Figure 17, 700 s) and later in the downstream tunnel (cf. Figure 18, 1400 s). In the downstream tunnel, the apparition of heat and smoke occurs after 1200 s (cf. Figure 15, point 4).

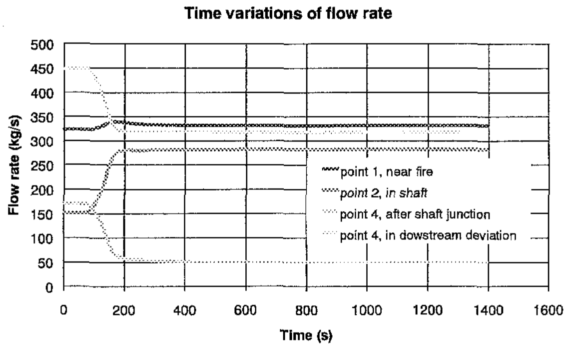


Figure 14

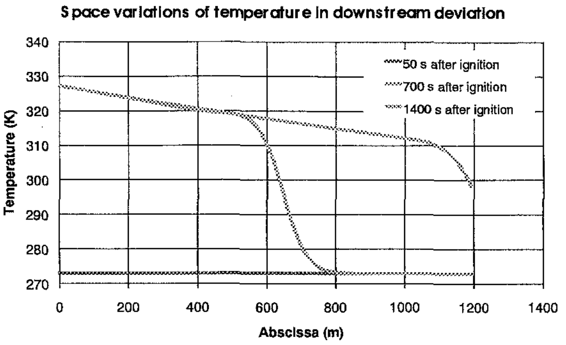


Figure 17

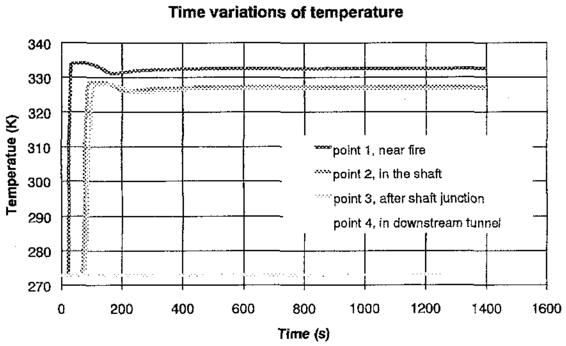


Figure 15

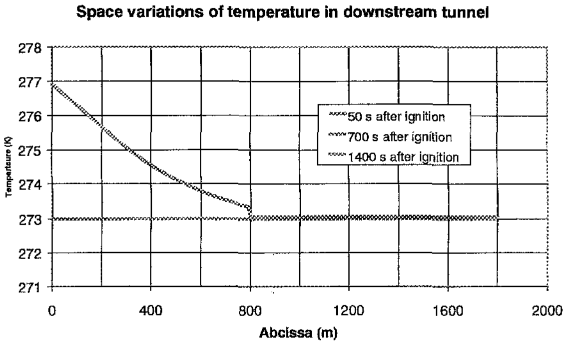


Figure 18

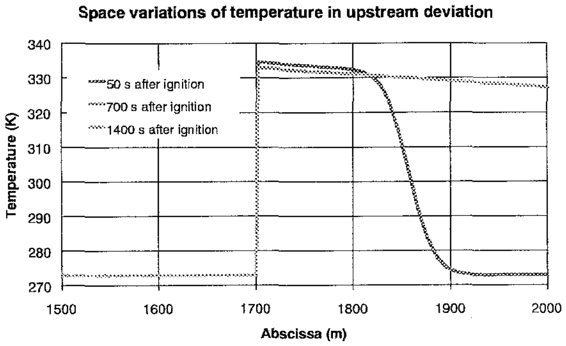
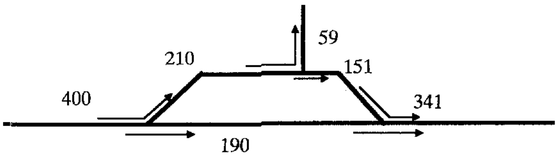


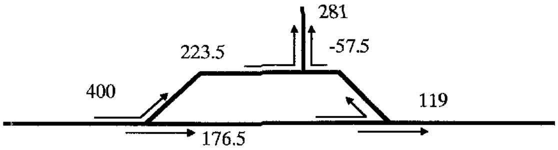
Figure 16

Concluding remarks

We represent on the following figures the steady airflow repartitions before and after fire ignition. *Figure 19* clearly shows the inversion of the flow in the downstream deviation when the main flow rate is equal to 400 kg/s. Figure 20 shows also the important variations of the value of the flow rates for a main flow rate of 600 kg/s.

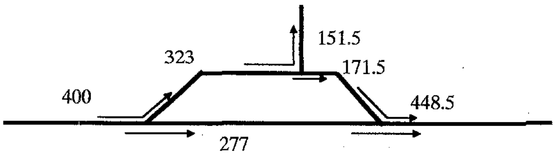


Before Ignition of fire

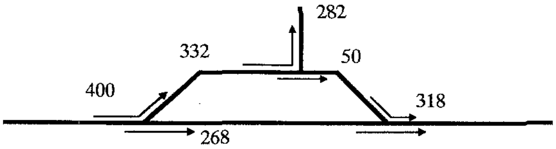


Steady flow with fire

Figure 19: repartition of the flow rates, for a main flow rate equal to 400 kg/s



Before Ignition of fire



Steady flow with fire

Figure 20: repartition of the flow rates, for a main flow rate equal to 600 kg/s

We insist on the fact that these flow variations are only due to the occurrence of a fire since all other parameters stay constant. In a more realistic aspect, this main flow variation can be induced by movements of trains in the tunnel, which may induce uncontrolled displacement of smoke around the accidental source. Then, the interest of accurate previsions of global accidental scenarios is obvious. The good response to an accident is strongly bound to the knowledge acquired from these scenarios and the previsions of the temporal evolution of accidental sources. These conclusions have been also expressed by a French Government commission which was created after the 'Mont-Blanc' catastrophe (cf. [5]).

PERSPECTIVES

INERIS further work will be devoted to a more accurate modelling of the accidental zone. In order to do that, we are going to extend the work regarding models coupling procedures to others kinds of model like zone-model and CFD model. The aim of the coupling is to provide precise transient boundary conditions for the field code while the global network also depend on the phenomena, which occur in the 3D zone. The coupling will be then completely interactive. This coupling will need an important validation phase. We will use well known cases to validate our results but we will also carry out some experiments to validate the transient aspects, the boundary condition problems and the exchange of data between the coupled codes. A reduced scale model has been specially built at INERIS in order to carry out experimental validations (cf. *Figure 21*).

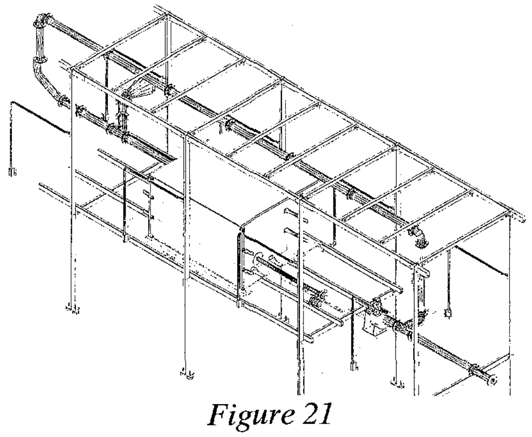


Figure 21

It will mainly consist to connect simple network (some branches and one or two fans) to a three–dimensional geometry. This three–dimensional geometry will be a parallelepiped rectangle representing for example:

- a ventilated room (office, small building scale 1/10, 1/2),
- a section of gallery (road tunnel, covered section scale 1/30, 1/10).

The simulation of accidental sources will be obtained by using air-helium mixtures. The helium proportion will characterise a hot gas source (e.g. an air-helium mixture with 84% of helium has a density equal to that of products of combustion at 800 °C). Using SF_6 mixtures the behaviour of heavy gas can also be studied.

ACKNOWLEDGEMENTS

The financial support provided by the French Ministry of the Environment, for both the numerical and the experimental aspects of this project, is gratefully acknowledged.

BIBLIOGRAPHY

[1] E.Ruffin, A.Carrau, C. Cwiklinski, Numerical simulations of locomotive fires in the Lyon-Turin tunnel, Conference ITC "Tunnel Fires and Escape from Tunnels", 5-7 May 1999, Lyon, France.

[2] E. Ruffin, S. Daëron, Development of a tool for simulation of accidental situations in underground networks, Rapport d'avancement annuel 1999, INERIS, in French.

[3] H. Cross, Analysis of flow on networks of conduits or conductors. Engineering Experiment Station, Bulletin 286, 1936, University of Illinois.

[4] G. Lotsberg, Measurement of the wall friction factor and the installation efficiency of jet fans, 9th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels, Aosta Valley, Italy, 1997.

[5] Rescue in case of tunnel fire. Work group managed by the 'Direction de la Défense et de la Sécurité Civiles'. January 2000, in French.