



Definition and optimisation of the efficiency of smoke extraction in a road tunnel

M Oucherfi, Bernard Gay, Antoine Mos, Pierre Carlotti

► To cite this version:

M Oucherfi, Bernard Gay, Antoine Mos, Pierre Carlotti. Definition and optimisation of the efficiency of smoke extraction in a road tunnel. ISAVT13, May 2009, New Brunswick, United States. hal-03793413

HAL Id: hal-03793413

<https://hal.science/hal-03793413>

Submitted on 1 Oct 2022

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Definition and optimisation of the efficiency of smoke extraction in a road tunnel

Oucherfi, M.^{1,2} ; Gay, B.¹ ; Mos, A.² ; Carlotti, P.³

¹ Université Claude Bernard Lyon I, 43 Bd du 11 Novembre, 69100 Villeurbanne, France

² CETU, 25 Av François Mitterand, 69500 Bron, France

³ CSTB, 84 Av Jean Jaurès, 77447 Marne La Vallée, France

ABSTRACT

Transverse ventilation is commonly used in long tunnels, tunnels with two-way traffic, or in tunnels which are frequently congested. This strategy consists of extracting the hot gases near the ceiling in order to prevent the smoke from propagating throughout the tunnel and to preserve the natural stratification of smoke. The aim of the design process is to obtain a high extraction efficiency at an optimal cost.

Thus far, the regulations related to the design of such systems have been based mostly on empirical considerations, for example the number of dampers or their size and shape. The aim of this work is to determine the most influential parameters leading to an increase in efficiency. As a first step, a universal expression of the efficiency is given, avoiding in particular the need to define the interface between fresh air and smoke, which is often unclear. To achieve this, only integral values over the whole section of the tunnel, i.e. fluxes, are used. The efficiency of the system can then be defined as the ratio of the buoyancy flux extracted through the damper to the buoyancy flux released by the fire. The yield is defined as the efficiency related to the cost of extraction.

A numerical study using the CFD code Fluent is carried out to investigate the influence of various parameters on the efficiency: the layout and shape of the dampers in the ceiling, the distance to the fire, the speed of the air flow in the tunnel, the flow rate in the extraction duct and the heat release rate. The modelling hypotheses are validated by comparative tests.

The study of the behaviour of a single damper is used to characterise the local phenomena. A full set-up with several dampers is then studied. It appears that the air flow in the tunnel is the most important parameter, whereas the shape of damper has little influence on the efficiency.

1 INTRODUCTION

1.1 General Description

Here we focus on the relatively rare event of a vehicle catching fire in a road tunnel. The fire behaviour is difficult to predict as it is dependent on vehicle type and cargo. For example, a small passenger car fire (with a 2 MW heat release rate) burns very differently to a petrol tanker (of more than 200MW heat release rate).

In a tunnel fire, the main dangers to the health of people trapped inside are :

- 1 . the reduced visibility due to the opacity of smoke (sometimes less than a few metres), which slows down passengers on their way to emergency exit.

- 2 . the toxic gases and low concentration of oxygen, which provoke suffocation to users that who remain trapped in the tunnel.

3 . the high temperature (up to 1300°C in large fires), which may burn people directly by radiation or compromise integrity of tunnel infrastructure such as the lighting or ventilation systems leading to heavy objects falling from above.

It is clear that the priority is to remove the smoke from the tunnel so that passengers can egress safely and emergency services are protected whilst fighting the fire.

1.2 Behaviours of smoke

When a fire occurs in a confined environment, hot smoke usually rises and remains beneath the ceiling. If there is no wind, smoke will propagate evenly in both directions with symmetric smoke fronts, at a velocity that can exceed 4 m/s in the case of a heavy goods vehicle fire. This front continues to propagate along the ceiling until the smoke cools and fills the entire tunnel cross-section. This may not occur until the front has propagated several hundred metres from the fire.

In real tunnels however, the environment is never still. A longitudinal wind always exists, its magnitude depending upon the length of the tunnel, its orientation in the landscape and the meteorological conditions at both portals. This current pushes smoke downstream. The base of the fire plume is an obstacle to the air flow and creates instabilities causing the upper part to lean downstream. Interactions between the plume and the boundary layer at the ceiling creates two counter-rotating vortices, pushing smoke to the side walls.

One may think intuitively that the presence of the air current would tend to push all of the smoke towards one end of the tunnel. However, if the air flow velocity is less than a critical velocity, backlayering can be observed. This is a phenomenon whereby a stable stratified smoke layer propagates upwind beneath the ceiling due to a lack of speed in the corners of the tunnel. Various experiments in typical geometries and forms exist, but the front speed of backlayering remains very hard to predict.

In the contrary, if the velocity of the air flow is sufficiently high, the stratification is destroyed by turbulent shear and the entire traffic area is filled with smoke.

1.3 Ventilation Strategies

In order to quickly remove smoke, different ventilation strategies exist, depending mainly on the type of traffic: one- or two-way, probability of congestion.

The first main strategy is the known as longitudinal ventilation. The objective of this method is to push all smoke downstream with the help of jet-fans attached to the ceiling. The passengers located are able to escape, while those blocked upstream by the fire are protected from the smoke by the inflow of fresh air from upstream. This strategy is simple and efficient but it can only be used for one way tunnels without traffic congestion.



Fig 1 . Longitudinal ventilation strategy

When there are passengers on both sides of the fire, the other main strategy is used. Transverse ventilation exploits the natural phenomenon of buoyancy by extracting smoke through dampers in the ceiling. This leaves the lower half of the tunnel clear of smoke and

prevents it from propagating over the whole tube length, enabling passengers to evacuate in both directions. Such a system requires stratification to be maintained.

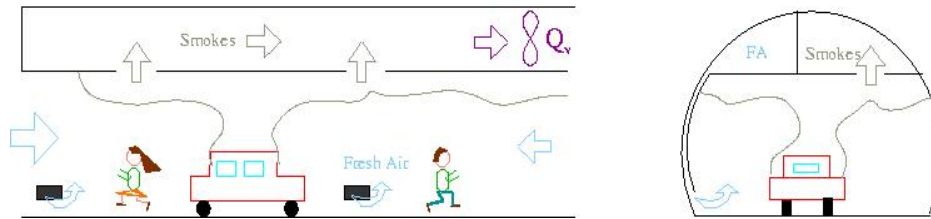


Fig 2 : Transversal ventilation strategy

This study focuses on the transverse ventilation system. Such a system is divided into three parts: the ventilation plant usually situated outside of the tunnel (contrary to the longitudinal system where the fans are near the fire), the ducts that are generally situated above a false ceiling and connect the plant to the dampers, and the dampers themselves, which can be remote controlled so that only the dampers close to the fire are opened. In most cases fresh air is also provided through small vents near the floor. This weak fresh air injection is not taken into account here.

1.4 Design Issues

Designing a transverse system is delicate. It must satisfy two objectives: limit the propagation of smoke over a large distance whilst not creating high air velocities and associated turbulence which have an adverse effect on the stratification. Furthermore, if the extraction is too strong it may extract the fresh air underlying the smoke layer, a phenomenon called plug-holing.

Regulations give empirical advice on the number of trap doors and the mass-flow rate of gases extracted through them, but most of the design work remains pure *savoir-faire*. That is why the aim of this work is to define which parameters the efficiency of such a system depends on.

Therefore, we should first give a good and simple definition of the efficiency criteria, then identify the possible parameters to be taken into account and finally to evaluate their influence.

2 DEFINITION OF EFFICIENCY AND YIELD

2.1 General considerations

The first idea is to define efficiency as the percentage of smoke produced by the fire that are withdrawn. It is therefore the ratio between the quantity of smoke extracted over the total quantity of smoke produced by the fire. Efficiency can then be increased by extracting larger quantities of gas to increase the volume of smoke extracted. However, as we have seen before, if extraction is too vigorous, a large amount of the extracted gases will be composed of fresh air.

We therefore introduce another parameter, the yield, which is a measure of the smoke extraction cost-efficiency. We define yield as the concentration of smoke in the extracted gases, which is the ratio between the volumes of smoke extracted and the total gases withdrawn.

2.2 Efficiency In Literature

In some experimental studies (for example \ref), the method consists of measuring the smoke concentration. While its composition strongly varies for each case, it is sometimes easier to measure the quantity of oxygen and deduce the smoke concentration. However, to calculate efficiency, the quantity of smoke produced must be determined, which is only possible with laboratory models.

In numerical studies (like \ref), efficiency could be based on volume flow rate as:

$$\varepsilon_i = \frac{Q_v^i}{(Q_{s1} + Q_{s2})}$$

where Q_v^i - volume flow rate extracted through the damper, m^3/s ; Q_{s1} and Q_{s2} - volume flow rate of smoke produced by the fire, m^3/s . All values are calculated by integrating values over the section where the flow is moving away from the fire.

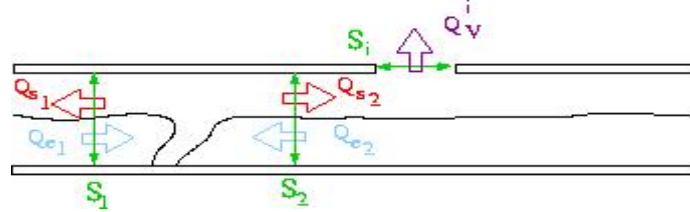


Fig 3 : Values used in the definition of efficiency by B.Ribbot in \ref

This method implies that the flow in the tunnel is very well stratified, which is not always the case, and that the user has to define the interface between fresh air and smoke, which is usually very unclear.

2.3 Chosen Definition

We search for a definition that does not require the interface between smoke and fresh air to be defined. We therefore use integral values over the whole tunnel cross-section.

Our definition is therefore based on buoyancy fluxes that characterise the motion of smoke due to the heat-induced density difference. This is interesting because buoyancy is the driving force of the smoke flow, both in the vertical direction (light smoke rise to the ceiling) and the longitudinal direction (propagation of smoke front). This is also advantageous as it can be easily related to the heat release rate.

We should recall that buoyancy rate over the tunnel cross-section is equal to the smoke layer buoyancy rate since the fresh air contribution is zero (no density difference). The turbulent mixing has also no influence on the value of buoyancy rate.

Buoyancy is created in this case by the fire, and is lost either at the walls or through the extraction of hot gas (extraction of fresh air is neutral).

For a given surface S , the buoyancy B_s is defined as :

$$B_s = \iint \frac{\rho_0 - \rho}{\rho_0} g V dS \quad (1)$$

where ρ – density of the gases $kg.m^{-3}$; ρ_0 - density of ambient air at the temperature T_0 , $kg.m^{-3}$; g - gravity, $m.s^{-2}$; V - average velocity of the flow over S , m/s and dS the normal to S , m^2 .

If S is the closed surface surrounding the fire, and if heat losses at the walls are neglected, then mass and enthalpy conservation leads to the calculation of the buoyancy B_P produced by the fire :

$$B_P = \frac{g P_c}{\rho_0 C_p T_0} \quad (2)$$

where P_c - convective heat release rate, W; C_p - air specific heat for T_0 , W.kg⁻¹.K⁻¹.

If S_i is the surface surrounding the damper i , the buoyancy B_{Di} for this damper can easily be related to the mass flow rate Q_{Mi} and volume flow rate Q_{Vi} through it as :

$$B_{Di} = g \int V dS_i - \frac{g}{\rho_0} \int \rho V dS_i = g Q_{Vi} - \frac{g}{\rho_0} Q_{Mi} \quad (3)$$

In this case, the efficiency \mathcal{E} of the system is defined as the ratio of the buoyancy flux extracted through all dampers to that produced by the fire. The yield Ψ is defined as the buoyancy flux extracted to the maximum that could be extracted by the ventilation plant.

As a result, we can write:

$$\mathcal{E} = \frac{\sum_i B_{Di}}{B_P} \quad (4) \quad \text{and} \quad \Psi = \frac{\sum_i B_{Di}}{g Q_V^{total}} \quad (5)$$

Studies of this subject can be conducted two different ways: with full or reduced-scale experiments or with numerical simulations. Given that full-scale tests in real tunnel are far to expensive and that lab models are not very convenient for frequent changes of geometry, we chose to complete this study with the use of numerical simulations. Due to the three-dimensional nature of fire flows in tunnels, the use of CFD (Computational Fluid Dynamics) is required. Although, computational time can be very long, it is a good way to investigate a large number of ventilation combinations, provided that numerical models are chosen correctly.

3 MODELING HYPOTHESES

To minimise the complexity of the study, we would like to neglect heat transfers at the walls and radiation in our calculations. However, in order to be as realistic as possible, we should first see if this influences the performance of a damper.

3.1 Description Of The Case

For a better understanding of local effects of withdrawal, the study is first reduced to a single damper.

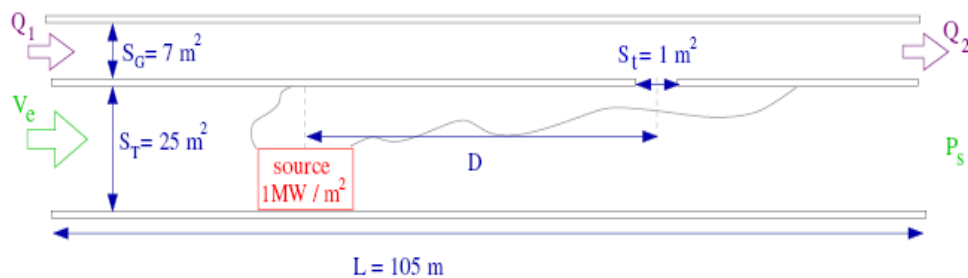


Fig 4 : Description of the one damper model

The tunnel model we study is 105 m long with a cross-section of 50 m² section. A smoke extraction duct, 7 m² in cross-section, runs parallel over the whole length of the tunnel. The two are connected separated by a damper of cross-sectional area 2 m² and are elsewhere separated by a false ceiling. As the model is symmetric about its centerline, we model only half of it in our computational mesh, which contains about 1 million elements.

Extraction through the damper is equal to the difference between the mass flow rate entering the duct Q_1 and the mass flow rate extracted at the exit Q_2 , which is 20 m³/s.

The fire is represented by a 27 m³ volume (almost the size of a small truck) located on the plane of symmetry. We do not use a combustion model, but a volumetric energy source of 1 MW.m⁻³. this value has been chosen because previous experiments (ref) have shown it gives realistic temperatures in the tunnel. The total convected heat release is then equal to 13,5 MW for the half-domain.

We run CFD models using the code Fluent. We choose a segregated solver, steady calculation (except for low air flow speed), a $k-\omega$ turbulence model and a SIMPLEC pressure-velocity coupling.

First computations have been made with the damper located 25 m downstream the fire and a inflowing air speed of 3m/s.

3.1 Convective Heat Transfers

There are two ways to model transfers at the wall: creating a layer of new elements with the properties of concrete over every surfaces or using one of the boundary conditions available in the code.

In the first case, Fluent will only solve the energy equation in solid elements, and give a good idea of how heat propagates in the concrete. Unfortunately, this method requires the height of the mesh to be increased by at least 10 elements, which for our 105 m model, represents over 100,000 additional elements. So for a complete tunnel, this corresponds to a large increase in computational cost.

To reduce this cost, we may use Fluent's convection boundary condition. This method consist of a thin layer conduction, that can be written:

$$\varphi = h_f (T_w - T_f) = h_{ext} (T_{ext} - T_w) \quad (6)$$

where φ - conductive flux, W.m⁻²; h_f - exchange coefficient of the fluid, W.m⁻².K⁻¹; T_f - temperature of the fluid, K; h_{ext} exchange coefficient in the solide set by user, W.m⁻².K⁻¹; T_{ext} - temperature outside the solid, K and T_w the temperature at the wall, the value that interests us.

T_{ext} is set at 300 K but a preliminary study is necessary to determine the value of h_{ext} . We create a two-dimensional axi-symmetric tube of length 50m covered by 25 elements layer of "solid," and in which a 1700 K flow is injected. After a transient simulation of 40 min, only the first five elements have seen their temperature increase. Drawing the exchange coefficient at the interface of between fluid and solid gives the value : $h_{ext} = 21 \text{ W.m}^{-2}.\text{K}^{-1}$.

A third case is then described: the one where only the false ceiling is meshed and the boundary condition is set on all other side walls.

As a first approach, no radiation model is used since it is assumed that 30 % of the total heat release is directly lost to the walls.

Comparison of the results in those three cases and the case without transfers at the walls shows a difference of less than 2 % for efficiency and 1 % for yield. The rest of the study will then be carried out with adiabatic walls.

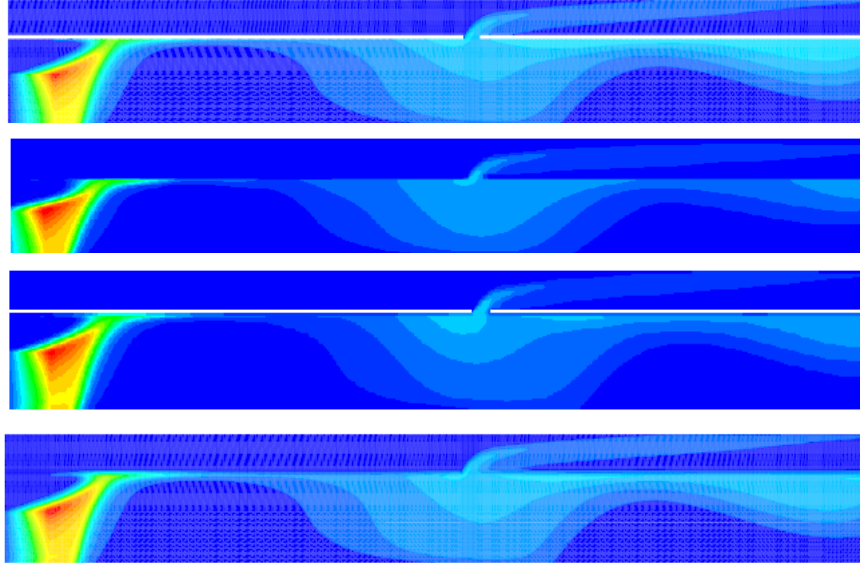


Fig 5 : Comparison of the contours of temperature on the plane of symmetry in the cases without heat transfer, with a full mesh of walls, with the boundary condition and mixed condition and mesh

3.2 Radiative Heat Transfers

In this case, the fire is at the same time a source of energy and of an imaginary gas that has the physical proprieties of CO₂, except that its absorption coefficient α_{smoke} is set to match the one of smoke. In reality, opacity is strongly dependent of the production of soot, which depends itself of the combustibles. In facts, those differ for each fire, so we will use a relation that links opacity in the tunnel to the concentration of smoke, which is also related to the volumetric fraction f_v of CO₂ and to the temperature of the fluid T by :

$$\alpha_{smoke} = 83\,000 \frac{f_{v,CO_2}}{T} \quad (7)$$

This is only true for visible wave-lengths, but it is the only reference we have got for tunnel fires (ref).

In this case, the source of energy is increased and set to 20 MW for half-domain. The source of gas to 1,35 kg/s, based on the typical creation of CO₂ in a confined fire. The walls also participate in the transfer of radiation, so we use a mixed convection and radiation boundary condition with the properties of pale concrete : emissivity $e = 0.85$.

We use the Discrete Ordinate radiation model and calculations are time dependant.

Results show that efficiency does not change much between cases with and without radiation respectively 33% and 35.2%. Yield decreases more significantly from 35.3% to 30% when radiative transfers are included. This is because a large fraction of the total energy is lost through the walls, so the power needed to extract everything is overestimated.

Nonetheless, the influence is still quite low, considering the facts that calculation time is highly increased and convergence problems appear when a radiation model is used. This is why we choose to carry on calculation without it.

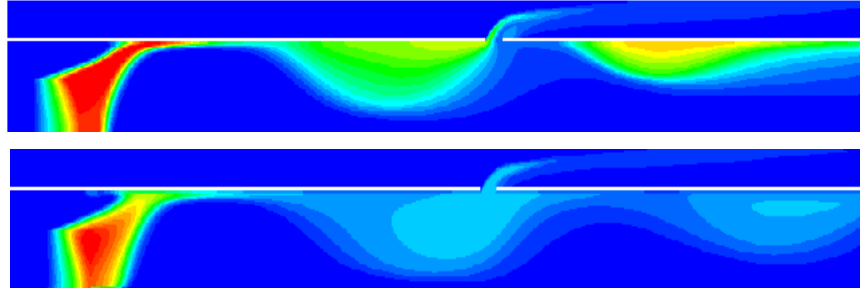


Fig 6 : Comparison of the contours of temperature on the plane of symmetry in the case without and with radiation models

4 SIMPLIFIED PROBLEM: SINGLE DAMPER

We will investigate the influence of three parameters on the flow in the tunnel and on the performance of the dampers: the air flow speed in the tunnel, the distance between the fire and the damper and the shape of the damper itself. We use the model described previously and vary each parameter separately, all others remaining constant.

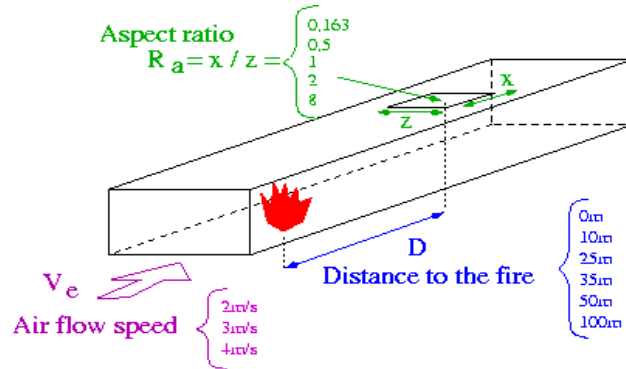


Fig 7 : Experiments planing

4.2 Influence Of The Air Flow Speed

Three inflowing air speeds are investigated in this study: 2 m/s, 3 m/s and 4 m/s. In each case, the damper is 2 m wide and 1 m long and is located 25 m downstream to the fire. Results are represented on a graph of efficiency against yield and contours of temperature on the plane of symmetry.

Both clearly show that the best results coincide with a low-velocity air flow because this helps maintain stratification, so the gases near the damper are hotter and the extracted buoyancy flux is higher. This shows why air speed control is so important.

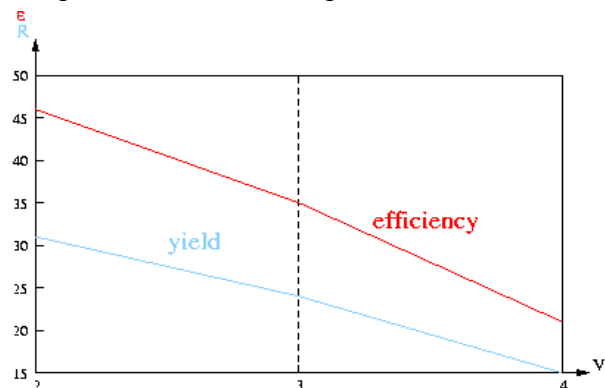


Fig. 8 : Efficiency and yield for 3 values of air flow speed

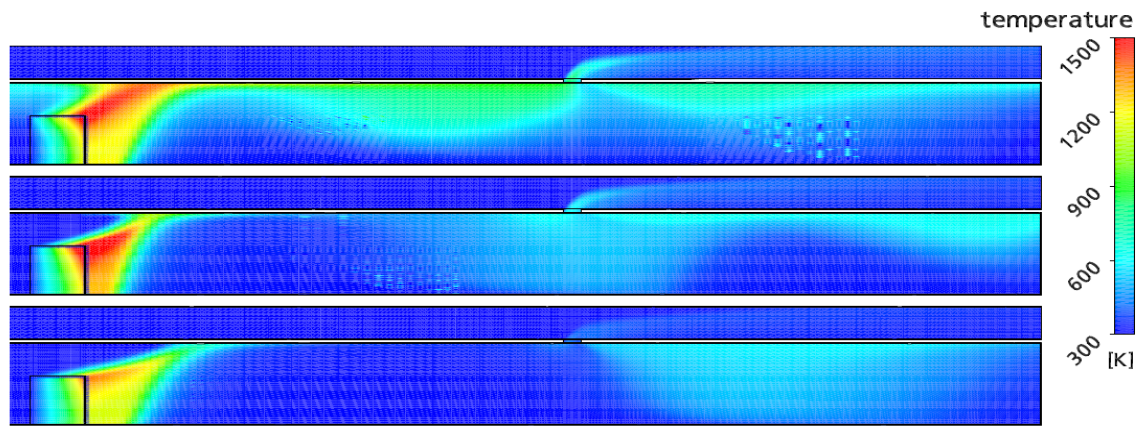


Fig 6 : Contours of temperature on the plane of symmetry for air speed of 2, 3 and 4 m/s

4.3 Influence Of The Distance Between The Damper And The Fire

Six distances between the damper and the fire are investigated here : $D = 0, 10, 25, 35, 50$ and 100 m. In each case, the damper is 2 m wide and 1 m long. The air flow velocity in the tunnel is 3 m/s.

Results are presented as before. It is shown that the best results are obtained where the smoke layer is fully developed, that is to say over 25 m from the fire. In the nearest 25 m, results are very poor, due to the three-dimensional effects upstream the fire. Over distances of 100 m, results tend to worsen due to mixing-induced cooling of the smoke.

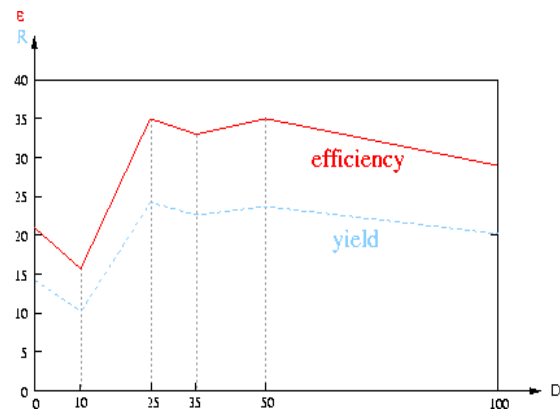


Fig 9 : Efficiency and yield for different values of D

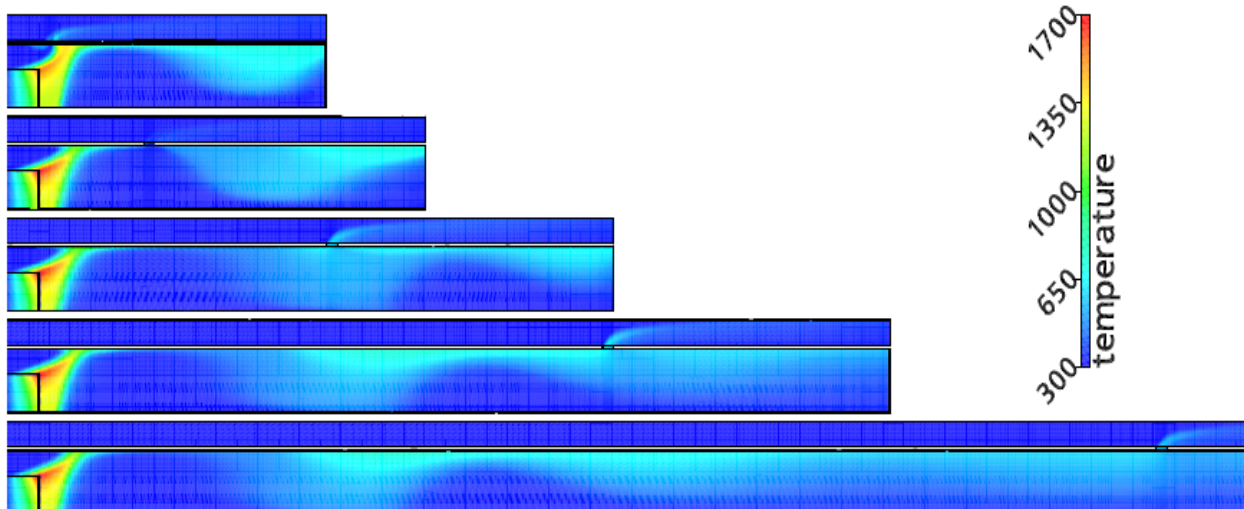


Fig 10 : Contours of temperature on the plane of symmetry for a distance of the damper to the fire of $0, 10, 25, 50$ and 100 m

3.4 Influence Of The Shape Of The Damper

Five damper shapes are investigated, each with equal opening area of 2 m². They are referenced by their aspect ratio R_a , defined as the streamwise length scaled on the width.

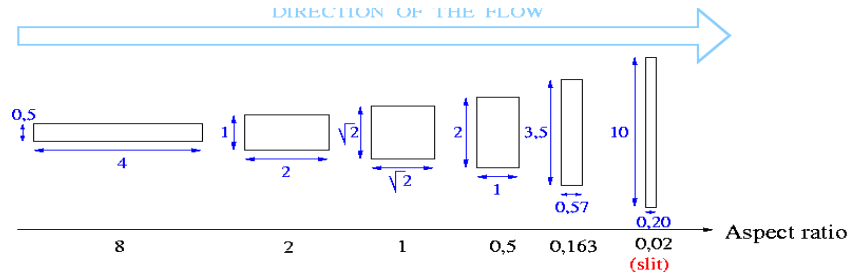


Fig 11: Representation of the damper shapes and their aspect ratios

We will begin by investigating aspect ratios of 8, 2, 1, 0.5 and 0.163 for common damper shapes. In each case, the damper is located 25 m downstream of the fire and the air flow velocity in the tunnel is 3 m/s.

We can see that, for this first set of cases, there are only small differences: less than 2%. These results have been verified for other damper locations: such as 10 m further downstream or by placing the damper away from the damper centerline.

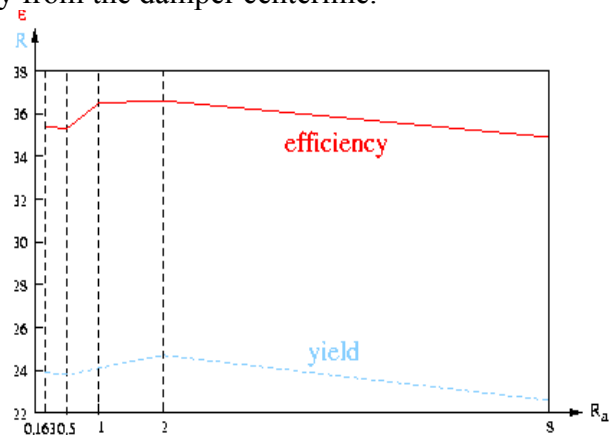


Fig 12 : Efficiency and yield for 5 usual damper shapes

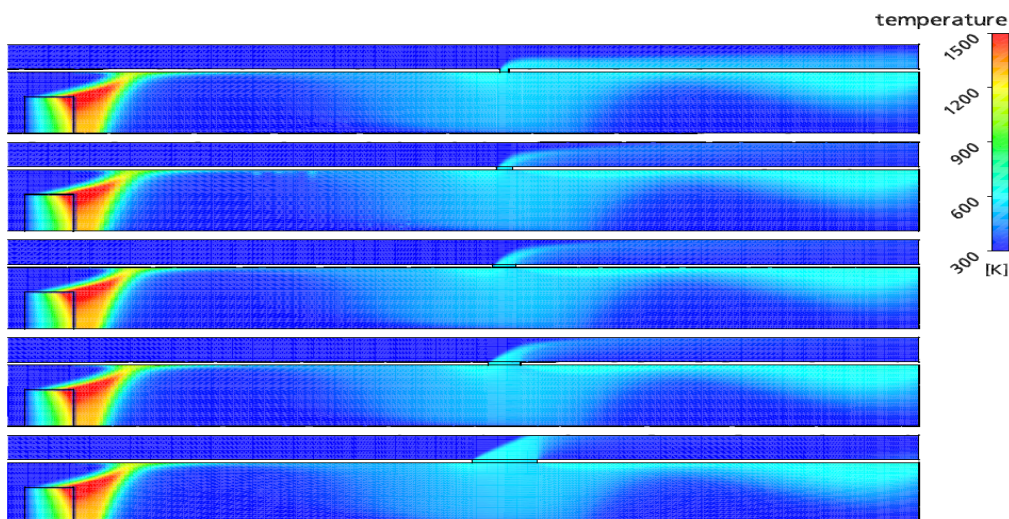


Fig 13 : Contours of temperature on the plane of symmetry for different damper shapes

However, the study of an unusual case, where the damper is as wide as the tunnel and particularly thin, like a slit, gives very different results. Efficiency increases from 35 to 60% and yield from 24 to 40%. This is because the slit acts like a barrier over the whole width of the tunnel and comparison of the streamlines shows that in this case, only the hotter smoke beneath the ceiling is withdrawn. Efficiency is then improved by reducing plug-holing.

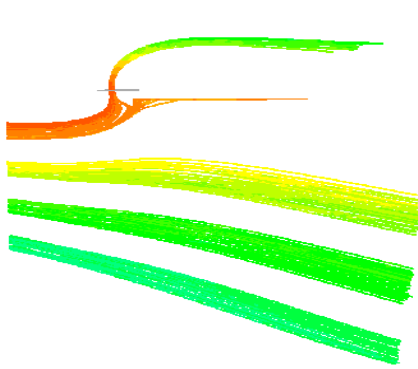


Fig 14 : Streamlines for a slit damper

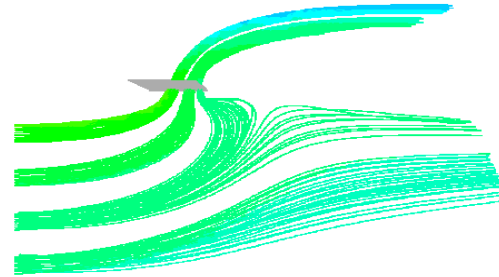


Fig 15 : Streamlines for a damper with $R_a = 0,163$

Unfortunately, this atypical shape would be very difficult to set up, so we will keep the idea that, in regular configurations, the damper shape for a given area does not significantly influence its performance.

5. TUNNEL WITH 8 DAMPERS

5.1 Description of the Case

For this case, the model is a 425 m long section of a tunnel, containing 8 dampers, located 50 m from each other. The duct is closed at one end, so that the total mass flow rate extracted through all dampers is 66 kg/s. The domain is also symmetric and numerical models are the same as those used previously. A half-domain has been meshed in excess of 2 million elements. Heat release rate of the fire in the half-domain remains 13,5 MW and we model neither radiation nor heat transfer at the walls.

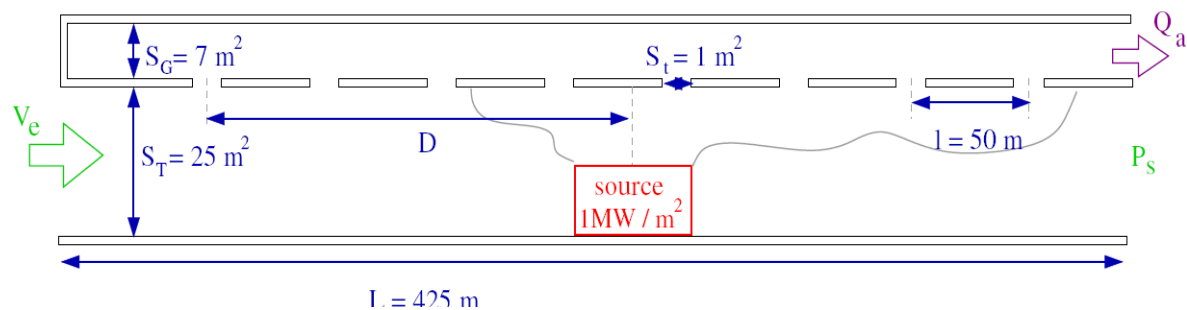


Fig 16 : Description of the 8 dampers model

Since we have seen that the shape of the dampers has no effect on the performance of the system, we will only investigate the influence of two parameters: the air flow speed and the position of the fire in the tunnel.

5.2 Influence of The Air Flow Speed

As for the case of a single damper, tests will be made with three air flow speeds, measured at the entrance of the tunnel: 2 m/s, 3 m/s and 4 m/s. Results are represented on a graph of efficiency and yield against speed and contours of temperature on the plane of symmetry.

Again, it is shown that low air velocity gives the best results, even if this time it appears that the stratification is not very well maintained.

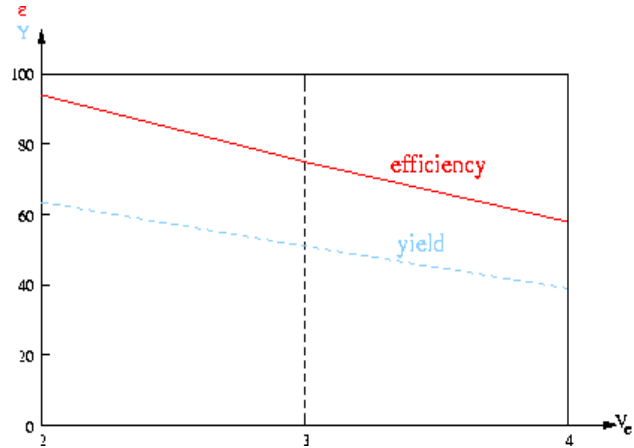


Fig 17 : Efficiency and yield for 3 air flow speeds

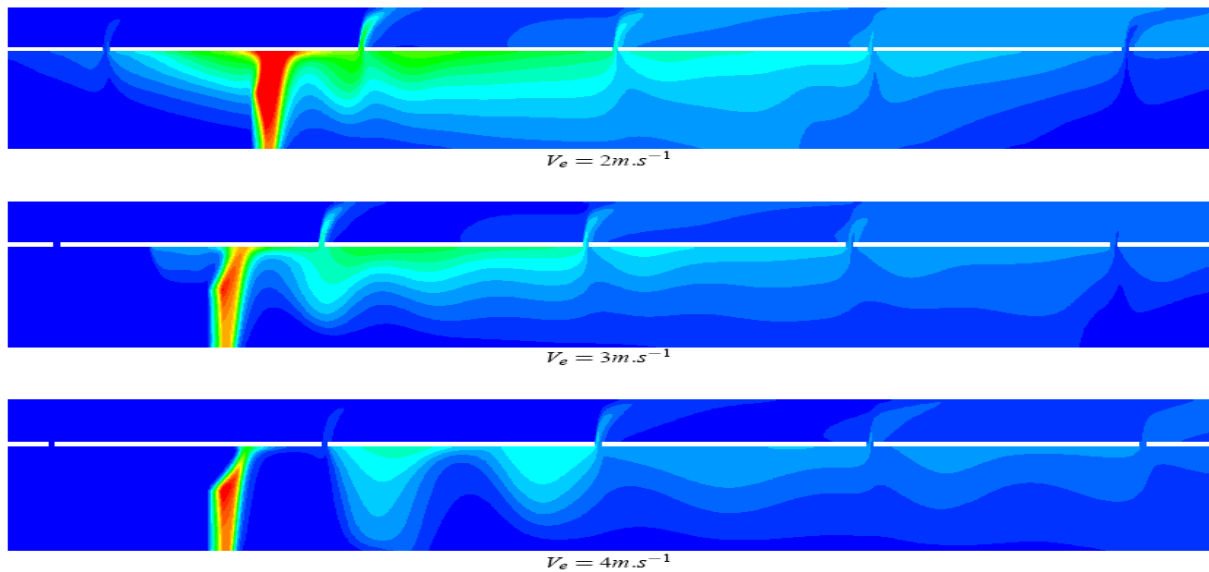


Fig 18 : Contours of temperature on the plane of symmetry for air speed of 2, 3 and 4 m/.
Image is scaled for a better viewing.

5.3 Influence of The Position of the fire

In this case, distance D to the fire is defined as the distance to the first damper. We investigate 3 different fire position: at the entrance of the tunnel between the first and the second damper ($D = 75$ m), in the middle of the tunnel between the fourth and fifth damper ($D = 175$ m) and at the end of the section between the sixth and seventh damper ($D = 325$ m). In each case, the air speed is 3 m/s.

The analysis shows that best results are obtained when the fire is located at the entrance of the section. Closer observation of the local efficiency shows that in the case of the fire at the exit of the tunnel, the last two dampers have a far higher efficiency than in the first case. This is because the longitudinal air flow in the tunnel is slowed due to mass flux in the extraction duct via the first six dampers. The behaviour of a damper is therefore highly dependent on the behaviour of the surrounding extractions.

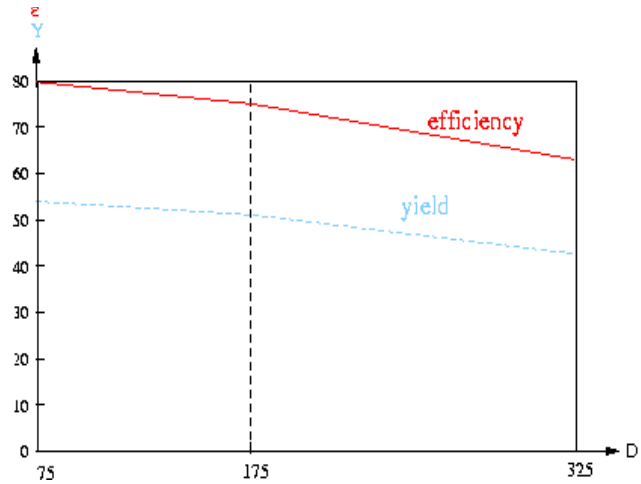


Fig 19 : Efficiency and yield for 3 positions of the fire

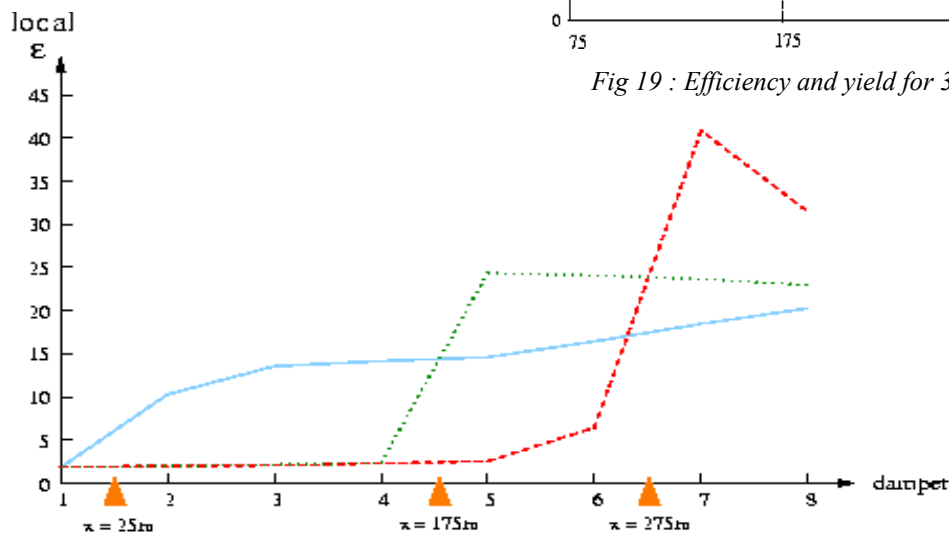


Fig 20 : Local efficiency for 3 position of the fire

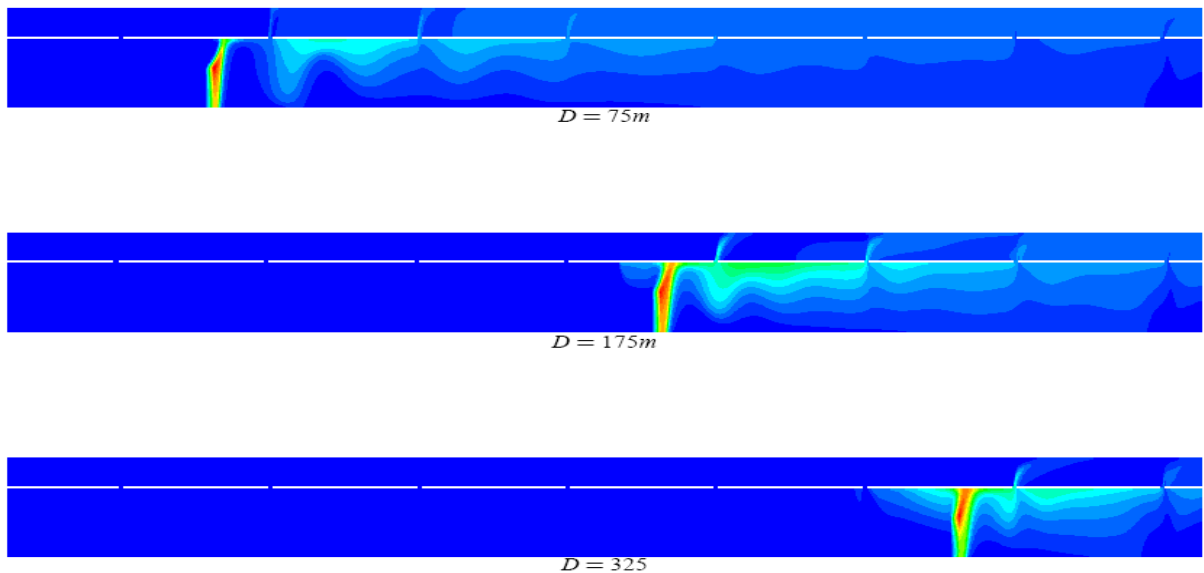


Fig 21 : Contours of temperature on the plane symmetry for 3 positions of the fire, scaled

6. CONCLUSION AND PERSPECTIVES

In order to evaluate the efficiency of a transversal ventilation system, two criteria have been defined, both based on the buoyancy fluxes. Numerical simulations have been performed and the results have led us to the conclusion that the most influential parameter is the air flow velocity. On the contrary, we found that the damper shape for a given cross-section area is not important. This is why optimising such a system should be done by controlling the inflow, rather than inventing unusual configurations of dampers.

Further work will be conducted to investigate the influence on the flow and the efficiency of other vehicles blocked in the tunnel during the fire, especially large trucks.

REFERENCES

1. Chassé, P. and Lacroix D. Etude sur Maquettes Aérauliques des Systèmes de Trappes de Désenfumage en Tunnel, Centre d'Etude des Tunnels, Technical report, 1994.
2. Danzinger, N.H. and Kennedy, W.D., Longitudinal Ventilation Analysis for the Glendwood Canyon Tunnels, Proceedings of the Fourth International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, 1982.
3. Demouge, F., Contribution à la Modélisation Numérique de la Stratification des Fumées dans le Cas d'un Incendie en Tunnel Routier, PhD These, University Claude Bernard Lyon I, 2002.
4. Fluent Inc, Fluent 6.1 User's Guide, 2003.
5. Grant, G.B., Jagger, S.F. and Lea C.J., Fires in Tunnels, The Royal Society, 356, 1998, pp 2873-2906.
6. Hinkley, P., Smoke and Heat Venting, SFPE Handbook of Fire Protection Engineering, 1995, pp. 3-160 to 3-173.
7. Ingason, H. and Werling, P., Experimental Study of Smoke Evacuation in a Model Tunnel, FAO Defense Research Establishment, Test report, 1999.
8. Mégret, O., Etude Expérimentale de la Propagation des Fumées D'incendie en Tunnel pour Différents Systèmes de Ventilation, PhD These, 1999, University of Valenciennes.
9. Ribot, B., Modélisation Numérique d'un Système de Ventilation d'un Tunnel Routier par une Trappe de Désenfumage dans le cas d'un Incendie, PhD These, 1999, University Claude Bernard Lyon I.
10. Roh, J.S., Ryou H.S., Kim, D.H., Jung, W.S. and Jang, Y.J, Critical Velocity and Burning Rate in Pool Fire during longitudinal ventilation, Tunneling and Underground Space Technology, 22, 2007, pp 262-271.
11. Spratt, D. and Heselden, A.J.M. Efficient Extraction of Smoke from a Thin Layer under the Ceiling, Fire Research Note N°1001, 1974.
12. Tien, C.L., Lee, K.Y. and Stretton A.J., Radiation Heat Transfer, SFPE Handbook of Fire Protection Engineering, 1995, pp. 1-65 to 1-79.
13. Versteeg, H.K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics, 1995, Longman Scientific and Technical Press.
14. Viollet, P.L., Mécanique des Fluides à Masse Volumique Variable, 1997, Presse de l'école Nationale des Ponts et Chaussées.
15. Wood, I.R., Selective Withdrawal From a Stably Stratified Fluid, Journal of Fluid Mechanics, 32, Part 2, 1968, pp 209-223