Haukur Ingason
Ludovic Romanov

Use of Mobile Fans in Tunnels

SP Swedish National Testing and Research Institute
SP Fire Technology
SP REPORT 2002:06

Haukur Ingason
Ludovic Romanov

Use of Mobile Fans in Tunnels

SP Swedish National Testing and Research Institute
SP Fire Technology
SP REPORT 2002:06

Abstract

Several accidents have recently occurred, revealing how deadly fires in tunnels can be. Tragic events such as the one in the Mont Blanc tunnel or the St. Gotthard tunnel showed how hard it is for the fire brigade to intervene with such fires. New equipment is now available to cope with fires in tunnels: namely, mobile fans placed at one end of the tunnel. By creating a jet-like air flow into the tunnel, which induces a fresh air flow with it, these fans can help to drive the heat and smoke in one direction, allowing fire fighters access to the fire. The air on the upstream side would become tolerable; giving fire fighters easier access to the fire source and improving their prospects of actually extinguishing the fire.

The aim of the work described here was to investigate the limitations of this technique by using a Computational Fluid Dynamic (CFD) code. First, comparison was carried out with 'cold' experiments performed in two existing tunnels in Europe; the Manesse tunnel (0.5 km long) and the Käferberg tunnel (2 km long). This was followed by a corresponding simulation of a 15 MW fire located in the middle of two trains at the centres of the Manesse tunnel and the Käferberg tunnel. Buoyancy effects due to the slope of the tunnels were considered in the CFD simulation.

Key words: mobile fan, tunnels, fire, CFD
Sökord: mobila fläktar, tunnlar, brand, CFD beräkningar

SP Sveriges Provnings- och Forskningsinstitut
SP Rapport 2002:06
ISBN 91-7848-895-8
ISSN 0284-5172
Borås 2002

SP Swedish National Testing and Research Institute
SP Report 2002:06
Postal address:
Box 857, SE-501 15 BORÅS, Sweden
Telephone: +46 33 16 50 00
Telefax: +46 33 13 55 02
E-mail: info@sp.se
Internet: www.sp.se

Contents

Preface

A consultative advisory group was established in order to ensure a high quality of the work. The group consisted of the following persons:

Anders Berqvist, Stockholm Fire Brigade
Håkan Frantzich, Technical University of Lund
Kjell Hasselrot, Fire Consultant, Stockholm
Sören Lundström, Swedish Rescue Services Agency

We gratefully acknowledge the input from the members of the advisory group.

One of the authors, Ludovic Romanov, carried out the simulations presented in this report and provided a significant contribution to the text in the report. As part of his engineering education at ENSIMEV – University of Valenciennes, in France, he did a five-month placement at SP Fire Technology. His contribution is gratefully acknowledged.

The project was financed by the Swedish Agency for Innovation Systems (VINNOVA).

Sammanfattning

Vid en insats i en rökfylld tunnel så uppstår många problem som räddningstjänsten måste lösa på plats. Röken begränsar sikten vilket kräver att andningsluft måste tas med. Räddningsledaren måste snabbt göra en bedömning över vad det är för typ av olycka och vad som håller på att hända. Därför finns det behov av att kunna påverka situationen inne i tunneln. En möjlighet är styrning av brandgaserna med hjälp av mobila fläktar. Att kunna styra rökgaser och i viss mån även värmen med mobila fläktar höjer säkerheten för räddningspersonalen vid en tunnelbrand. De beräkningar som genomförts i detta projekt visar på möjligheterna med denna typ av utrustning för räddningstjänsten. Jämförelse med Computational Fluid Dynamics (CFD) beräkningar med datorprogrammet FLUENT och experiment med mobil fläkt i tunnlar utan brand har genomförts vid två järnvägstunnlar i Zürich, Schweiz; Manessetunneln som är 523 m lång med tvärsnittsarea på 23,4 m^2, och Käferbergstunneln som är 2118 m lång med en tvärsnittsarea på 45,4 m^2. Även simuleringar med 15 MW brand i tåg placerad mitt i tunnlarna genomfördes för att undersöka inverkan av tåget och branden på fläktens effektivitet. Resultaten från simuleringarna visas i tabellen nedan.

		Manessetunnel	Käferbergtunnel
Etablering av flödet i en tunnel med mobil fläkt utan ett tågsätt och utan brand	Den slutliga reverserade lufthastigheten inne i tunneln (m/s)	3,75	2,24
	Tiden det tog att reversera luftflödet inne i tunneln (minuter)	4	10
Etablering av flödet i en tunnel med mobil fläkt med ett tågsätt men utan brand	Den slutliga reverserade lufthastigheten inne i tunneln (m/s)	3,63	2,24
	Tiden det tog att reversera luftflödet inne i tunneln (minuter)	1	3
Etablering av flödet i en tunnel med mobil fläkt med ett tågsätt och med 15 MW brand	Den slutliga reverserade lufthastigheten inne i tunneln (m/s)	2,5	2,05
	Tiden det tog att reversera luftflödet inne i tunneln (minuter)	1	3

Fläkten som användes i simuleringarna hade en luftkapacitet på 37,5 m^3/s och en diameter på 1,22 m. Den visade sig vara mycket effektiv för de scenarier som simulerades. Resultaten i tabellen visar detta entydigt. Det tog högst 10 minuter att vända luftflödet (reversera) i den långa tunneln (Käferbergtunneln) medan det tog fyra minuter i den korta tunneln (Manessetunneln). De sluthastigheter som uppnåddes var 2,24 m/s respektive 3,75 m/s. När tåget stod i tunneln så förkortades tiden det tog att vända luftflödet inne tunneln avsevärt. Istället för 10 minuter i den långa tunneln utan tåg så tog det 3 minuter med tåg och stället för 4 minuter i den korta tunneln utan tåg så tog det 1 minut i den korta tunneln. Branden påverkade inte tiden det tog att reversera flödet men däremot sluthastigheten.

Tekniken fungerar bra för den typen av tunnlar (<2 km, <45 m^2) och brandstorlekar (<15 MW) som användes här. Med fläktarna kan man styra rökgaserna och komma snabbare fram till brandhärden. De kan också vara till hjälp om de yttre faktorerna förändras, t.ex. om vinden vänder, vilket ökar säkerheten för räddningspersonalen. Men det finns begränsningar, så som tunnelns längd, brandens storlek och yttre vindförhållanden, vilket behöver undersökas vidare. Det gäller även vilken taktik man ska använda för att utnyttja fläktarna optimalt, även om resultaten ser lovande ut.

1 Introduction

The recent catastrophic fires in Europe have put the focus on problems for fire fighters dealing with these types of fires. A good understanding of fire dynamics in tunnels has become more apparent owing to these huge catastrophic fires, especially the influence of ventilation on the fire development. We know that a fire in a passenger car is no greater danger, and even a fire in a bus will not necessarily present an immediate danger to other users of the tunnel - provided, of course, that the passengers can be got out of the bus. But a fire in a heavy goods vehicle (HGV) or a tanker is a much more difficult and dangerous situation, both for other vehicles and persons in the tunnel and for the rescue services. If the fire starts to spread to other vehicles, there will be great danger to persons inside the tunnel. Most of those who die in tunnel fires are killed by the poisonous gases that quickly spread through the tunnel, as the gases cannot disperse in the same way as from a fire in the open.

There are several factors that have played a major part in the growth of recent tunnel fires in Europe. First and foremost, there is the high fire load represented by many heavy goods vehicles involved. Fires in flammable goods in the load develop very quickly. Ventilation spreads the fire between vehicles, and the rapid evolution of smoke surprises drivers who do not get out in time, or who cannot find their way out. The rescue services have great difficulty in reaching the fire: vision is obscured by smoke, and the high heat levels prevent fire fighters from getting to the fire, even when the smoke has been ventilated away. A new type of equipment is now being used to deal with the fires in tunnels: mobile fans placed at one end of the tunnel, see Figure 1. The fan creates a longitudinal flow inside the tunnel and pushes the heat and the smoke downstream of the fire. Thus the fresh air on the upstream side becomes more tolerable, and the fire fighters have better access to the heart of the fire and consequently are more likely to extinguish it. This technique has been applied in many large-scale experiments in recent years [1]. Unfortunately, none of these tests has used a real fire, and so there is a need to validate the technique in tunnels with fires. Instead of large-scale testing, we have used Computational Fluid Dynamics (CFD), using the FLUENT 5.0 program [2].

In this project we have been mainly concerned with the flow that a fan can create inside a tunnel, and how long time it takes to establish this flow. No tactical aspect of fighting fires using this technique has been considered. First, we will compare experimental and computational results for two tunnels of different size, which will also allow us to evaluate the accuracy of the calculation. This will be followed by simulation of fires inside the two tunnels we simulated previously, with the fan being started in order to see whether it is possible to reverse the flow created by the fire and how long it takes. The size of the fire was taken to be 15 MW, which is the average maximum heat release rate obtained in a large-scale tests of a train coach [3].

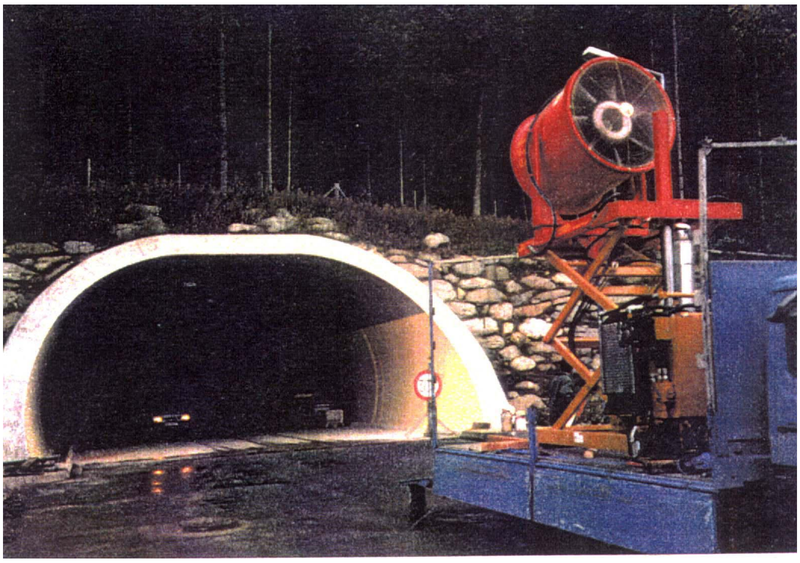

Figure 1 A mobile fan placed outside the entrance of a road tunnel.

Figure 2 The position of the mobile fan unit can be quickly changed, which makes it easier to find the most effective position.

2 CFD simulations of mobile fans used in tunnels without a fire

Numerous tests have been carried out in different large tunnels in order to demonstrate the possibility of using mobile fans in tunnels [1]. Figure 2 shows the same type of equipment as used in the large tunnel tests described here. The aim of the tests was, firstly, to measure the flow that a fan could create inside a tunnel and, secondly, to see how long it takes for the flow to be established. No fires were used in these tests. The tests provide good experimental data, which can be used to investigate the accuracy of CFD programmes for this type of application. The CFD code used in the simulation was FLUENT 5.0 [2]. We decided to use two of these tests for comparison with FLUENT. One test took place in the Käferberg tunnel, which is a railway tunnel (45 m^2) about 2 km long, and the second took place in the Manesse tunnel, which is a smaller railway tunnel (23 m^2), about 0.5 km long. Both these tunnels are in Zurich, Switzerland. We chose them since they have different cross-sectional areas and lengths.

2.1 The calculation domain

The computational geometry was first created with Gambit [4] software. For both tunnels, the calculation domain consisted of the tunnel itself with, on one end that was considered as the entrance, a large parallelepiped. This parallelepiped was split into two volumes by a face parallel to the section of the tunnel, containing a circular element used to simulate the fan itself. This face was split with another face representing the section of the tunnel. This was necessary to allow a correct meshing of the parallelepiped with structured hexahedral elements. This basic grid scheme could be used here since the geometry is fairly simple. Moreover, this problem required only a relatively coarse grid. Figure 3 shows the calculation domain with the mesh at the entrance of the Käferberg tunnel.

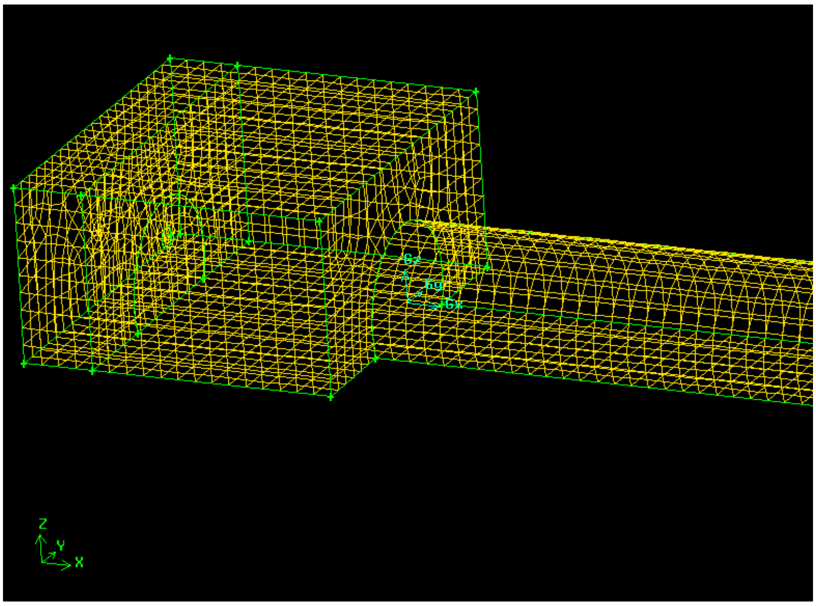

Figure 3 The calculation domain for the fan in front of the tunnel entrance of the Käferberg tunnel. The lines indicate the volume element used in the calculation.

The exit of the tunnel and some of the faces of the parallelepiped were defined as the outlet vent, and the atmospheric conditions were set to be the same as in the tests. The roughness of the walls and the roof is not known, but since the walls were relatively smooth we assumed 1 cm high protuberances (friction). The roughness of for the ground was set to 10 cm. To simulate the fan, the 'fan' boundary condition was chosen. It was found that the 'intake fan' boundary condition in FLUENT was best suited for the simulation: see Appendix A. The 'intake fan' boundary condition generates only a straight flow, with very little expansion, whereas the 'fan' boundary condition generates a flow that has the shape of a cone and its expansion can be set as required. This means that the flow is defined not only by its direction, the fan hub radius and a pressure rise, but also by radial and tangential velocities. Although more parameters have to be set, it undoubtedly gives better results. The turbulence model chosen in FLUENT was the standard k-ε model [2].

2.2 Comparison with a test in the Käferberg tunnel

The Käferberg tunnel in Zurich is a twin-track railway tunnel. It is 2118 m long, with a cross-sectional area of 45,43 m². The entrance and exit altitudes are 420 m and 445 m respectively. During the test, the temperature was 13,3 °C, the atmospheric pressure was 965,3 hPa and there was no wind outside the tunnel. The measuring points were located 150 m from the entrance (Point 1) and 150 m from the exit (Point 2). Both were positioned 2 m above the ground, and were located on the centreline of the tunnel. As shown in Figure 4, a mobile fan was placed in front of the entrance, at a distance of 14 m (A). It was mounted 3,85 m above the ground (H), tilted 7° down (α) and 15° to the right (β). The diameter of the fan was 1,22 m, and its hub radius was 0,22 m. It delivered approximately 37,5 m³/s (135 000 m³/h). The simulated fan was given the same position and direction as in the test.

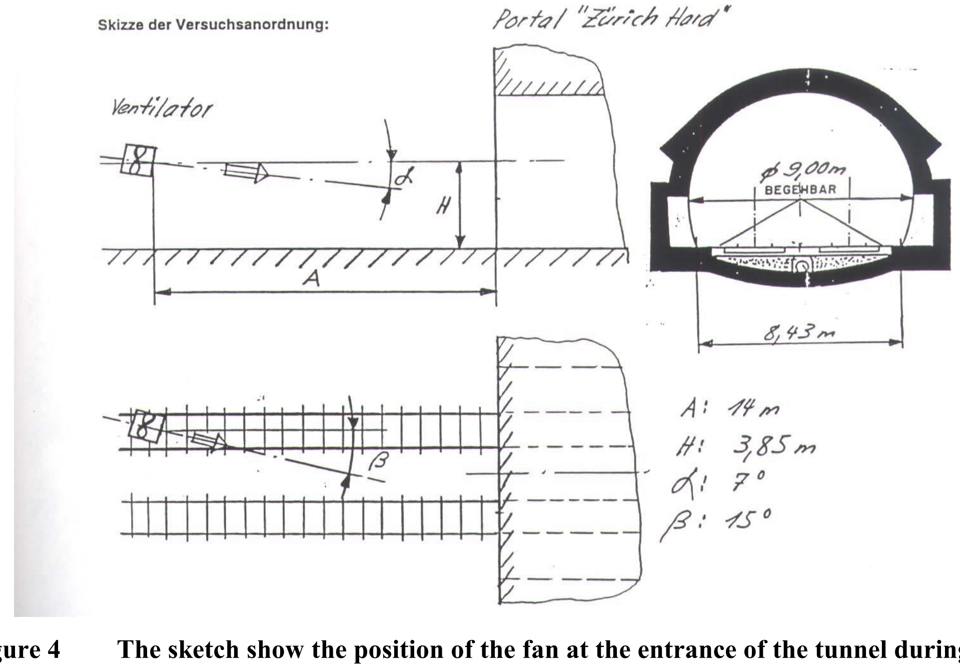

Figure 4 The sketch show the position of the fan at the entrance of the tunnel during the test. It also show the cross-section of the Käferberg tunnel.

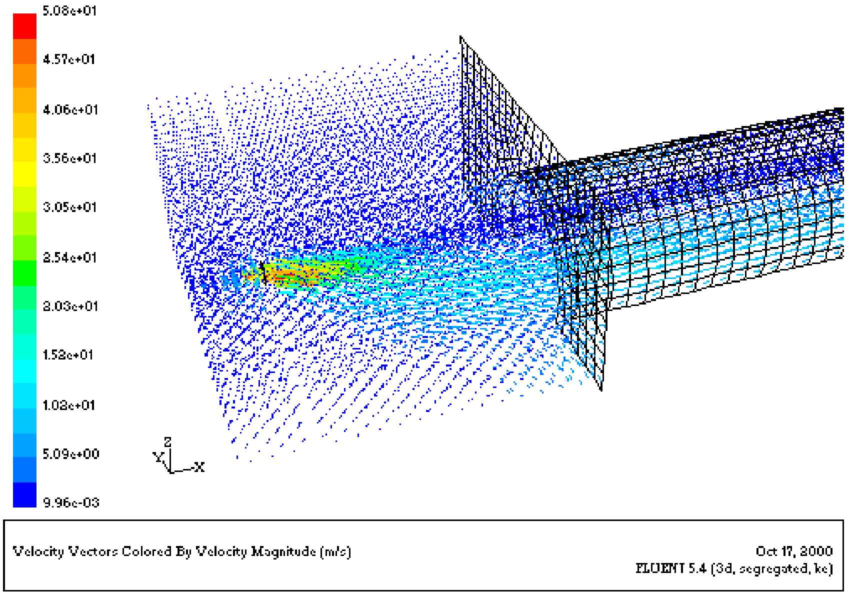

Figure 5 The figure show the velocity vectors obtained in front of the entrance.

Figure 5 shows the velocity vectors obtained at the entrance of the tunnel. The legend on the left shows the magnitude of the velocity. Relatively high velocities are obtained just behind the fan outlet, in the range of 25 – 35 m/s, whereas the velocities just in front of the tunnel entrance are about 1 - 10 m/s. The reduction in velocity is due to expansion of the cone and increased volume flow (entrained [injected] air in the air cone). Figure 6 shows the results of the calculated (transient) velocity at the two measuring points 150 m from the openings.

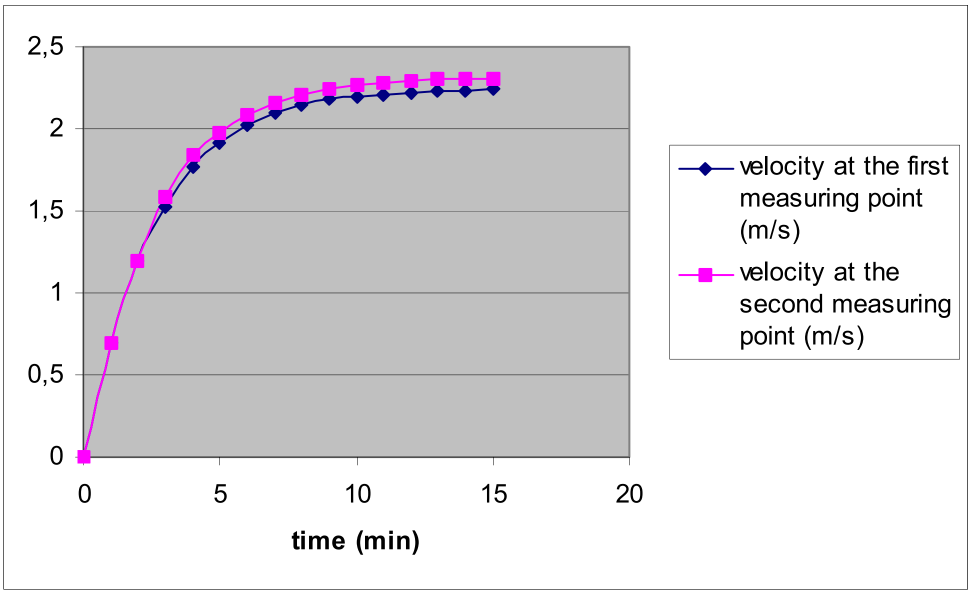

Figure 6 The calculated data for the two measuring points in the Käferberg tunnel.

Now we can compare these results with the experimental data. Figure 7 shows the measured velocities at Point 1 (black marks) and Point 2 (white marks).

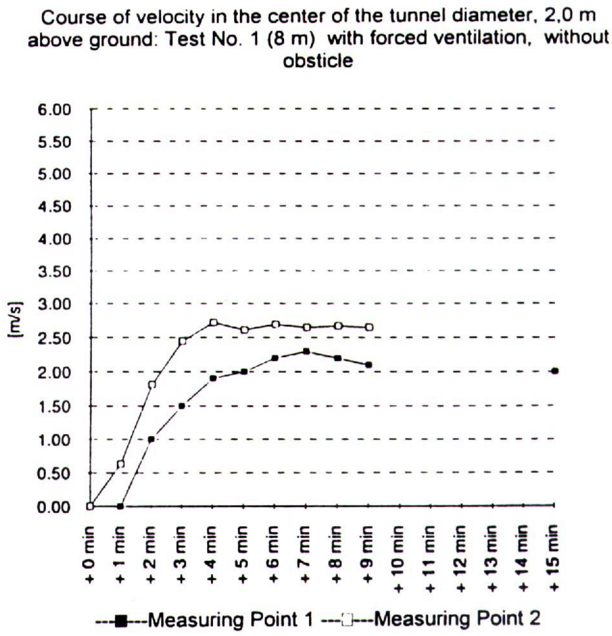

Figure 7 **The measured velocity at the two measuring stations. The black marks are at measuring Point 1, 150 m from the entrance, and the white marks are 150 m from the exit opening.**

The measured results for the two measuring points are quite different. The velocity measured near the entrance is about 25 % lower than the velocity near the exit of the tunnel. The FLUENT simulation showed the same tendency, although the difference was much lower (see Figure 6). It is difficult to draw any definite conclusions about the experimental results, as we have only limited information on the quality of the measurements. We know, however, that the energy loss along the tunnel is dominated by the friction loss, and that the air mass flow rate is conserved at both measuring stations. We do not know what the gas temperatures were inside the tunnel, but the temperature at the entrance station was probably slightly higher than that at the exit station (cooling effect of the tunnel walls). This would mean that we should expect higher velocities at the entrance, provided that the velocity at the measuring station represented the average velocity. This was apparently not the case, and therefore one would expect the reason for this discrepancy to be related to the different flow situations at both measuring stations. The only possible explanation is therefore that the fan creates some effect, such as an acceleration of the swirl, which FLUENT is not able to simulate.

Nevertheless, it is important to note that the velocities calculated with FLUENT appears to be quite close to the experimental data. It takes the same time, about 10 minutes, to reach steady-state flow inside the tunnel, and the air velocity speed is about 2,25 m/s according to the simulation, which is exactly the same as the average of the measured values. This indicates that the fan simulations with FLUENT give reasonably accurate results and therefore should be acceptable for this type of tunnel.

In the tests, the velocity profiles across the tunnel width were measured at the two measuring stations, and so the measured and calculated results will be compared at corresponding measuring points. Figure 8 shows the calculated air velocity profile across the tunnel 150 m from the entrance and 1.6 m above the ground, while Figure 9 shows corresponding measurements 150 m from the tunnel exit.

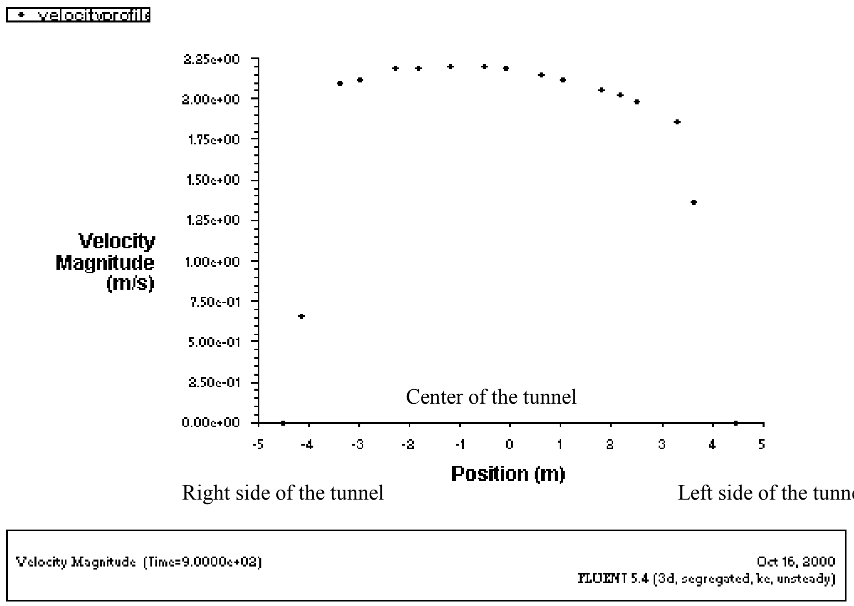

Figure 8 The calculated air velocity across the tunnel 1.6 m above ground and at the measuring station 150 m from the entrance.

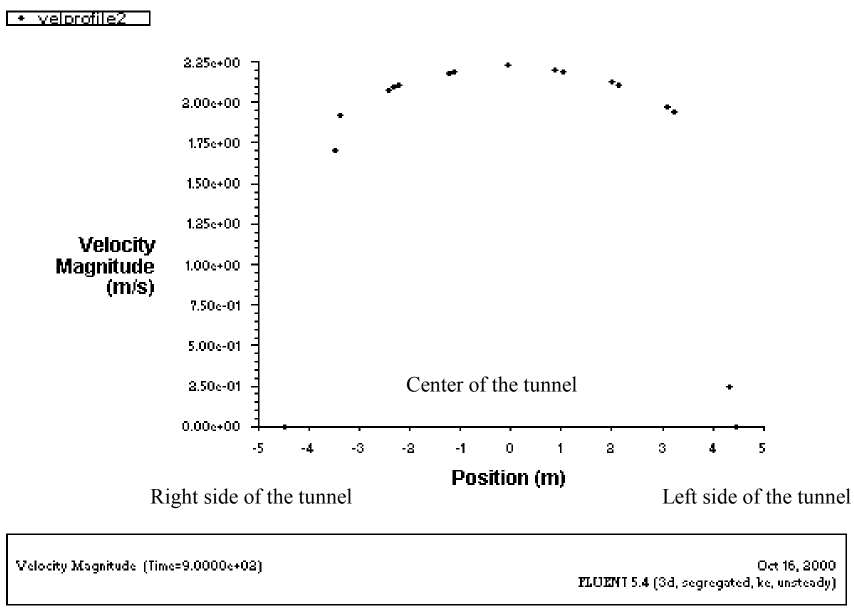

Figure 9 The calculated air velocity across the tunnel 1.6 m above ground and at the measuring station 150 m from the exit.

Figure 8 shows that the airflow is more concentrated on the right-hand side, since the fan was tilted in this direction. This could explain why the velocity is slightly higher on this side of the tunnel. However, near the exit of the tunnel, the flow has become more developed and is nearly symmetrical at the exit measuring station.

The experimental results at the exit measuring station are shown in Figure 10. We can note that, contrary to what we observed with FLUENT, the velocity is still slightly higher on the right-hand side of the tunnel, even near the exit, which indicates that we do not have fully developed 'tube flow'. The measured air velocity on the 'inner half profile section' side is very similar to that obtained in Figure 9.

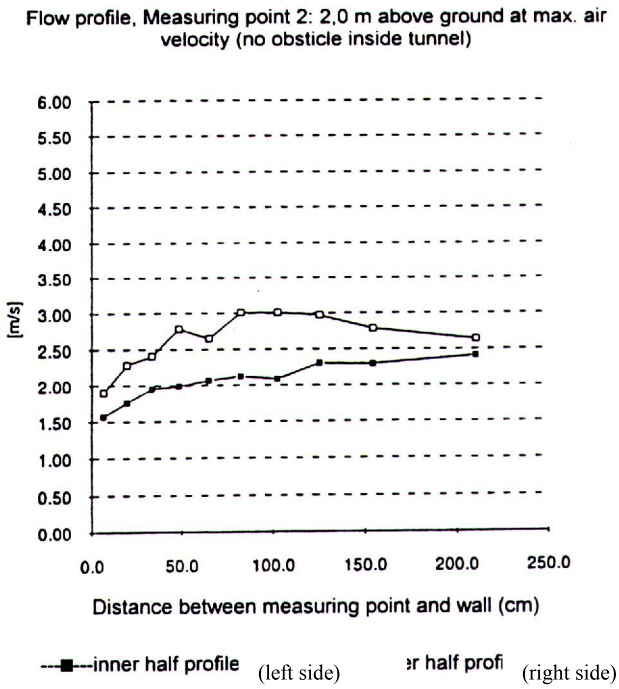

Figure 10 The measured air velocity at the measuring station 150 m from the exit.

2.3 Comparison with a test in the Manesse tunnel

The Manesse tunnel is a single-track railway tunnel. It is 523 m long with a cross-sectional area of 23,44 m². The altitude varies from 406 m to 411 m. During the experiment, the temperature was 13,1 °C, the pressure was 963,5 hPa and there was no wind. The two measuring stations were situated 150 m from the entrance (Point 1) and 100 m inside the exit (Point 2). Both were positioned 2 m above the ground, and were located on the centreline of the tunnel. As shown in Figure 11, a mobile fan was positioned in front of the entrance, at a distance of 8 m (A). It was mounted 3,25 m above the ground (H), tilted 7° down (α) and 5° to the right (β). The diameter of the fan was 1,22 m, and its hub radius was 0,22 m. It delivered approximately 37,5 m³/s (135 000 m³/h). The simulated fan was given the same position and direction as in the test.

The difference of altitude was not taken into account in the simulation, for the same reason as previously (no buoyancy). As for the Käferberg tunnel, the Manesse tunnel simulation did not require a very fine grid, as it is much smaller. Only 14 402 cells were needed in order to reach reasonable solution.

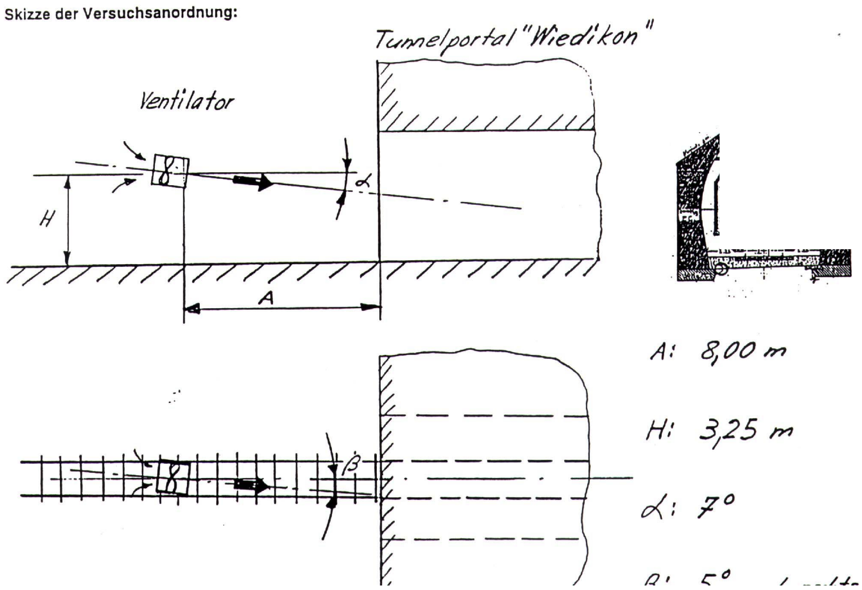

Figure 11 **The sketch show the position of the fan at the entrance of the tunnel during the test. It also show the cross-section of the Manesse tunnel.**

Figure 12 shows the results of the calculated (transient) air velocity at the two measuring points: Point 1, 150 m from the entrance, and Point 2, 100 m from the exit.

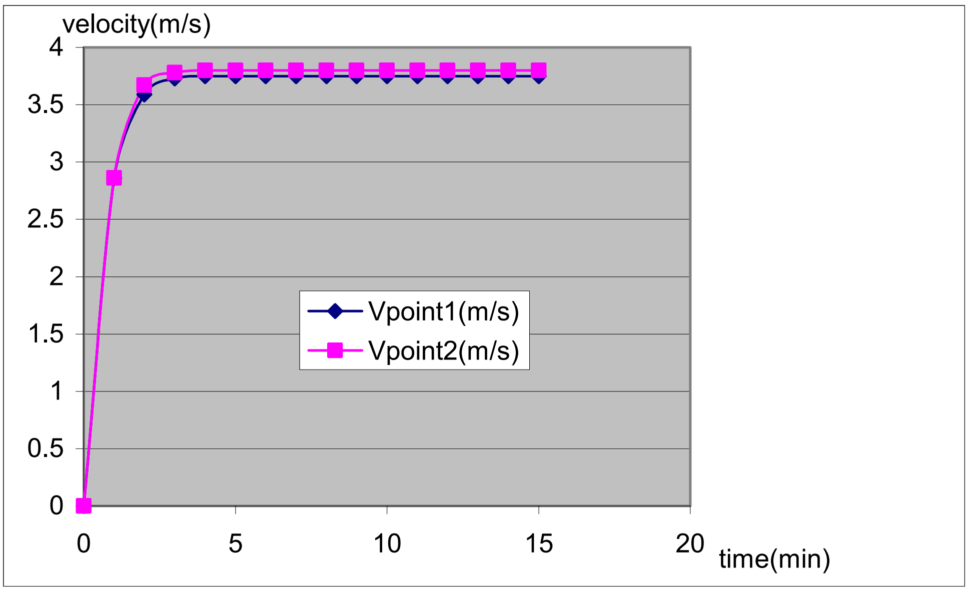

Figure 12 **The calculated air velocity at the two measuring stations in the Manesse tunnel.**

Figure 13 shows the measured air velocity 150 m from the entrance (Point 1) and 100 m from the exit (Point 2). The velocities measured at Point 1 and Point 2 are different, whereas the values obtained with FLUENT are very close to each other. Since we cannot determine precisely which factor caused this discrepancy for the measured values, we will compare the results only qualitatively. The values calculated with FLUENT at the first measuring point are quite close to the experimental data. In general, we can say that, qualitatively, the results appear to be reasonable. The flow was established within 3 to 4 minutes, and the air velocity reached by the flow inside the tunnel is somewhere between 4 m/s and 5.5 m/s, according to the experiment, and slightly less than 3.75 m/s according to the simulation. Despite this discrepancy, the results are encouraging due to the fact that the flow situation at the entrance is quite complicated.

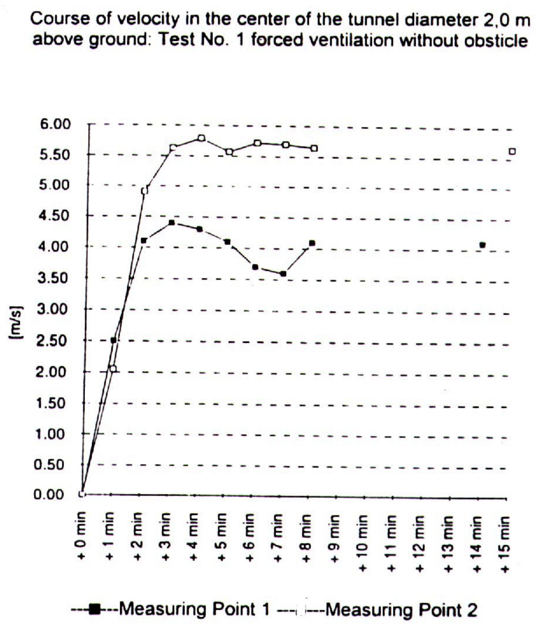

Figure 13 The measured air velocity at measuring Point 1 (150 m from entrance) and Point 2 (100 m from exit).

2.4 Simple theoretical model

It is of interest to compare the FLUENT simulation with a simple theoretical treatment of the problem. Here we develop a simple lumped dynamic flow model, which calculates the average velocity within the tunnel as a function of time. If we assume the air velocity at time t = 0 is equal to zero, and the temperature in the tunnel is at ambient conditions, we can establish a basic differential equation describing how the air velocity within the tunnel will change with time if we have an external wind source at the entrance of the tunnel. Thus we obtain, with aid of a force balance (N/m^2), the following equation:

$$\rho_0 L \frac{du}{dt} + \frac{1}{2}\rho_0 u^2 (1 + k_L + f_D \frac{L}{D_h}) = \frac{1}{2}\rho_0 U_e^2 \qquad (1)$$

where ρ_0 is the ambient density (kg/m^3), L is the length of the tunnel (m), u is the average air velocity in the tunnel, k_L is the entrance loss coefficient, f_D is the Darcy friction factor, D_h is the hydraulic diameter of the tunnel and U_e is the wind velocity at the entrance of the tunnel. This equation can easily be solved if we know the entrance wind velocity U_e created by the fan.

We assume that the air velocity U_e can be derived with aid of the equation for conservation of momentum for the air cone between the fan and the tunnel entrance. The static pressure along the air cone is assumed to be uniform. Air is drawn into the air cone (entrainment of air) and the mass flux of air, $\dot{m}$, will increase with the distance from the fan. The momentum flux will be conserved along the air cone since we assume uniform static pressure within the air cone. Consequently, the momentum flux at the outlet of the fan will be the same as that at the tunnel entrance:

$$\dot{m}_0 u_0 = \dot{m}_k U_e \qquad (2)$$

where $\dot{m}_0 = \rho_0 u_0 \frac{\pi D_0^2}{4}$ and $\dot{m}_k = \rho_0 U_e \frac{\pi D_e^2}{4}$. Here ρ_0 is the ambient density of air, D_0 is the diameter of the fan, D_e is the effective diameter of the air cone at the tunnel entrance (and is sufficient to cover the entire opening) and u_0 is the velocity of the air leaving the fan. Now we can determine the velocity at the entrance opening from Equation (2):

$$U_e = \frac{u_0 D_0}{D_e} \qquad (3)$$

The diameter of the air cone is the same as whichever is the higher of the height (H) or width (B) of the entrance opening. In most cases, it is the height that determines the necessary size, and so $D_e = H$. The primary air flow, Q_p, from the fan can be calculated from the following equation:

$$Q_p = u_0 A_0 = u_0 \frac{\pi D_0^2}{4} \qquad (4)$$

From equations (2), (3) and (4), we can obtain a relationship between the dynamic pressure at the entrance of the tunnel, $\frac{1}{2}\rho_0 U_e^2$ in equation (1), and the primary air flow from the fan, Q_p, the diameter of the fan, D_0, and the height H of the tunnel opening (or its width, B, if this is greater than H):

$$\frac{1}{2}\rho_0 U_e^2 = 8\rho_0 \left(\frac{Q_p}{\pi D_0 H}\right)^2 \qquad (5)$$

Thus, with the aid of Equations (1) and (5), we get the differential equation:

$$\frac{du}{dt} + \frac{1}{2L}(1 + k_L + f_D \frac{L}{D_h})u^2 = \frac{8}{L}\left(\frac{Q_p}{\pi D_0 H}\right)^2 \qquad (6)$$

As u(0) = 0, we can solve this differential equation, giving:

$$u(t) = \frac{2\phi}{\beta} Tanh\left[\frac{\phi\beta}{L} t\right] \quad (7)$$

where $\beta = \sqrt{1 + k_L + f_D \frac{L}{D_h}}$ and $\phi = \frac{2Q_p}{\pi D_0 H}$ (m/s). The function $Tanh(x)$ can also be written as $\frac{e^x - e^{-x}}{e^x + e^{-x}}$. Equation (7) can be used to study the effects of different parameters on the air velocity within the tunnel. For example, the maximum velocity (steady-state) obtained for a specific fan and tunnel will be:

$$u_{\max} = \frac{2\phi}{\beta} \quad (8)$$

as $Tanh(x) -> 1$ as the time increases. The time it will take is governed by the quotient $\eta = \frac{\phi\beta}{L}$ (s^{-1}). Increased η means that it will take a shorter time to reach steady-state air flow within the tunnel. As would be expected, this shows that the primary flow capacity of the fan and the geometry of the tunnel are the most important parameters. Increased capacity increases the maximum velocity and shortens the time required to reach steady-state conditions for a tunnel with the same cross-sectional area. Long tunnels, however, reduce the maximum velocity and it takes longer time to reach steady-state conditions.

From Equation (7), we can calculate the time to reach nearly fully developed flow, i.e. the time to reach a certain ratio of the steady-state velocity (u_{max}). In the example given here we chose to define the ratio which determine the degree of established flow as 95% of u_{max} . Thus, by using Equation (7) and assuming that $\frac{u(t)}{u_{\max}} = Tanh(\eta t) = \frac{95}{100}$ we can easily find the time t_{95};

$$t_{95} = \frac{1.83}{\eta} \quad (9)$$

where 1.83 was derived from the following expression; $\frac{1}{2} Ln\left(\frac{1 + n/100}{1 - n/100}\right)$ and n = 95 % (n can be easily changed). Now we can make some calculations using Equations (8) and (9), plotting the results using equation (7).

First we take the Käfenberg tunnel, for which the following data will be used: Q_p=37.5 m3/s, D_0=1.22 m, L=2118 m, H=6 m, B=8 m, D_h=4 A/P i.e. 4 * 6 * 8/(2*6+2*8)=6.86 m (A = cross-sectional area and P is perimeter of the tunnel), f_D = 0.025 (rough surface in a duct) and k_L = 0.5. This gives β = 3.037, ϕ = 3.261 m/s, η = 0.0047 s^{-1}. Equation (8) gives u_{max}=2.15 m/s and Equation (9) gives t_{95} = 389 s or 6.5 min. This means that already after 6.5 minutes we have obtained 95 % of the steady state velocity. Figure 14 is a comparison of the air velocity as calculated by FLUENT, and the air velocity determined by Equation (7).

Now we can make a corresponding calculation for the Manesse tunnel, using the following data: Q_p = 37.5 m³/s, D_0 = 1.22 m, L = 523 m, H = 4.6 m, B = 5 m, D_h = 4 A/P i.e. 4 * 4.6 * 5/(2*4.6 + 2*5) = 4.79 m, f_D = 0.025 (rough surface in a duct) and k_L = 0.5. This gives β = 2.057, ϕ = 4.254 m/s, η = 0.0167 s^{-1}, while Equation (8) gives u_{max} = 4.14 m/s and Equation (9) gives t_{95} = 110 s or 1.83 min. Figure 14 is a comparison of the air velocity as calculated by FLUENT, and the air velocity determined by Equation (7).

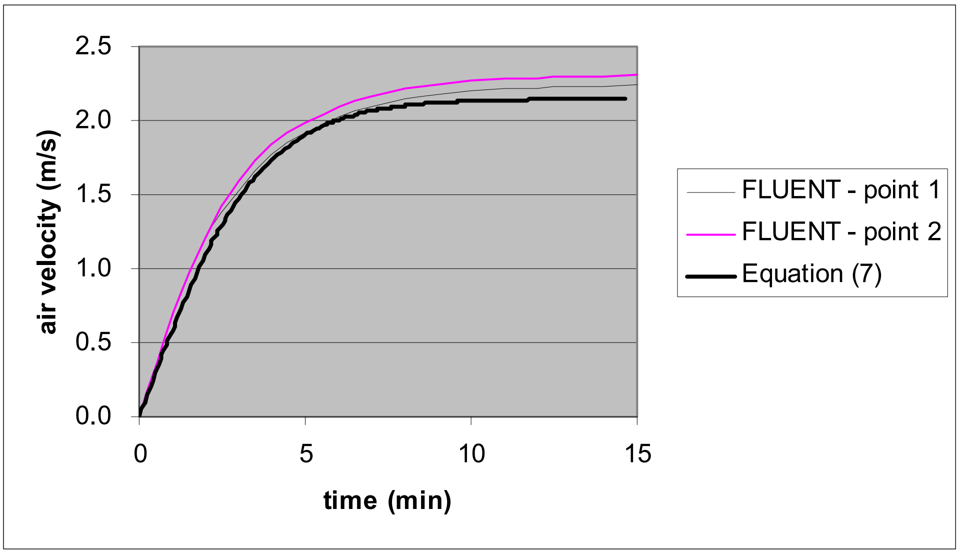

Figure 14 Comparison between calculated air velocities for the Käferberg tunnel.

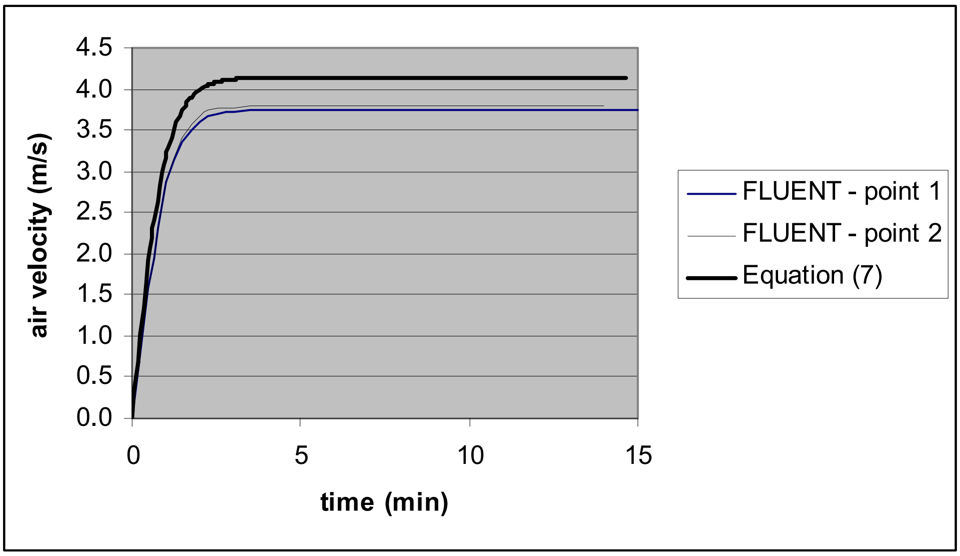

Figure 15 Comparison between calculated air velocities for the Manesse tunnel.

2.5 Discussion of the results

The flow in the Manesse tunnel induced by the fan was established after about four minutes, which is about 60 % less than the time needed in Käferberg tunnel (about ten minutes). The Käferberg tunnel is a much larger tunnel than the Manesse tunnel, with cross-sectional area and length respectively about twice and four times as large as those for the Manesse tunnel. The air velocity reached by the flow in the Manesse tunnel is about 4 m/s, whereas it was only about 2,25 m/s (i.e. 57 % less) in the Käferberg tunnel. Consequently, the larger the tunnel is, the longer it takes to establish the flow. However, the time needed to establish the flow is not proportional to the length of the tunnel, as otherwise it would have taken almost 16 minutes to achieve a steady flow in the Käferberg tunnel. With aid of Equation (9), we can show the influence of the tunnel length on the time to reach nearly fully developed flow (95 % of u_{max}) for different cross-sectional areas but the same type of fan. Figure 16 shows these effects on the time to reach a 95 % developed flow for two different cross-sectional areas.

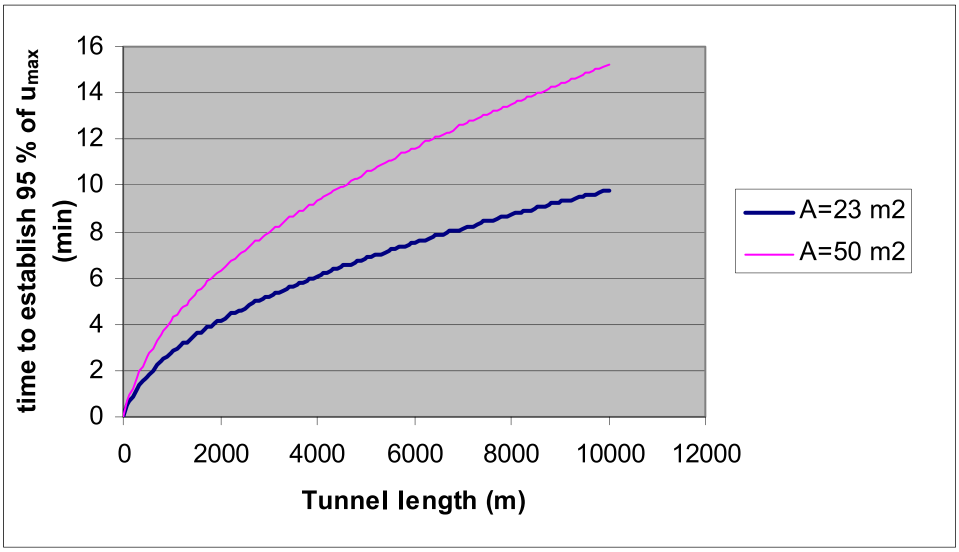

Figure 16 The influence of the tunnel length on the time to reach 95% u_{max} for two cross-sectional areas (Q_p = 37,5 m^3/s, D_0 = 1,22 m).

It is obvious from Figure 16 that the size (length and the cross-sectional area) is an important parameter. It can also be shown that the capacity of the fan and its diameter are important parameters.

The longitudinal air velocities created by a fan placed at the entrance of a tunnel and providing about 37.5 m^3/s should be strong enough to reverse the flow induced by a fire in tunnels such as the Manesse tunnel and the Käferberg tunnel. Usually, we need about 1,5 – 3 m/s to control the smoke spread upstream of the fire. The problem is that the time required to establish a steady flow inside a tunnel is obviously closely linked to the size of the tunnel, see Figure 16. It may therefore still be difficult to try quickly to extinguish fires inside very long and wide tunnels. However, these calculations show that the results provided by FLUENT and Equation (7) are reasonable when compared with the experimental data. This is true at least in situations where there is no combustion. The next step is to involve the fire in the CFD simulations.

3 CFD simulation of a mobile fan used in a tunnel with a fire

Let us now investigate what happens when a mobile fan is used when there is a fire in the tunnel. The aim is to see how long it takes to push all the heat and the smoke away in order to make access to the seat of the fire easier for the fire fighters. We will continue to use the tunnels analysed earlier (Käferberg and Manesse). The following simulations assume a train unit inside each tunnel in order to create an obstacle to the flow. In a real accident in a railway tunnel, the effects of the train unit on the flow may be considerable.

We start by calculating the flow that a fan can create inside the tunnel without a fire, to evaluate the effect of the obstacle on the time needed to establish the flow, and on the maximum velocities obtained within the tunnel. This is followed by investigating the effect of a 15 MW fire at the middle of each tunnel. This corresponds to a fully developed fire in a train wagon [3]. The fire will be located at the centre of the train. Finally we study the effect of the fan on this flow, starting the fan is started when the flow due to the fire is fully established. The calculation of the fire was carried out using the steady state option in FLUENT to obtain the established flow. Then the fan was turned on and the calculation was performed with the 'transient' option in order to see how long it takes to reverse the flow.

Moreover, contrary to the previous calculations, the difference of altitude between the entrance and exit of each tunnel was taken into account. This has a significant effect on the buoyancy flow created by the fire. Further, it is the worse case scenario that is investigated, i.e. with the fan at the higher end of the tunnel, since the hot air naturally tends to go upwards. By doing this, we expect it to be more difficult to reverse the flow.

3.1 Simulation of the Manesse tunnel using fire

A fire was placed in the middle of the tunnel, which is 523 m long, and in the middle of a 100 m train (12 m^2). Part of the train unit was subtracted from the domain and replaced by a kerosene pool fire measuring 5 m x 3 m. The fan was located at the entrance of the tunnel, which is 6 m higher than the exit. The simulation was carried out with FLUENT. As earlier, the k-ε model with the 'full buoyancy effects' option was used for the turbulence effects, and the Rosseland model was used for the radiation. Convection and conduction through the walls were taken into account, as well as the roughness of the walls and of the ground. The Eddy dissipation model was used for the chemical reaction. The thermal capacities of the different components were defined as polynomial functions of temperature, while their thermal conductivity and viscosity were calculated using kinetic theory. In addition, for this problem, the domain had to be divided into a quite large number of control volumes. The grid consisted of 307 955 tetrahedral cells, with the grid being finer in the fan area than in the fire area. To calculate the steady-state fire, the exit of the tunnel was first defined as a 'pressure inlet', and some faces of the volume where the fan stands at the entrance of the tunnel were defined as 'pressure outlets'. This, together with a small initial longitudinal velocity directed towards the entrance, facilitated convergence of the solution. FLUENT at first had some difficulties in obtaining a solution. However, once the calculation was on the right path - that is, after a few hundreds iterations - both ends of the tunnel were changed to outlet vent boundaries and calculation was carried out up to almost 1000 iterations.

The first calculation shows how long time it takes to establish the flow, taking into account the slope of the tunnel and the presence of the train, but without any fire. The fan was positioned 8 m from the entrance and tilted 7° down, but not angled to one side, as was done in Chapter 2. The primary flow of the fan was 37,5 m^3/s, as previously.

The longitudinal velocity (which corresponds to the X velocity) at measuring Point 1 (located in the middle of the tunnel, 150 m from the entrance and 2 m above the floor) is shown in Figure 17.

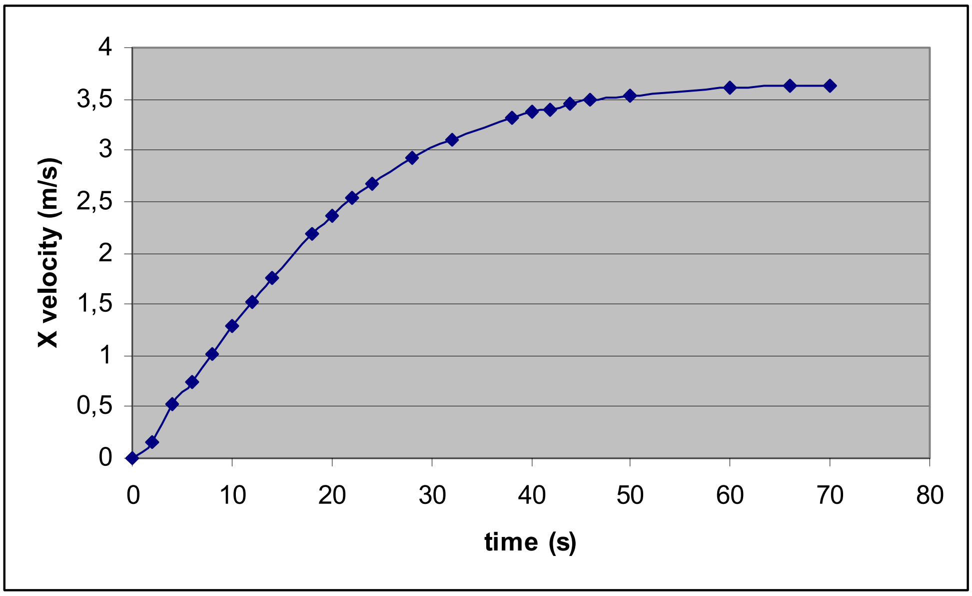

Figure 17 The air velocity in the tunnel, 150 m from the entrance. A 100 m long train is located in the middle of the tunnel.

It can be seen that, because of the presence of the train, the flow is established more quickly, but the blockage also leads to a slightly lower maximum velocity. It takes about one minute for the flow to be established, as against about 2,6 minutes without any blockage. After one minute, and without the train unit, the velocity at Point 1 reached 2,86 m/s and then continued to rise to 3,75 m/s, whereas with the train unit, it reached 3,63 m/s after one minute and then remained constant. Actually, Equation 7 can be modified by assuming the same mass flow rate in the tunnel and consequently an increase in the velocity of $u_t = u\ A/(A-A_t)$ where u_t is the wind velocity around the train unit and A_t is the cross-sectional area of the train wagon. This action would increase the flow resistance and consequently the parameter β. Thus, from Equation 8, we can see that u_{max} is decreased and η would be increased, reducing the time to reach steady-state, or u_{max}. This appears to be in accordance with the results obtained by FLUENT.

Actually, it has been shown experimentally that the flow establishes more quickly, but reaches lower velocities, when there is a blockage in the tunnel. Some experiments were performed in Zurich in the Zimmerberg tunnel [1] which is a single-track railway tunnel like the Manesse tunnel. The cross-section is nearly the same but the tunnel is much longer (1980 m). Without any blockage, it took about nine minutes for the flow to be established, and the air velocity 2 m above the ground was measured at about 2,5 m/s. But with some wagons placed inside the tunnel, it only took four minutes for the flow to be established, although the final velocity was lower, at about 1,75 m/s. This also indicates that the results obtained with FLUENT are reasonable.

In our simulations with the fire, we assumed that the fire source consisted of a kerosene pool, located in the middle of the tunnel and in the middle of the train unit. The pool was 5 m long and 3 m wide and 0,38 kg/s of kerosene was released. This corresponds to a 15 MW fire, assuming a chemical heat of combustion of 39,5 MJ/kg.

As expected, the calculation shows that the heat tends to go upwards in the tunnel. The mass flow rate at 15 m from the centre of the tunnel on the entrance side is 6,87 kg/s, while on the exit side it is 6,07 kg/s at the same distance from the centre of the tunnel. Figure 18 shows the gas temperature profile along a line situated 30 cm below the roof. The distance x increases from the entrance opening towards the exit opening. The peak gas temperature (1600 K = 1327 °C) is reached at the centre of the tunnel (x = 261,5 m) where the pool fire is located. Since the heat tends to move towards the entrance, the gas temperature does not fall as rapidly in that direction.

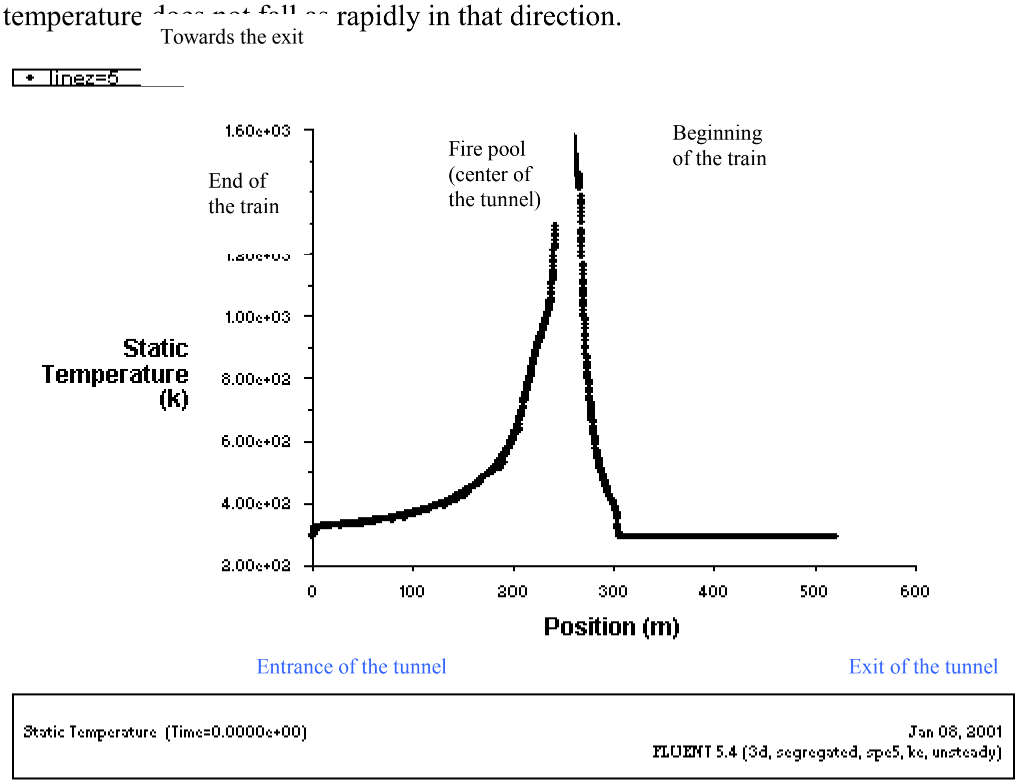

Figure 18 The temperature distribution along the tunnel 30 cm below the roof.

The air velocity created by the fire is shown in

Figure 19. It shows the longitudinal velocity profile along a line located 30 cm below the roof, i.e. in the hot smoke layer. The centre of the tunnel corresponds to the position x = 261,5 m. In accordance with the previous results, velocities are negative and higher on the entrance side, ranging from -13 to –1 m/s, than on the exit side where the velocities are positive, 4,7 m/s at maximum, close to the fire but slightly negative further away. This indicates that some air is drawn in from the exit.

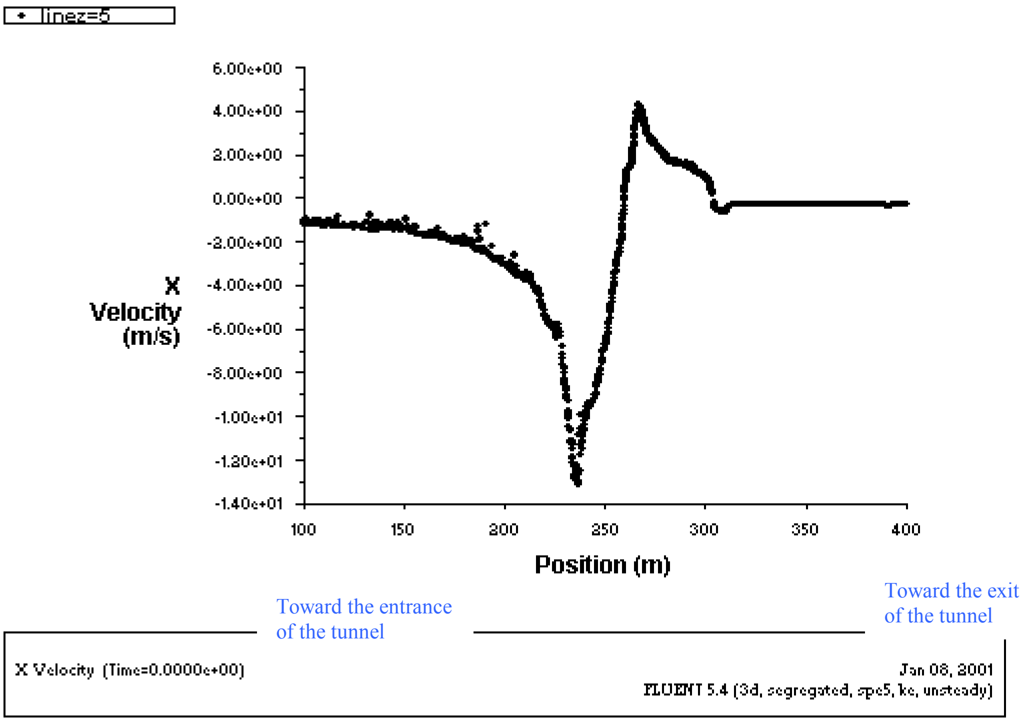

Figure 19 The air velocity along the tunnel 30 cm below the roof.

Once we obtained a steady-state solution inside the tunnel, the fan at the entrance of the tunnel was started and the calculation was carried out using the transient mode. The calculation shows that the fan easily reverses the flow. In fact, only six seconds after the fan is started, the air velocity at the top of the tunnel is no longer negative. At maximum, it is about twice what it was initially at the bottom. The longitudinal velocity then increases rapidly. During the time period from 10 to 36 seconds, the velocity distribution becomes more and more regular from the ground to the top of the tunnel. The velocity continues to rise until it reaches the maximum value of 4,5 m/s. After the velocity reaches a maximum value of 4,5 m/s, it starts to decay and from 36 s to 60 s, it decreases to less than 2,50 m/s at maximum.

The time it takes to reverse the flow is clearly linked to the fact that the X-velocity was already positive at the lower part of the tunnel, whereas it was negative in the hot layer at the top of the tunnel. Moreover, the velocity for a short time period reaches higher values than without any fire, or up to 4,5 m/s, while in the case of the fan blowing without a fire, it was not more than 3,6 m/s. One explanation may be that once the flow is reversed, all the heat from the fire is driven towards the end of the tunnel so that the air in the left part of the tunnel boosts the flow for a short time period. But then the velocity decreases, which is due to the resistance opposed by the fire since the hot air would naturally tend to go in the opposite direction. The final reversed velocity will be 2,5 m/s instead of 3,6 m/s.

Now we will explore the development of the temperature profile along a line 80 cm below the roof. As expected, the temperature in the left part of the tunnel – towards the entrance - decreases rapidly, since the fan is blowing fresh air into the tunnel. The hot air - and the smoke - is pushed from left to right. The hot air concentrated above the fire region is suddenly pushed toward the right side (flipped over). The initial temperature peak moves to the right and decreases gradually, until finally disappearing. At the same time, the temperature above the fire remains very high. To enable the fire brigade to have an easy access to the heart of the fire, it is important that the temperature close to the fire is reasonable. Figure 20 shows the development of the temperature from the entrance of the tunnel up to 255 m, that is to say only 4 m from the fire pool. After one minute, the temperature has already fallen to 52°C and the temperature on the fan side has become tolerable for fire fighters. This shows that, for the scenario presented here, the fan can easily expel the heat and toxic gases and produce a safe environment for fire fighters on the upstream side. All the heat and smoke is pushed away from the fire source.

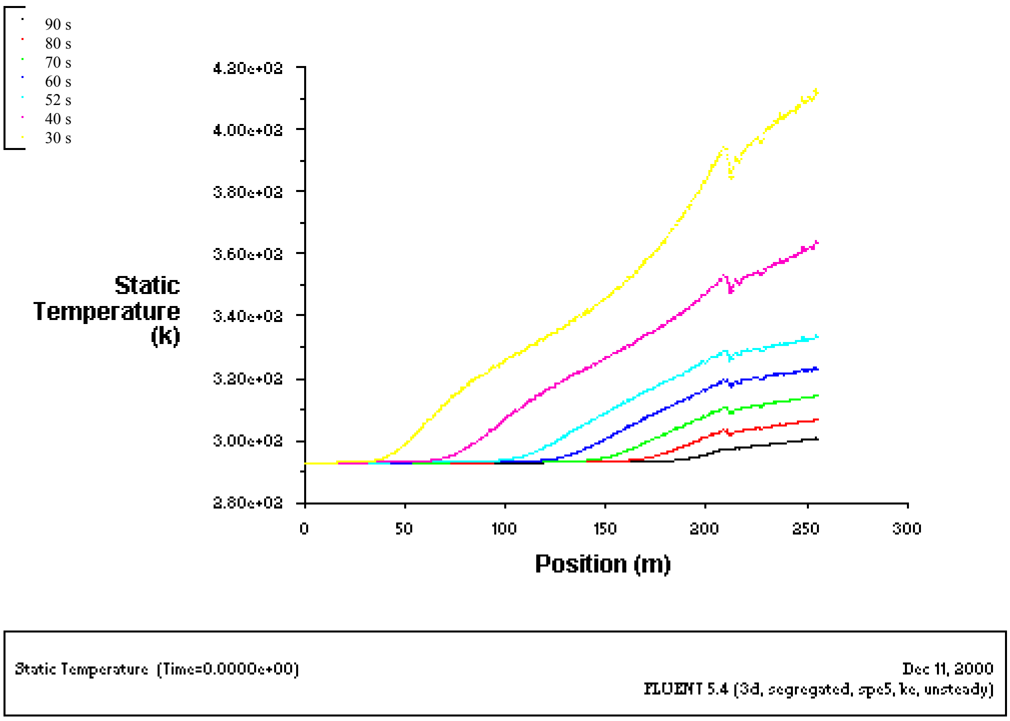

Figure 20 Temperature development 80 cm below the roof at different times and positions. The highest line to the left corresponds to 30 seconds and the lowest line to the right corresponds to 90 seconds from the start of the fan.

3.2 Simulation of the Käferberg tunnel with fire

The same type of simulation was performed for the Käferberg tunnel (A = 45,43 m^2, L = 2118 m). The previous 'cold' studies showed that it took much longer to establish a flow with a fan in the Käferberg tunnel than in the Manesse tunnel. We can therefore expect that it will also take longer to reverse the flow created by a fire. This time, the fire is located in the middle of the tunnel and in the middle of a 500 m long train with the same cross-sectional area as the Manesse train (12 m^2). The slope of the tunnel was also taken into account in the calculation. The fan was placed in an unfavourable position: that is, at the entrance (the altitude is 25 m higher than at the exit). The same type of fire source was used as in the Manesse tunnel, i.e. a 15 MW kerosene fire.

Since the Käferberg tunnel is much larger, the number of cells in the calculation domain increased, despite the fact that the mesh was made fine only where it was needed. The grid was divided into 677 802 cells. As a consequence, the calculations took much longer time, more than three days, for example, for the transient case of the fan reversing the flow induced by the fire (Pentium II - 750 Mhz, 500 Mb Ram). The inputs and the method used to make the calculations with FLUENT are the same as for Manesse tunnel.

The same type of fan was positioned outside the tunnel, 14 m from the entrance and tilted 7° down, but not aimed sideways. Some experimental tests were made in Käferberg tunnel with an obstacle above one track. Two meters above the floor, velocities between 0,3 and 0,6 m/s were initially measured along the tunnel and, once the fan was activated, the velocity increased almost immediately to values between 1,5 m/s to 2,3 m/s along the tunnel. There is almost no reduction of the velocity in this wide tunnel due to the presence of the train. This should give us an idea of the final velocities that will be predicted by FLUENT. The simulation was carried out with zero initial longitudinal velocities, as was done previously. Figure 21 shows the development of the air velocity in the centre of the tunnel, 2 m above the floor and at a distance of 150 m from the entrance.

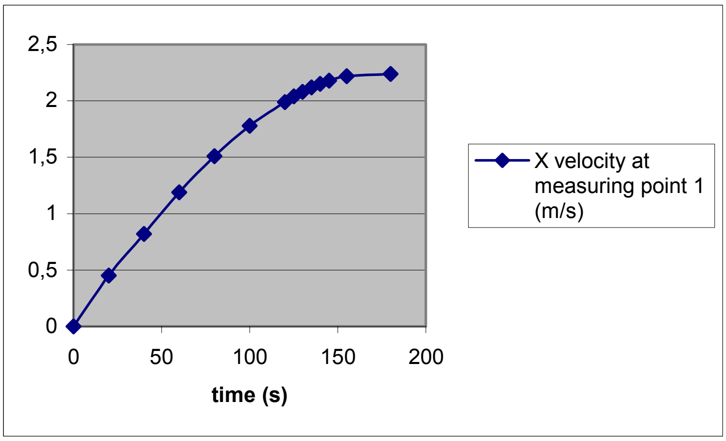

Figure 21 The air velocity 150 m from the entrance of the Käferberg tunnel and with a 500 m long train unit.

The results are in accordance with the experimental data obtained without any blockage. Only one track was obstructed, and the final air velocity appears to be nearly the same as without any obstacle. However, the air flow was fully developed within three minutes, and not ten minutes as in the case without any blockage. Thus, in the same way as for the Manesse tunnel, the flow was established much more quickly when there was a blockage within the tunnel.

Figure 22 shows the temperature profile along two longitudinal lines, one situated 2 meters above the ground, and the other 1 m below the roof (or 5 m above the floor). The temperature peak occurs above the fire at the centre of the tunnel (which corresponds to the position x = 1059 m in the plot). The temperature then decreases rapidly with distance from the fire due to the effects of convection and radiation losses to the walls. At a distance just less than 50 m from the fire source, the temperature has already decreased to half the maximum value, and continues to decrease towards the exit. The temperature decreases even more quickly towards the entrance as a result of the incoming flow of fresh air from the exit.

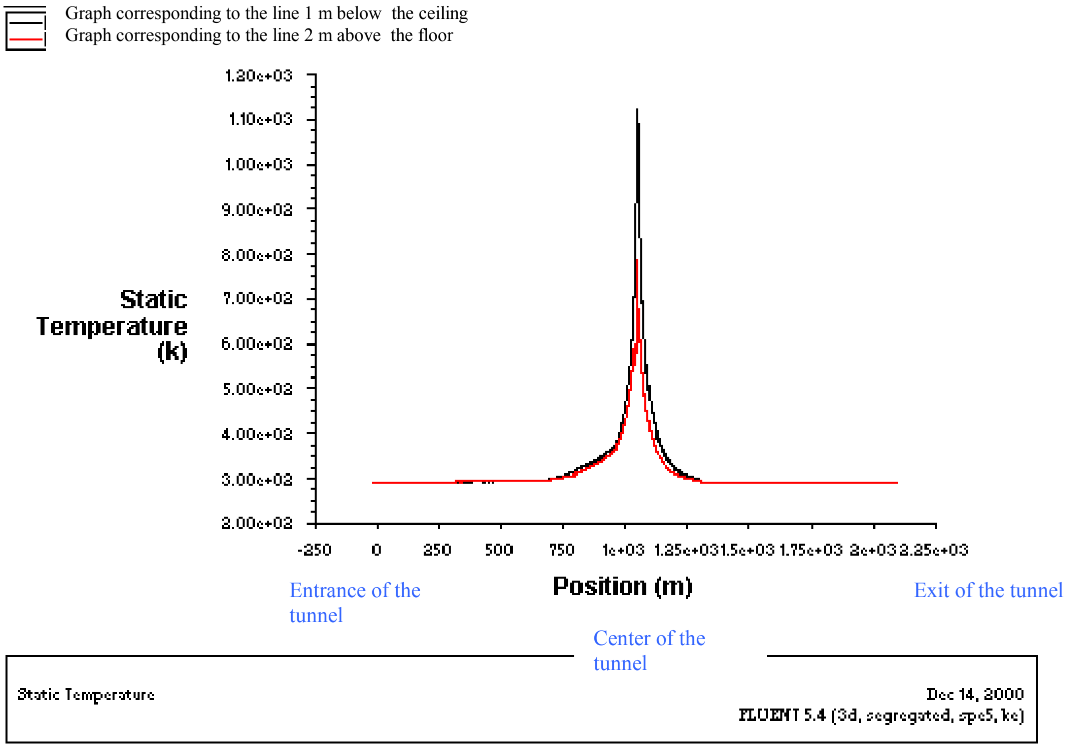

Figure 22 **The temperature in Kelvin 1 m under the roof and 2 m above ground at different locations inside the tunnel (steady-state conditions).**

In the same way as we observed for the Manesse tunnel, above the fire, the majority of the flow moves towards the entrance (negative x velocities) and the rest move towards the exit (positive x velocities). However, closer to the floor, the velocities are all negative even on the entrance side. Thus, the fire creates velocities which are positive in the upper part of the tunnel and negative at the bottom, varying from –2,75 m/s to 3.75 m/s. At the entrance side, the velocity is always negative so all the air is moving towards the entrance. This means that when the fan will be started it will probably take longer time to obtain a steady flow in the opposite direction. In the Manesse tunnel we had some help form inflowing air in the lower parts of the tunnel cross-section.

Once a steady-state solution of the flow created by the fire was obtained, the calculation was carried out using the transient option with the fan turned on. The maximum x velocity was 2,1 m/s and it was reached after about three minutes from the time the fan started. Then the velocity decreased slightly, to 2,05 m/s. We can note that the time needed for the flow to be established, is slightly longer than was calculated without the fire (the flow become almost steady after 2 min 35 s). The maximum velocity is also slightly lower than was calculated without any fire (2,24 m/s), due to buoyancy effects. The buoyancy forces try to maintain a flow in the opposite direction. The time required to reverse the flow completely is also three times as long as that for Manesse tunnel.

4 Discussion of results

The calculations presented here have shown that mobile fans can be very useful in cases of fire in a tunnel. The flow that a fan provides appears to be strong enough to reverse the flow created by a fire that is up to 15 MW. Although more time is required for a larger tunnel than for a small one, the heat and the smoke are efficiently pushed on one side of the tunnel. It is found that the time to reverse the airflow created by a fire is not much longer than the time needed to reverse the airflow with no fire. In some cases, where we have an initial bi-directional flow created by the fire, it may actually take less time. In the simulation with the Manesse tunnel, the airflow along the floor of the tunnel was directed towards the fire, which apparently extended the time required to reverse the airflow. The velocity of the airflow moving towards the fire rose to 4,5 m/s within only 36 s, and then fell to 2,5 m/s after one minute, i.e. exactly the same time as without any fire. If we consider the case of quite a long and wide tunnel, such as the Käferberg tunnel, where the bi-directional flow appears only at a considerable distance from the entrance, we can note that the establishment of the flow is more progressive. The velocity finally reaches a value of 2,05 m/s in not much longer time than it takes without a fire, i.e. about three minutes. In both cases, the final velocities are lower than without a fire because of the buoyancy forces of the fire.

An interesting observation from these calculations is that the time required to establish a fully developed airflow inside the tunnel is much shorter when there is a blockage inside the tunnel. This can be seen in Table 1, which recapitulates the key figures of this study. All of these figures are computational results.

Table 1 Summary of the results obtained by FLUENT simulation for the Manesse and the Käferberg tunnels.

		Manesse tunnel	Käferberg tunnel
Establishment of a flow with a mobile fan without an obstacle or a fire in the tunnels	Final longitudinal velocity reached inside the tunnel (m/s)	3,75	2,24
	Time needed to reverse the airflow (minutes)	4	10
Establishment of a flow with a mobile fan with an obstacle (train unit), but no fire in the tunnels	Final longitudinal velocity reached inside the tunnel (m/s)	3,63	2,24
	Time needed to reverse the airflow (minutes)	1	3
Establishment of a flow with a mobile fan with an obstacle (train unit) and a fire in the tunnels	Final longitudinal velocity reached inside the tunnel (m/s)	2,5	2,05
	Time needed to reverse the airflow (minutes)	1	3

5 Conclusions

In this report we have considered the use of a mobile fan for railway and road tunnels. We studied the case of a fairly small tunnel, the Manesse tunnel (523 m), and of a larger tunnel, the Käferberg tunnel (2118 m). We carried out comparison between experimental and computational results, which showed that FLUENT yields reasonable results when we have no blockage and no fire. In addition, the calculations showed that the time required to establish a steady flow inside a tunnel is closely linked to the size of the tunnel. The FLUENT calculations showed that the flow supplied by the fan was established in the Manesse tunnel and the Käferberg tunnel within 4 and 10 minutes respectively, and the final velocities were 3,75 and 2,24 m/s. These values are reasonably close to experimental value obtained. A simple theoretical model was also developed, showing similar results. We can therefore conclude that the simulations are acceptable for this type of work. The next step was to include a fire and blockage (train unit) within the tunnel.

We performed two simulations assuming a 15 MW fire in the middle of a train unit in the middle of the Manesse tunnel and of the Käferberg tunnel. We took into account the differences of altitude between the ends of both tunnels because of the buoyancy forces of the fire. After that we activated the fans and continued the calculations. It appeared that the flows supplied by the fan are established within almost the same times, one minute and three minutes respectively, as when there is no fire, but with the trains in the tunnels. We also saw that obstacles inside tunnels have a strong influence and considerably reduce the time required for the flow to become established. However, the final velocities are slightly lower. They are also lower in the case of fire, as there is a resistance to the flow caused by the convection effect of the heat from the fire. Thus, the use of mobile fans positioned at the entrance of a tunnel similar in size as the one investigated here should allow rapid intervention of the fire brigade directly at the heart of the fire and also in a safe atmosphere.

However, one must keep in mind that these calculations are only predictions. We have no experimental data available to check the results of the with-fire calculations, but considering the accuracy of the previous calculations where we were able to compare to experimental data (no fire), there is no doubt that the order of magnitude, a few minutes, is reasonable. So although mobile fans alone may not be the panacea that can save everyone inside the tunnel, it will undoubtedly create a much safer environment for the fire fighters and it will considerably shorten the time for access to the tunnel.

6 References

1 Bader, J., Mobile ventilation unit for positive pressure ventilation in large spaces, Report from Brandschultz-Innovationen in Germany.

2 Fluent 5 User's Guide Volume 1 and 2, see also; http://calmip.cict.fr:8010/fluent/fluent5/index.htm

3 Fires in Transport Tunnels: Report on full-scale tests, EUREKA-Project EU499; Firetun, Studiengesellschaft Stahlanwendung elV. D-40213 Dusseldorf, 1995.

4 GAMBIT, Users Guide http://zsc.zcu.cz/sw/manuals/gambit/users_guide

Appendix A – Test of fan model in FLUENT

If used correctly, positive pressure ventilation is a very effective means of ventilating smoke and heat from a fire compartment. The method involves placing mobile fans in front of openings in the building to blow toxic fire gases and heat out of the building. This lowers the temperature and improves visibility, resulting in improved working conditions for the fire fighters.

Some experiments to test the fan model in FLUENT were carried out in a test chamber, of which the dimensions were 10 m*16 m*5 m. The area of the door was 3,6 m^2 (2,52*1,43) and the area of the exhaust opening was A = 3,39 m^2 (3 m*1,13 m). The fan was placed 3 m far from the door and angled upwards at 20°. The diameter of the fan was 0,48 m, its hub radius was 0,07 m and its speed 3600 r/min. It provided a primary flow of 3,33 m^3/s.

The test showed that, once a steady flow was established, the velocity in the middle of the exhaust opening was 2,42 m/s. Based on this value, we were able to determine the airflow from the exhaust opening; $Q = 0,61 * A * U = 5$ m^3/s. The pressure difference inside the room at a distance of 4 m from the exhaust opening was measured to be 4.6 Pa.

The simulation was performed with the standard fan model in FLUENT assuming a step pressure rise. The temperature was set to 290 K and the pressure to 101325 Pa. To reach the correct primary airflow for the fan, the step pressure rise was set to 1404 Pa. The tangential velocities and radial velocities were set to 3 m/s and 6 m/s respectively.

The results obtained with FLUENT are very close to the experimental data. The velocity in the centre of the exhaust opening was 2,72 m/s., which can be compared with the experimental value of 2.42 m/s. The flow as calculated previously is 5,62 m^3/s and the pressure inside the room is 5,43 Pa. The experimental value was 4.6 Pa.

The results provided by FLUENT can be used to show the distribution of velocity across the exhaust opening and inside the room. For example, the average velocity at the exhaust opening is 2,06 m/s and the resulting flow through it is 7 m^3/s, which also tends to show that the approximations made to calculate the flow from the experimental value of U are not appropriate. In general we can say that the fan model used in FLUENT appears to give reasonable results for a mobile fan.

SP Swedish National Testing and Research Institute
Box 857, SE-501 15 BORÅS, Sweden
Telephone: +46 33 16 50 00, Telefax: +46 33 13 55 02
E-mail: info@sp.se, Internet: www.sp.se

SP REPORT 2002:06
ISBN 91-7848-895-8
ISSN 0284-5172