AIRFLOW GENERATION IN A TUNNEL USING A SACCARDO VENTILATION SYSTEM AGAINST THE BUOYANCY EFFECT PRODUCED BY A FIRE

J D Castro\textsuperscript{a}, C W Pope\textsuperscript{a} and R D Matthews\textsuperscript{b}

\textsuperscript{a}Mott MacDonald Ltd, St Anne House, 20-26 Wellesley Road, Croydon, CR9 2UL, UK
\textsuperscript{b}Centre for Tunnel Aerodynamic Research, South Bank University and Impulse Ventilation Systems Ltd UK

ABSTRACT

It is generally difficult to provide ventilation shafts for tunnels which pass under water or high mountains. An alternative is to ventilate the tunnel using the Saccardo system with impulse fans located at the portals. Tunnels that pass under water descend to a low point and there is concern over the capability of a Saccardo system to provide adequate levels of ventilation control in the case of a fire occurring in an inclined section of tunnel. This paper considers the problem in relation to a train fire occurring in a single track rail tunnel 3 km long. In order to investigate the effect of Saccardo ventilation nozzles on the flow generated by buoyancy from the fire, a computational fluid dynamics (CFD) model was used. The results indicate that a practical Saccardo system can be developed to provide effective smoke control against the effect of buoyancy in a fire situation. Moreover, the predictions indicate that fan “start up” may be achieved against an established buoyancy driven flow by the fire. The time taken to clear the air behind the fire is dependent on the response time of the fans.

Key words: Saccardo nozzles, fire, smoke control, CFD

1. INTRODUCTION

The design of rail tunnels demands that adequate ventilation systems are installed in order to ensure effective control of smoke during a fire incident. The Saccardo system (Ref 1) is a very convenient means of ventilating a tunnel under emergency conditions where the provision of a conventional shafts based ventilation system is difficult. In this paper, consideration is given to the question of whether a Saccardo impulse fan ventilation system has the ability to control the heat and smoke from a train fire in an inclined tunnel, where buoyancy forces are an important factor.

The rail tunnel under consideration is 3 km long with a Saccardo ventilation installation located at the entry portal. The tunnel falls on a gradient of 2.5% to the mid-point and then ascends at 2.5% to the exit. The injection nozzle is positioned in a cut and cover section 44 m from the bored section of the tunnel and directed towards the exit portal. The entrance at the tunnel has been designed as a cut and cover section with the intent of encouraging the development of a forced flow through the tunnel by the diffusion pressure of the Saccardo nozzle.

Three different scenarios were considered in an ascending order of operating difficulty for the ventilation system to establish a robust flow over the train. The first simulation was for cold flow conditions (no train fire) to predict the proportion of ventilation air entering the tunnel. The second simulation was undertaken for the Saccardo fan switched on at the same time as a fire breaks out on the train. Finally, a simulation was undertaken for the ventilation system initiated after a buoyancy driven flow had been established in the tunnel by the fire.
A train 400 m long with a 7 MW fire at the leading end is positioned 600 m from the entry portal. A diagram of the train and tunnel configuration is given in Figure 1. The critical flow velocity for the case under consideration was determined from a one-dimensional flow analysis to be $1.5 \text{ m s}^{-1}$.

In order to investigate the effect of Saccardo type ventilation nozzles on the propagation of smoke from a train fire within the tunnel, the problem was analyzed using a simplified CFD model. A description of the model and the predicted results are given in the sections which follow.

![Figure 1: Diagram of Rail Tunnel Configuration (Scaled Vertically to Aid Viewing)](image)

### 2. COMPUTATIONAL FLUID DYNAMICS MODEL OF TUNNEL

A CFD model was developed in order to perform three-dimensional simulations of the movement of heat and smoke from the fire.

In order to investigate the basic characteristics of the problem, a simple rectangular blockage was used to represent the train, the front of which was located 1000 m from the mouth of the bored tunnel. A diagram showing the geometrical arrangement is given in Figure 2. To enable the CFD calculations to be performed within a short period of time, a half-model of the tunnel was constructed, a plane of symmetry being imposed along the centre-line of the tunnel.

In order to perform the simulations, the CFX code from AEA Technology was used. The code solves the fluid mechanics equations using the method of finite volumes. To represent the effects of turbulence, the standard $k-\varepsilon$ model was employed. The flow was treated as weakly compressible, the density given by the equation of state. Time steps of 5 s were chosen, with a suitable number of iterations for each time step in order to ensure adequate convergence. A close-up view of the computational mesh is shown in the lower half of Figure 2. Time constraints prevented any assessment of the effect of grid resolution on the results. This should, however, form a key element of any future in-depth study.
The 7 MW fire was represented as a volumetric source of heat, located at the end of the train farthest from the tunnel portal. The heat was specified as a constant source in the enthalpy equation and the volume of the source was updated at each time step to ensure reasonable temperatures at the source. No account was taken of radiation, hence the results will err on the conservative side with respect to the propagation of heat and smoke towards the tunnel exit. Smoke from the fire was represented by adding a source of passive scalar. A scalar concentration of 1% has been adopted as an acceptable limit with regards to obscuration and visibility but the actual mass concentration of soot will depend on the properties of the fuel being burnt. The 1% value is considered a suitable indicator here, since it is only required in this study to evaluate the effectiveness of the Saccardo nozzle in establishing a flow of fresh, smoke-free air over the train.

The Saccardo nozzle was represented by adding an additional block to the roof of the tunnel. This is required to properly capture the flow from the fan down into the cut-and-cover section of the tunnel. The area of the nozzle inlet was 4 m², with a volumetric flow from the fan of 120 m³/s. A turbulence intensity of 5% was specified at the fan inlet, with the hydraulic diameter of the inlet used as the dissipation length scale.
3. SIMULATIONS PERFORMED

Cold Flow Case

The first simulation undertaken was a cold flow simulation (no fire source) with the Saccardo fan in operation. This was undertaken to assess the capability of the Saccardo system in establishing a strong enough tunnel flow in the absence of a fire. The average velocity predicted in the bored tunnel was 2.7 m s$^{-1}$, indicating a flow of 96 m$^3$s$^{-1}$ from the Saccardo nozzle through the tunnel and 24 m$^3$s$^{-1}$ out of the south portal. The friction factor, $f$, applied on the walls of the tunnel was taken to be 0.007.

A one-dimensional unsteady tunnel flow program (Ref 2, based on the method of characteristics) was also used to undertake simulations of the ventilation system. The results from this analysis indicate a steady cold flow air velocity of 2.3 m s$^{-1}$, rather than 2.7 m s$^{-1}$ as predicted in the CFD analysis. The difference can be related to the use of a momentum transfer coefficient in the 1-D network studies, the CFD analysis giving higher diffusion pressures in the region of the Saccardo nozzle. The 1-D simulations are naturally unable to take into account any three-dimensional variation in the flow field.

Plots showing the predicted velocity field in the vicinity of the Saccardo nozzle and over the train are given in Figure 3.

![Figure 3: Predicted Flow in Region of Saccardo Nozzle and over Train](image)

The circulation of air immediately behind the Saccardo inlet stream is clearly apparent. The CFD simulation predicts a more or less uniform flow velocity at the boundary between the Cut and Cover and Bored sections of tunnel. The profile becomes fully developed by the time it reaches the front of the train.

7 MW Fire, Saccardo Fan activated at time zero

The first fire simulation undertaken was for the case of the Saccardo fan activated at the same time as when the fire breaks out, i.e. at time zero. No back-layering from the fire is predicted. The flow velocity down the bored tunnel towards the train reaches its steady-state value by 7.5 minutes. Figure 4 illustrates the results of the simulation:
Figure 4: Predicted Smoke Layer at 1, 3, 5 and 7.5 minutes

7 MW Fire, Saccardo Fan activated at 10 minutes

In this simulation, no ventilation is applied for 10 minutes. During this time, a buoyancy driven flow is established in which smoke and hot gas from the fire reaches the exit. By 10 minutes, virtually steady state conditions are attained, the smoke layer having reached the exit and subsequently venting out to the atmosphere.

At 10 minutes, the Saccardo fan is switched on to work against the effect of buoyancy and reverse the flow. The CFD simulation predicts that it takes a further 10 minutes to establish a steady flow in the opposite direction, with a column of clean air between the fan and location of the fire. Thus fan “start up” may be achieved against an established buoyancy driven flow, although it takes a while to achieve the desired result. It is expected that an earlier start up time would result in a more rapid clearing of the air behind the fire. However, time constraints prevented a more thorough examination at this stage.

Plots of smoke concentration during the scenario are illustrated in Figure 5. The CFD simulation predicts the movement of the smoke layer past the location of the Saccardo nozzle by 7.5 minutes. By 10 minutes, virtually the entire cross-section of the tunnel upstream of the fire is affected by smoke. The Saccardo fan flow takes a couple of minutes to begin fully clearing the smoke layer in the bored tunnel. Five minutes after the fan is switched on, nearly the entire length of bored tunnel upstream of the train is clear of smoke. A steady flow of smoke out of the north end of the tunnel is established by 20 minutes.
On examination of the results of the standard k-ε model, it is worthwhile noting the thickness of the smoke layer which is predicted by the model. Modifications to the flow equations are possible in order to account for the suppression of turbulence in the smoke layer, which would be expected to lead to a thinner and more well stratified layer.

It would be worth re-running the model at a later stage to examine the effect of modifying the flow equations on the predicted results. Since a better stratified layer may be expected in practice, the Saccardo fan could take longer to drive the smoke back down the tunnel.

Figure 5: Predicted Smoke Layer at 5, 7.5, 12.5, 15 and 20 minutes
(Contours of 1% - 5%+ concentration)
4. CONCLUSIONS

Consideration has been given to the problem of generating a forced ventilation flow in a tunnel using a Saccardo system against an established buoyancy driven flow produced by a train fire. The predictions have shown that the system is capable of overcoming the buoyancy forces to produce a robust unidirectional flow directing heat and smoke away from the train. The study also demonstrates the potential usefulness in applying CFD techniques to the prediction of smoke movement in underground rail tunnels using impulse ventilation systems.

For future studies, it would be worth performing a grid-dependency analysis, as well as examining the effect of choice of turbulence model and differencing scheme on the results. Ideally full-scale experimental tests should be carried out to confirm or otherwise the general smoke behaviour predicted using CFD.

ACKNOWLEDGMENTS

The authors wish to thank the directors of Mott MacDonald for permission to publish this paper.

The work was carried out in connection with the Channel Tunnel Rail Link (CTRL) project. South Bank University is working solely in an advisory capacity to Rail Link Engineering (RLE) on the CTRL project and the design is that of RLE.

REFERENCES

