Colella F., Rein G., Carvel R., Torero J.L., (2010) Tunnel ventilation effectiveness in fire scenarios, FS-World Magazine, Spring 2010, Special Edition on Tunnel Safety, pp. 36-40

Tunnel ventilation effectiveness in fire scenarios

F. Colella[a], G. Rein[a], R. Carvel[a], J.L. Torero[a]
[a] University of Edinburgh, BRE Centre for Fire Safety Engineering, UK

Introduction

Over the past two decades, computer models of tunnel ventilation and fire behaviour have changed from being rough mathematical approximations of real world phenomena to being highly detailed and complex simulations able to reproduce turbulent flow patterns, fire dynamics and smoke movements. Computational Fluid Dynamics (CFD) analysis of tunnel ventilation systems has become an integral part of almost every tunnel project, whether it is for a new-build tunnel or refurbishing existing systems. However, the computational resources required for state-of-the art modelling remain very high, both in terms of computer equipment and in terms of simulation run-time. The very large aspect ratio between longitudinal and transversal length scales in a tunnel leads to very large meshes and the number of grid points becomes impractical for engineering purposes, even for short tunnels (less than 500 m long). There always has to be a trade off between the level of detail of the simulations, the number of simulations able to be carried out and the cost of the project.

The computational cost leads to a practical problem arising when conditions or flow characteristics in remote locations far way from the region of interests are to be considered in the CFD model. This is the case of tunnel portals, ventilation stations or jet fan series located long distances away from the fire. In these cases, even if only a limited region of the tunnel has to be investigated for the fire, an accurate prediction of the flow behaviour requires that the numerical model includes the ventilation devices and therefore the whole tunnel. For some tunnels, this could mean that the computational domain is several kilometres long.

This article describes a novel approach to model such systems using CFD simulations to describe those parts of the tunnel network where capturing the full 3D flow pattern is required, while simulating other parts of the tunnel network using a simpler 1D model. This new approach enables the model to run up to 100 times faster than using the conventional full-CFD approach, without losing accuracy in the predictions. Projects using this methodology can investigate a much wider range of ventilation conditions or fire scenarios in the same time and budget

The multiscale concept

The study of ventilation and fire induced flows in tunnels provides the evidence that for most the tunnel domain the flow is approximately 1D and it can be accurately represented by a conventional network model. These regions of domain are usually named far field. However, in the vicinity of operating jet fans or close to the fire source the flow field has a complex 3D behaviour with large transversal and longitudinal gradients in the flow and temperature fields. These regions are usually named near field and their behaviour can be predicted only by means of complex CFD tools.

The multi-scale concept is an extension of the conventional 1D and CFD modelling techniques where the two models are coupled together with the latter providing the boundary condition to the former and vice-versa. The multiscale model is solved on a hybrid computational grid, where 1D elements are linked to 3D ones generating a continuous domain in the streamwise direction (see Figure 1). The behaviour of far field regions is modelled using a conventional 1D model. The near field regions are treated by introducing a 3D module in the network thus solved by a CFD tool. During the solution procedure 1D and CFD models constantly exchange information at the 1D-3D interfaces and thus run in parallel.

The 1D solver of the multi-scale model is based on the conventional techniques for fluid network systems. It adopts a generalized unsteady Bernoulli equation solved on a networked system representing the tunnel layout [1]. The tunnel domain is discretized in oriented elements called branches, interconnected by nodes. The 1-dimensional model accounts for pressure losses due to blockages (minor losses) and wall friction (major losses), global chimney effect and it has been previously validated against a wide range of experimental data. An example of a 1D network grid is presented in Figure 2.

For instance, once the CFD modules are introduced in the network, they could be used to analyze the

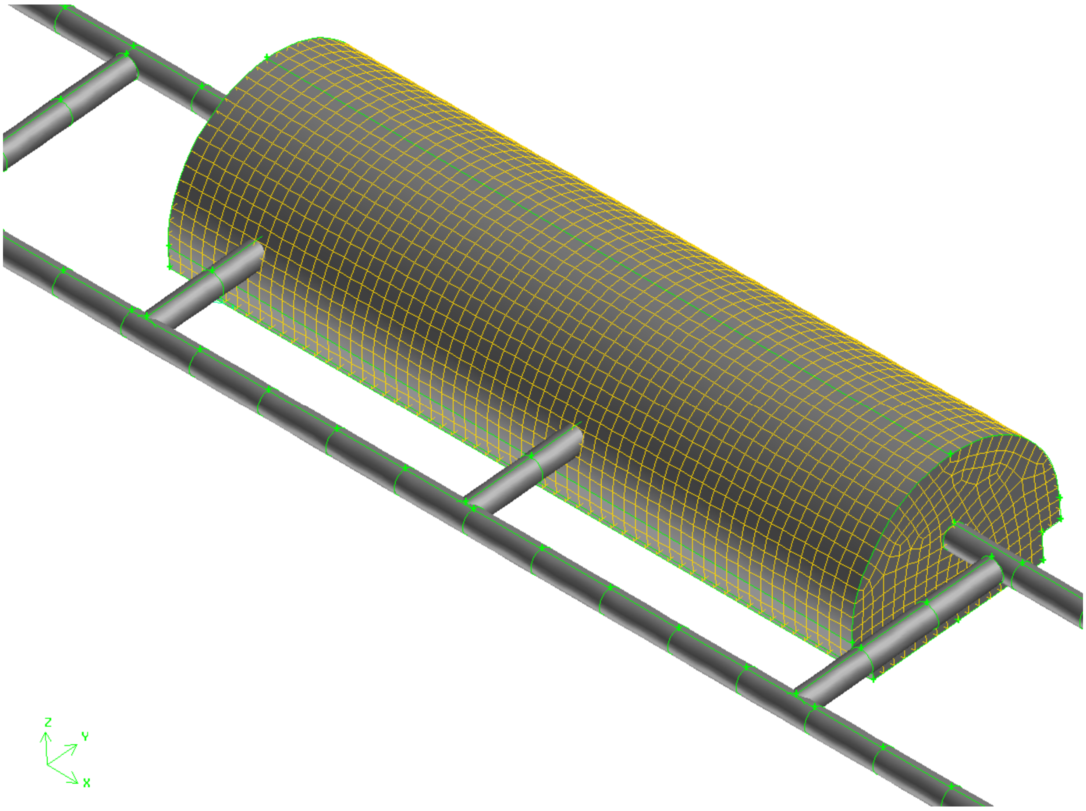

Figure 1. *Schematic of a hybrid computational grid: a 3D CFD module is inserted in a 1D network*

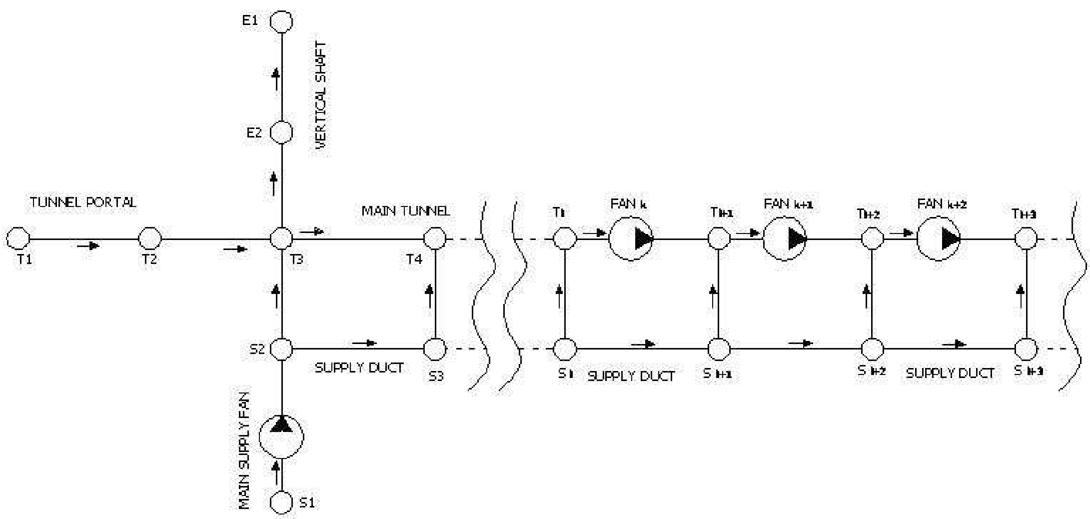

Figure 1. *Example of a 1D network representation of a complex tunnel ventilation system*

flow around large vehicles blocking the tunnel, junctions with vertical shafts, fire plume regions or flow patterns in the vicinity of ventilation devices. This article will only discuss two types of module, that of a module containing a pair of jet fans and a module containing an object on fire.

The jet-fan module

Assessments of the ventilation system performance often require a detailed study of the flow field around operating jet fans. These details are fundamental to understand how the jet fans interact with their surroundings (walls, niches, nearby fans) and how their installation affects the overall performance of

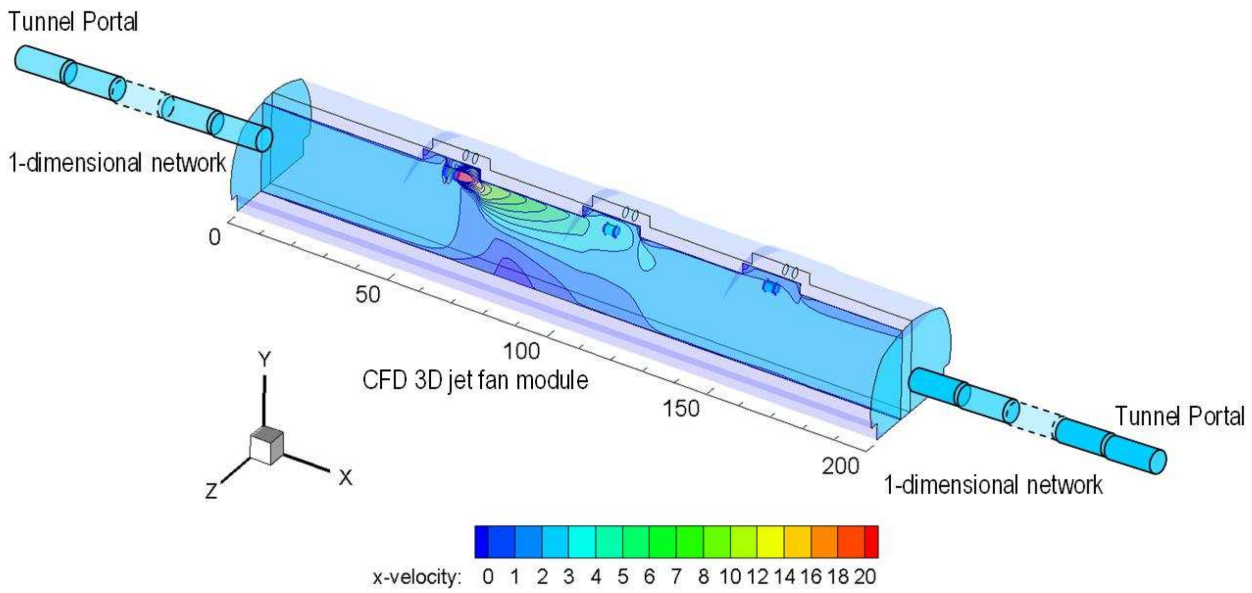

Figure 3. *A jet-fan module and the computed velocity field in the 3D CFD and in the 1D network domain; velocity values are expressed in m/s (not to scale)*

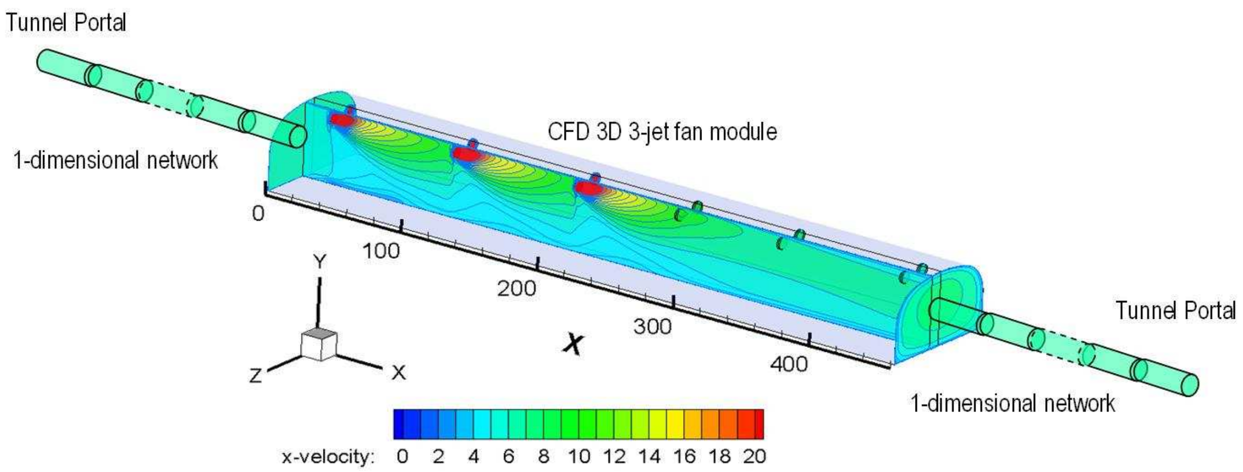

Figure 4*. A three-jet-fan module and computed velocity field in the 3D CFD and in the 1D network domain; velocity values are expressed in m/s (not to scale)*

the system and their capabilities of producing thrust. This task can be easily accomplished by adopting a multiscale modelling approach. The introduction of a single CFD jet fan module in the 1D tunnel network allows an accurate representation of the flow field surrounding the jet fans. Furthermore, the 1D model simulates the hydraulic behaviour of the rest of the tunnel domain providing accurate boundary conditions to the tunnel CFD module and enhancing the global accuracy of the model.

Our work shows that a CFD module length around 17 times the tunnel hydraulic diameter guarantees highly accurate results (less than 1% difference when compared to a CFD model of the entire system). However, the resulting computational time is reduced by 100 times [2].

Figure 3 shows an example of jet fan discharge cone calculation for a standard horseshoe cross section tunnel. As it can be seen the flow conditions at the 1D-3D interfaces are largely 1D with mild velocity gradients and therefore can be accurately represented by the 1D network model.

The interaction among a series of operating jet fans and the fluid flow pattern established in the tunnel domain can be also represented by the multiscale model. Figure 4 shows an example of the fluid flow field generated by three operating jet fan pairs in a tunnel. Also, in this case, the fluid flow at the boundaries is mainly 1D and it can be successfully

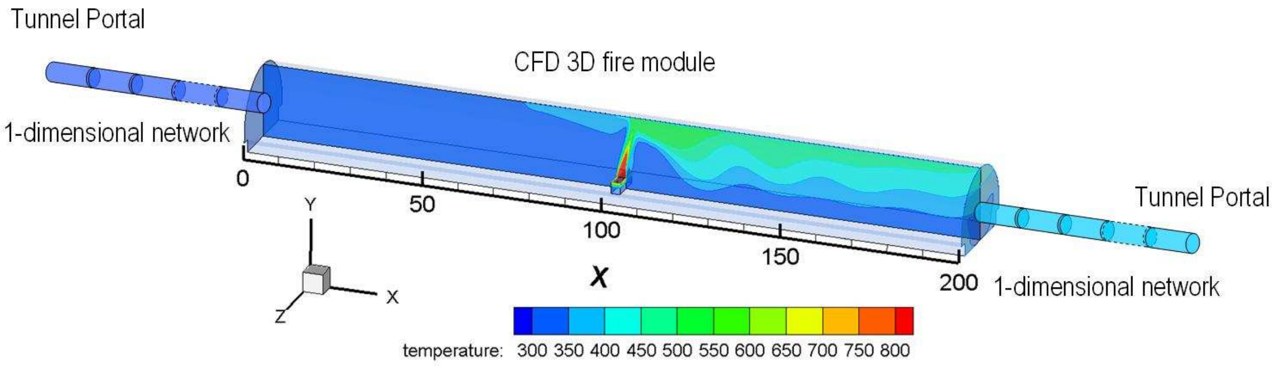

***Figure 5**. A 20MW fire module and computed temperature field in the 3D and in the 1D network domain; temperature values are expressed in K (not to scale)*

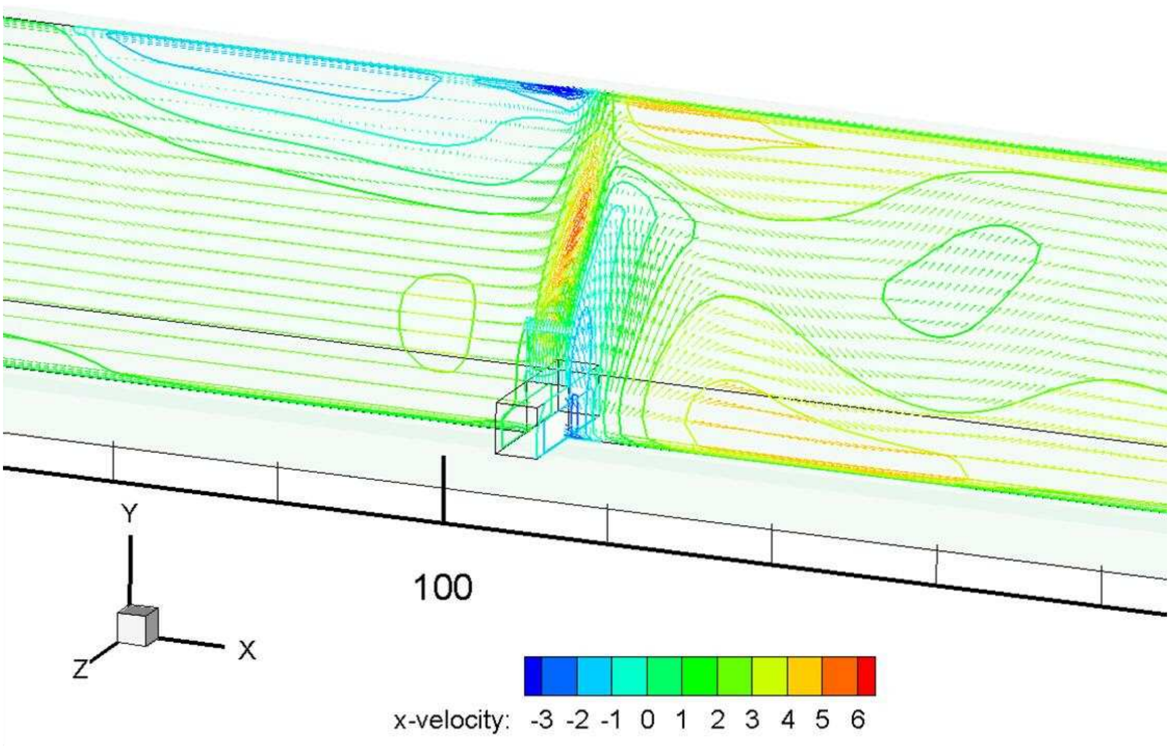

***Figure 6**. A 20MW fire module and computed horizontal velocity field in the 3D (not to scale)*

represented by using a conventional network model.

The fire module

An accurate risk assessment of tunnel fires often requires the calculation of temperature and velocity fields in the tunnel. In this case a fire module is introduced in the tunnel 1D network providing a detailed prediction of the temperature and velocity field in the near field. The resulting model is more versatile and flexible and can be adopted to conduct parametric studies as well as comprehensive analysis of the ventilation system under several fire hazards [3].

Figure 5 shows the temperature field established in the vicinity of a 20 MW fire in a tunnel with standard horseshoe cross section. The simulation has been conducted at a ventilation velocity as low as 2 m/s. A small degree of back-layering can be seen but it is largely within the 3D domain. A typical horizontal velocity pattern in proximity of the plume can be observed in Figure 6. A large region upstream the fire plume characterized by sustained backward motion confirms the presence of back-layering. The fire has been modelled as a volumetric source of energy at a constant rate without using a dedicated combustion model. This simplification is more than acceptable when compared to the large uncertainty of the real meteorological conditions at the portals, actual fire load, effective lining roughness, presence of vehicles and obstructions, etc.

Our work shows that a fire module length around 13 times the tunnel hydraulic diameter guarantees highly accurate results (less than 1% difference when to a CFD model of the entire tunnel). Also in this case the resulting computational time is reduced by about 100 times.

Conclusions

Throughout most of a tunnel network the ventilation behaviour may be approximated with a simple 1D flow model. However, there are some important - but relatively small - regions of the tunnel that require CFD analysis. The multi-scale model is the ideal tool for such tunnel studies as it allows accurate flow field predictions in some locations, yet allows simplifications where highly detailed data are not required.

It is shown that the accuracy of the multi-scale model is as high as the full CFD approach. The 100 times lower computational time is of great advantage because many ventilation scenarios can be explored and extensive sensitive parametric studies can be conducted.

The developed model has been adapted to conduct a comprehensive study of the ventilation system in several tunnels including the 1.5 km long Dartford tunnels (near London, UK). In this instance, the high accuracy of the model has been confirmed by comparison with experimental measurements. The model has been used to study a wide range of ventilation strategies [4].

References

[1] Ferro V, Borchiellini R, Giaretto V (1991) Description and application of tunnel simulation model. Proceedings of Aerodynamics and Ventilation of vehicle tunnels conference: pp 487-512

[2] Colella F, Rein G, Borchiellini R, Carvel R, Torero JL, Verda V.,(2009) Calculation and Design of Tunnel Ventilation Systems using a Two-scale Modelling Approach, Building and Environment 44, pp. 2357-2367

[3] Colella F., Rein G., Borchiellini R., Torero J.L. (2010), A Novel Multiscale Methodology for Simulating Tunnel Ventilation Flows during Fires, Fire Technology, DOI: 10.1007/s10694-010-0144-2.

[4] Colella F., Rein G., Carvel R., Reszka P., Torero J.L. (2010), Analysis of the ventilation systems in the Dartford Tunnels using a multiscale modelling Approach, Tunnelling and Underground Space Technology, doi:10.1016/j.tust.2010.02.007