# CFD MODELING OF ROAD TUNNEL FIRES

**M. Derudi, S. Tavelli, A. Frassoldati, A. Cuoci**
marco.derudi@polimi.it
Politecnico di Milano, Dip. di Chimica, Materiali e Ingegneria Chimica “G. Natta”/
CIIRCO, Milano

## Abstract

Tunnel fire experiments are often based on a specific test condition such as air velocity, geometry or tunnel slope which may be different from the design conditions of an actual tunnel project. The capability of the Fire Dynamics Simulator (FDS) to reproduce the consequences of pool fires in confined environments, such as those in road tunnels, has been tested. The use of computational fluid dynamics (CFD) in this field can be useful especially for the development of fire protection systems and for designing adequate ventilation systems and the necessary escape routes.

Different tunnel fire scenarios, realized at different scales, have been considered in this study; it was evidenced the influence of different parameters on the smoke and fire dynamics. For all the analyzed cases, FDS has made a fairly good description of the experiments, evidencing a good agreement between model predictions and experimental measurements. The successful comparison of the code to the experimental results further support the use of this code for the simulation of fire dynamics and for the evaluation of the risk associated with fires in confined environments and in road tunnels.

## 1. Introduction

Accidental fires often develop in congested environments such as production areas, warehouses or urban areas [1]. Tunnel fires in particular can easily lead to catastrophic outcomes (the Mont Blanc, Gotthard and Tauern tunnel fires are a few examples) due to a combination of factors: high volume of flammable materials (fuel as well as freight especially if Heavy Good Vehicles, HGVs, are involved), availability of ignition sources (due to the overheating of the vehicles brakes or engines), potential involvement of many vehicles at the same time in a confined environment, the flow of toxic gas and smoke from the fire.

The potential consequences in terms of fatalities (of both users and rescue teams), as well as the economic loss related to property and damaged infrastructures (which often leads to prolonged closure of the tunnel) must be considered and highlight the need for a comprehensive risk analysis, as well as the implementation of properly designed prevention and protection measures, together with the development of adequate emergency procedures.

In this context, from the engineering point of view emphasis should be given to temperature profiles in proximity of the flame and within the plume of the hot toxic smoke, due to their consequences on the structural integrity of the enclosure and the accessibility of the escape routes.

For this reason, the ventilation system in a tunnel is of critical importance for both the ordinary and the emergency operations [2]. The project of a correctly performing ventilation system is highly dependent on the geometrical and physical configuration of the tunnel [3]; given the impossibility to perform a high number of experimental tests for each tunnel configuration, it is necessary to focus the attention on the simulation of fire dynamics. In this work, the performances of the FDS code have been tested using literature experimental data of tunnel fires [4,5].

## 2. Modeling approach

The Fire Dynamics Simulator (FDS) [6], developed at NIST, is a CFD model of fire-driven fluid flow. FDS solves numerically a form of the Navier-Stokes equations appropriate for the low-speed, thermally-driven flow with an emphasis on the smoke and heat transport from fires. FDS model solves the equations for the conservation mass, species, and momentum, taking into account conductive and radiative heat fluxes. The overall computation is treated as a Large Eddy Simulation (LES). The description of the numerical schemes used for the solution all equations is described in [7]. The geometry of the domain, mesh resolution, obstacles, boundary conditions, material properties and different simulations parameters are all inputs for the simulation. In the case of tunnel fire, boundary conditions are prescribed on the walls and vents. To improve the geometry representation, the BlenderFDS open-source graphical interface developed for FDS has been used [8]; BlenderFDS is based on Blender, an open-source 3D content creation suite. These tools allows the users to draw the geometry of the system directly into the 3D environment of Blender, thus converting it in an FDS input file which contains all the required information. This helps to reduce the time required to set up the case geometry as well as the possibility to quickly generate the computational grid.

## 3. Results and Discussion

The model predictions obtained using the Fire Dynamics Simulator (FDS) have been compared with experimental data related tunnel pool fires. First, a series of real-scale fire tests has been simulated, then, a series of tunnel fire experiments, carried out at the laboratory scale has been reproduced.

### 3.1 Fire scenario #1

Three experiments were performed by Apte et al. [4] to investigate the smoke flow under different ventilation rate in a reproduction of a real-scale mine tunnel (130 m long x 5.4m width x 2.4m high) with rectangular section. Airflow with a given speed was introduced into the tunnel by two exhaust fans installed at one end and a circular pool of n-octane (with a diameter of 1 m) was burnt in each test.

For the numerical analysis, the pool fire has been defined as a square size source term with an equivalent area; the estimated heat release rates (HRR) of the fire (ranging from 2 to 2.4 MW according to the experiment) have been used to describe the pool fire, so avoiding the description of the flame counteraction on the liquid pool, the fuel evaporation, etc. The grid cells close to the fire source has been modeled, after a series of tests, coherently with the Ma and Quintiere criterion [9], as cells with a length of 0.1 m, while at distance the grid has been conveniently stretched, resulting in a computational domain with more about 205 kcells (410k for the full 3D simulations). The forced longitudinal ventilation of the tunnel (0.5, 0.85 and 2 m/s) was assigned with a vent condition at a portal, and in all the simulations the fire input has been activated once a fully developed airflow was achieved in the tunnel.

Figure 1a shows the computational domain, while Tab. 1 evidences that the model is able to reasonable predict the flame tilt observed for these experiments.

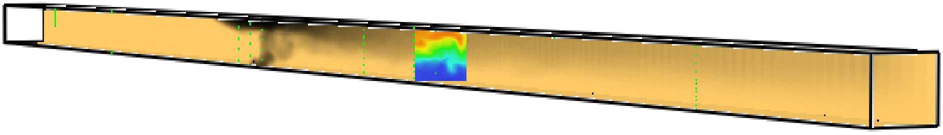

**Figure 1.** Sketch of the computational domain of the Apte tunnel fire.

**Table 1.** Comparison between the calculated and measured flame angles.

| Air flow speed [m/s] | Measured flame angle [10] | Predicted flame angle |
|---|---|---|
| 0.5 | 46° | 40° |
| 0.85 | 56° | 56° |
| 2.0 | 66° | 67° |

Moreover, the FDS code successfully predicts the effect of stratification, which means higher temperatures in the region close to the ceiling of the tunnel and the reduction in temperature observed at larger distances from fire. Good results have been obtained by modeling the wall with the physical characteristics of concrete (in terms of thermal conductivity, specific heat and emissivity); as reported in Fig. 2, a fairly good agreement between the model predictions and the experimental data has been found with this assumption for the boundary conditions. This behavior can be ascribed to the fact that in elongated tunnel the heat exchange through the walls plays a key role. With reference to the results of Fig. 2, a proper description of the wall properties helped to improve the capability of the code to reproduce the experimental data, as can be seen by the comparison between the profile obtained in this work and other literature predictions [11].

### 3.2 Fire scenario #2

One of the experiments recently performed in a laboratory-scale tunnel by Maschio et al. [5] has been modeled. A 6 m long tunnel, with an internal radius of 0.15 m, has been realized to reproduce a real highway road tunnel; the HRR of the small-scale fire has been defined according to the Froude similarity criterion.

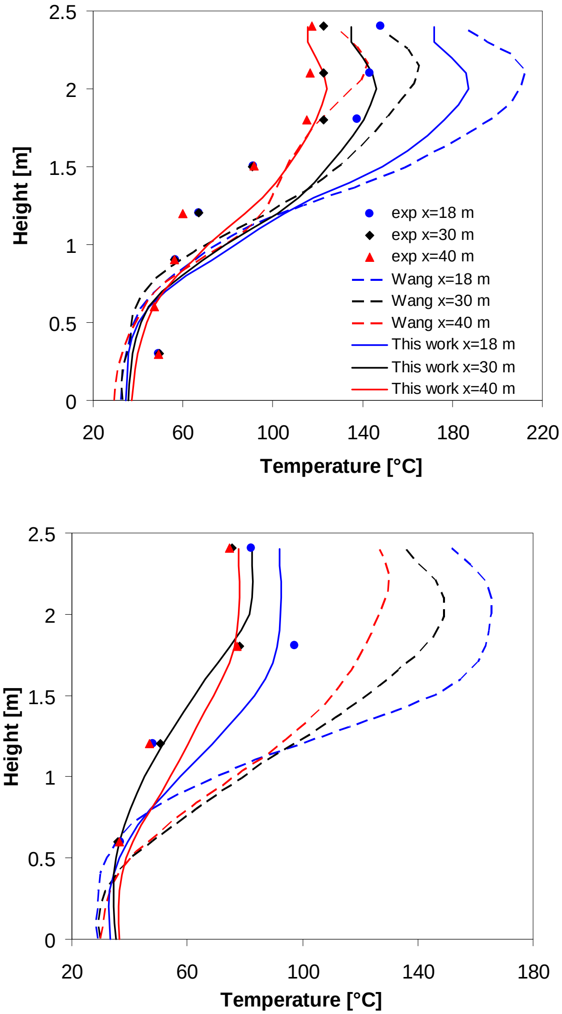

**Figure 2.** Comparison between predicted and measured temperature profiles for the runs performed at u=0.85 m/s (left) and u=2.0 m/s (right) [4]; a comparison with the predictions of Wang [11] has been also evidenced.

Blender has been used to prepare the curved tunnel geometry and the computational mesh for the FDS simulations (Fig. 3); in particular, a computational grid with cubic cells of 1 cm, uniformly distributed in the domain has been defined, thus resulting in a mesh of about 720 kcells. A pool fire scenario mimicking a real-scale fire with an approximate HRR of 4 MW has been modeled; this means that, at laboratory scale, only 12 mL of fuel were burnt in a pool with a diameter of 62 mm. Apart from the small size of the system, which allowed for obtaining quasi steady-state conditions into the tunnel only for 2-3 min, the main problem was the definition of the ventilation conditions to be set for the simulation. In the experiments, the authors allowed the establishment of a natural longitudinal ventilation driven from the flame that had non-stationary characteristics due to macroscopic movement of the flame itself.

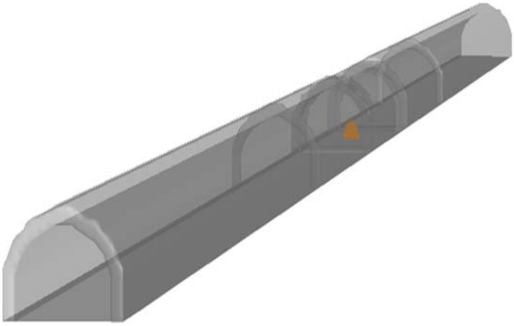

**Figure 3.** Graphical representation of the computational domain which represents the investigated lab-scale tunnel [5].

Poor information where available concerning the recorded airflow velocities; for this reason a sensitivity analysis has performed by testing different wind conditions. All the simulations have been carried out until steady-state profiles for temperatures and velocities are obtained; then, predicted temperatures have been compared with the average steady-state values measured by the thermocouples located at different distances from the tunnel ceiling. A satisfactory agreement between model predictions and experimental data has been obtained only for the simulations where a “no wind” condition is used for the FDS boundary conditions, thus allowing the hot plume produced by the flame to spread symmetrically into the laboratory-scale tunnel. Figure 4 compares model trends with the experimental data collected at different axial distances from the fire section (section C = 1.2 m and section D = 1.9 m).

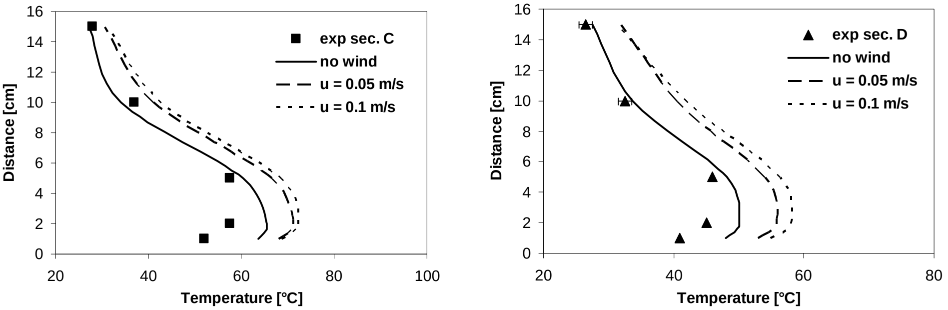

**Figure 4.** Comparison between model predictions and experimental measurements obtained at different distances from the tunnel for a fire scenario mimicking a real-scale tunnel fire with a HRR of about 4 MW; section C = 1.2 m, section D = 1.9 m downwind the pool fire.

Considering the uncertainties related to the lab-scale runs, the model represented fairly good the investigated experiment, however further simulations will be performed to gain a better understanding of the event dynamics.

## 4. Conclusions

In this work the CFD code FDS has been used to model different experimental datasets related to tunnels fires; for this kind of environments, it was found that it is essential to correctly define the boundary conditions of the computational domain. The model performed well in reproducing experiments that where carried out both at laboratory scale and at full-scale, with natural and forced ventilation; in particular, ventilation had a key role for the description of the hot smoke dispersion, strongly influencing both the backlayering and the destratification of the smoke.

## Acknowledgments

The authors wish to thank Politecnico di Milano for the financial support (Progetto 5 per mille Junior - CUP code D41J10000490001).

## References

[1] Pontiggia M., Landucci G., Busini V., Derudi M., Alba M., Scaioni M., Bonvicini S., Cozzani V., Rota R., Atmos. Environ. 45:3913-3923 (2011).

[2] Beard A., Carvel R., *The handbook of tunnel fire safety*, Telford Publishing, London, UK (2005).

[3] Grant G.B., Jagger S.F., Lea C.J., *Phil. Trans. Royal Soc.* 356:2873-2906 (1998).

[4] Apte V.B., Green A.R., Kent J.H., *Fire Safety Science* 3:425-434 (1991).

[5] Maschio G., Vianello C., Palazzi E., Fabiano B., *Chem. Eng. Transactions* 19:207-212 (2010).

[6] McGrattan K., McDermott R., Hostikka S., Floyd J., *Fire Dynamics Simulator (Version 5). User's Guide,* NIST Special Publication 1019-5 (2010).

[7] McGrattan K., Hostikka S., Floyd J., Baum H., Rehm R., Mell W., McDermott R., *Fire Dynamics Simulator (Version 5). Technical Reference Guide*, NIST Special Publication 1018-15 (2010).

[8] Gissi E., *Installing BlenderFDS* <code.google.com/p/blenderFDS> (2012).

[9] Ma T.G., Quintiere J.G., *Fire Safety J.* 38:467-492 (2003).

[10] Fletcher D.F., Kent J.H., Apte V.B., Green A.R., *Fire Safety J.* 23:305–325 (1994).

[11] Wang H.Y., *Fire Safety J.* 44:394–406 (2009).

10.4405/35proci2012.III3