CFD MODELING AND EXPERIMENTAL ACTIVITY ON REAL SCALE TUNNEL FIRES

alessio.frassoldati@polimi.it
* Dipartimento di Chimica, Materiali e Ingegneria Chimica “G. Natta”, Politecnico di Milano
**Dipartimento INDACO, Laboratorio Mobilità e Trasporti, Politecnico di Milano, Milano
*** Regione Autonoma Valle d’Aosta, Direzione dei servizi antincendio e di soccorso, Aosta
**** SITRASB S.p.A. – Società Italiana Traforo Gran San Bernardo
Italy

Abstract
Fires represent a very complex phenomenon involving combustion, radiation, turbulence, fluid dynamics and other physical and chemical processes. In this paper, we compare the model predictions of the Fire Dynamics Simulator (FDS) with experimental measurements of fires in road tunnels. The successful comparison of the predictions 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 road tunnels. Moreover, the results of a training exercise of firefighters in the Gran San Bernardo tunnel allowed to effectively evaluate of firefighting procedures, security teams activities, and showed the ability to produce a large amount of smoke to test procedures and materials in severe conditions. New measurement of temperature and flue gases compositions were also made.

Introduction
The severe fires in Europe, such as those of the Mont Blanc, Gotthard and Tauern tunnels, have clearly displayed the dramatic urgency of adapting the road and rail tunnels to higher safety standards. Fires in tunnels are a threat not just for the safety of users but also for rescue teams. Beside the presence of victims, even the economic consequences related to damage to infrastructure and often prolonged closure of the tunnel must be considered. Moreover, dangerous goods are transported through many of these tunnels. These issues push public authorities and tunnel designers to take increasing account of risks connected with fire. An important step towards safety in tunnels in Europe is the recent Tunnel Directive (2004/54/EC). According to this Directive, a risk analysis is required for existing tunnels that are considered below the minimum safety standards. Italy implemented the EC directive in 2006 with the Dlsg n° 264. Safety investigations are today well recognized as an essential element to improve transport safety. In a risk analysis procedure it is then particularly important to develop and combine proper models to calculate the flux of danger between the sources and the targets (people, infrastructure and environment). A key role can nowadays be played by computer simulations and fire models. Fire Dynamics Simulator (FDS) [1] is one of the
leading Computational Fluid Dynamics (CFD) models for fire modeling and is widely used in the field of fire protection engineering. To reach the goal of validating and improving the CFD codes, to obtain a better design of tunnels and their security systems, to perform risk analysis and evaluate the best strategies for intervention of rescue teams, it is useful to conduct tests using full-scale structures that can simulate a real event, with known boundary conditions (eg geometry, devices and systems installed, etc.). This need has led to a fruitful collaboration between researchers of the Politecnico di Milano, the Corpo Valdostano dei Vigili del Fuoco, and the SITRASB SpA, manager of the Gran San Bernardo tunnel. In this work, the performances of the FDS code is tested using literature experimental data of tunnel fires [2,3]. In the final part of the paper, we will describe the experimental activity recently performed in the Gran San Bernardo road tunnel, where a team of the firefighters of Valle d’Aosta performed one of the regular safety and training exercises. During this activity, not only the safety procedures and infrastructures of the tunnel were tested but it was also possible to measure the temperature, wind and CO levels in several locations inside the tunnel.

The Fire Dynamics Simulator (FDS) [1], 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 [4]. The geometry of the domain, mesh resolution, obstacles, boundary conditions, material properties and different simulations parameters are all inputs for the simulation. In a tunnel fire, boundary conditions are prescribed on the walls and vents.

**Fire Scenario #1 (Road tunnel Tests [2])**

The first fire scenario refers to the experimental activity performed in the Yunnan region in China [2]. Two tunnels were used, all the details of the tunnels geometry and experimental conditions are reported in table 1. Fig. 1 shows the shape of one of the tunnels and its corresponding computational domain. For this first test, we simplified the geometry and have constructed a two-lane road tunnel model with dimensions 10.8x22x8m. The fire source is represented by a pool with prescribed dimensions and Heat Release Rate (HRR), and placed in the tunnel just above the floor level as in the experiments. We referred to the work of Ma and Quintiere for the best practices about the creation of the computational mesh for a pool fire simulation using FDS [5,1]. The proper size of the cells of the computational grid is defined [5] on the basis of fire power and of the parameter $\lambda$, with $0.05 \leq \lambda \leq 0.1$.

$$\max(\delta x, \delta y, \delta z) = \lambda \cdot \left( \frac{Q}{\rho c_p T_0 \sqrt{g}} \right)^{\frac{2}{5}}$$
Figure 1. Tunnel [2], computational domain and grid (cell size ≈ 0.07 m).

Table 1. Real scale tunnel fire tests [2]

<table>
<thead>
<tr>
<th>Tunnel</th>
<th>Length [m]</th>
<th>Inclin. [%]</th>
<th>Pool fire size [m]</th>
<th>HRR [MW]</th>
<th>Longitudinal wind velocity [m/s]</th>
<th>Ambient Temperature [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3270</td>
<td>1.29</td>
<td>1x1 m</td>
<td>1.6</td>
<td>0.5</td>
<td>17.5</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1x2 m</td>
<td>3</td>
<td>0.8</td>
<td>17.5</td>
</tr>
<tr>
<td>2</td>
<td>1032</td>
<td>2.01</td>
<td>1x1 m</td>
<td>1.6</td>
<td>0.3</td>
<td>21.5</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1x2 m</td>
<td>3.0</td>
<td>0.7</td>
<td>25</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1x2 m</td>
<td>3.0</td>
<td>0.9</td>
<td>24</td>
</tr>
</tbody>
</table>

Table 2. Range for Cell size in tunnel fires of Table 1

<table>
<thead>
<tr>
<th>HRR [MW]</th>
<th>λ</th>
<th>Max cell size [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.6</td>
<td>0.05</td>
<td>≈ 0.07</td>
</tr>
<tr>
<td></td>
<td>0.08</td>
<td>≈ 0.10</td>
</tr>
<tr>
<td>3.0</td>
<td>0.05</td>
<td>≈ 0.12</td>
</tr>
<tr>
<td></td>
<td>0.08</td>
<td>≈ 0.18</td>
</tr>
</tbody>
</table>

Where $Q$ [MW] is the fire power, $\rho_\infty$ [kg/m$^3$] air density, $c_p$ [J/Kg/K] air specific heat, $T_\infty$[K] air temperature, $g$ [9.81 m/s$^2$], $D$ [m] the size of the pool [5]. For values of $\lambda$ larger than the upper limit, the height of the fire tends to be overestimated, while the reverse occurs when $\lambda$ becomes lower than 0.05. Table 2 shows the range of typical cell dimensions as a function of the pool fire power. The computational grid is shown in fig. 1. Figure 2 and 3 show the comparison of model predictions and experimental results.

Figure 2. Experimental [2] (left) and predicted (right) smoke temperature under the ceiling of the tunnel during the fire.
Fire Scenario #2 (Road tunnel test [3])

Apte et al. [3] studied transient characteristics of plume flows generated by 0.57 m-2.0 m diameter octane fuel pool fires in a 5.4 m wide x 2.4 m high x 130 m long tunnel for ventilation rates in the range of 0.5-2.0 m/s. This full scale test was also modeled using the FDS code. Figure 5 shows the computational domain, while figure 6 shows the comparison between predictions and experimental results [3] at two distances from the pool fire. As observed 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. The conditions simulated in Fig. 6 refers to a pool diameter of 1 m and a wind velocity of 0.5 m/s. Figure 7 shows that the predicted critical wind speed required to stop the backflow is 2.2 m/s (for the 2.4 MW fire). This value is good agreement with the experimental result of Apte et al. [3] (1.85 m/s).

Figure 5. Computational domain and position of the measurements. Mesh includes 122880 cells.

Figure 6. Comparison between predictions and experimental results [3] at two distances from the pool fire (fire power 2.4 MW).

Figure 7. Effect of wind speed on the Smoke movement for the 2.4 MW fire of Apte [3].
Fire Scenario #3 (San Bernardo tunnel – this work)
The tunnel, inaugurated on March 19, 1964, was the first road tunnel opened to traffic through the Alps. It has a length of 5798 meters and a roadway with two lanes. The southern entrance on the Italian side is located at an altitude of 1875 m above sea level and the Swiss side at 1918 m. The plants and traffic management of the Gran San Bernardo tunnel is made from two control rooms located in the South and North entrances of the tunnel. These rooms are active 24 hours on 24, and receive information about the different plants, alarms and emergency calls.

Figure 8. Planimetric map of the fire scenario at San Bernardo tunnel.

Figure 9. Position of the fixed and mobile instruments used to measure the fire (straw) dynamics during the firefighters training activity.

According to Directive 2004/54/EC, which imposes the organization of regular and realistic safety exercises corresponding to defined incident scenarios, a Binational safety exercise was performed on June 16, 2011 and its organization involved also the acquisition of experimental data using fixed and mobile devices: air speed, opacity, temperatures, concentrations of CO, NO, NO₂, SO₂, CO₂, NH₃ and HCN were measured. One of the most relevant element of a safety exercise is the ability to reproduce conditions as similar to the actual emergency situations, to better study the critical points of intervention procedures. In this case, the accident scenario simulated an accident between a semi-trailer of a heavy vehicle, and several light vehicles, with the presence of injured people (see Fig. 8). After the impact, the semi-trailer catches fire, causing a fire with smoke. The presence of fire, and then smoke, is able to efficiently test the alarm activation devices, the passive safety systems, such as the configuration of the emergency ventilation, and finally the rescuers’ teams, in their approach and fighting the fire with the means and resources available. We used a roll-off containers properly modified to simulate a fire of small size and limited power, but able to produce, by burning straw and wood, a large amount of smoke in the tunnel.

This exercise was a useful first case study for the definition of the critical tasks in the activities of measurement and data collection. Thanks to the availability and cooperation of partners involved, we are organizing a new exercise that it will be a trial independent of the traditional regular exercises, with the definition of a scenario specifically aimed at the project, using the precious feedback obtained
from this first experiment. This experimental activity will be used, together with the results of the successive experimental campaign, for CFD modeling purposes.

**Figure 10.** Measured wind speed during the test in the fire zone (anemometer in fig. 10).

**Figure 11.** Measured CO (ppm) during the test in several locations in the tunnel.

Figure 8 and 9 shows the map of the fire scenario and the position of the measuring instruments. Figure 10 shows that the initial wind speed (blowing towards the Swiss exit) was about 1.5 m/s and was effectively controlled, using the ventilation equipments, and reduced to almost 0 m/s during the test. The ventilation was restored at the end of the test. The effect on CO is quite evident (figure 11).

**Acknowledgements**

The authors wish to thank Politecnico di Milano for the financial support (Progetto 5 per mille Junior–CUP code D41J10000490001). We also acknowledge the fruitful cooperation with the Corpo Valdostano dei Vigili del Fuoco and thank the SITRASB SpA team for their hospitality and competence.

**References**


10.4405/34proci2011.III23