Numerical Analysis of Pool Fire Consequences in Confined Environments

Silvia Tavelli, Marco Derudi, Alberto Cuoci, Alessio Frassoldati

Politecnico di Milano, Dip. di Chimica, Materiali e Ingegneria Chimica "G. Natta", piazza Leonardo da Vinci 32, 20133, Milano
marco.derudi@polimi.it

In this paper, a critical comparison between model predictions obtained with the Fire Dynamics Simulator (FDS) and experimental measurements of pool fires in confined environments, such as those in road tunnels, has been performed. 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 defining necessary escape routes.

For this reason, different pool fire scenarios have been considered; preliminarily, several simulations have been carried out taking as a reference few experimental tests related to small pool fires in confined environments available in the literature. These tests have been performed in rooms with different geometries, evaluating the influence of the fire position on the smoke and fire dynamics, the role of fire heat release, but also of the presence of doors and windows; this allows to validate the computational code and to perform a sensitivity analysis on the role of different parameters such as the grid size, the boundary conditions, the choice of the computational domain, etc.

Then, experimental tests obtained in a full-scale road tunnel have been modeled; for all the analyzed cases, FDS has made a good description of the experiments, evidencing in particular a good agreement between model predictions and experimental measurements. The successful comparison between the model and 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 (Pontiggia et al., 2011); fires may as well grow in enclosures such as rooms and warehouses, or involve the transportation of goods in road or railway tunnels. All these situations, if not properly handled, may lead to severe consequences in terms of loss of life and property, thus highlighting the need for a more comprehensive study of these phenomena form the safety point of view.

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.

The recent trends towards building larger tunnels with complex geometries, as well as the increase in traffic to levels which often are well above the estimated project levels, highlight the need for an adequate methodology for tunnel safety decision making (Ortolani, 2007).

In this context, the aim of mathematical models which describe the evolution of compartment fires is to identify the thermal stress to which elements and structures would be exposed in the event of fire in a particular enclosed space, thus providing an alternative to current standards and regulations for design purpose. 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 (Beard and Carvel, 2005). The project of a correctly performing ventilation system is highly dependent on the geometrical and physical configuration of the tunnel (Grant et al, 1998); 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.

2. Fire Modelling Approach

Fire represent a very complex phenomenon involving various chemical and physical processes, such as combustion reactions, heat exchange (radiation, conduction and convection), fluid dynamics and turbulence. Given a fire source with defined heat release rate, the growth of the flames and the combustion products flow are controlled by the buoyancy forces created by the energy released by the fire, and also influenced by the heat transfer to the surroundings and the turbulent mixing of the gaseous fuel and the combustion products with the local atmosphere surrounding the fire; these aspects strongly depend on the geometry of the system and the ventilation characteristics (Beard and Carvel, 2005).

Zone models are commonly adopted in risk analysis to study accidents involving fires in confined environments, such as simple rooms and buildings; they rely on the assumption of a vertical subdivision of the room in two homogeneous zones (an hot layer with combustion products near the ceiling and a cold layer with fresh, clean air on the bottom). Fires in tunnels originates complex three dimensional flows of combustion products, which rise up to the ceiling of the tunnel and then expand forwards and backwards (backlayering), depending on the interaction between buoyancy and the ventilation conditions; thus, zone models are of little use when there is the need to analyze the spreading of the hot combustion products at the tunnel ceiling level, since they are not able to describe the backflow or the turbulence and instability that downwind of the fire causes the smoke to spread to the whole section (Chow, 1996).

The need for a flexible tool in terms of boundary conditions as well as accuracy of the phenomenological descriptions led to focus on computational fluid dynamics (CFD) techniques. This approach allows to model the complex physical interactions which occur simultaneously in a fire accident, helping to understand their influence on the event evolution.

In this study, different fire scenarios have been analyzed using the Fire Dynamic Simulator (FDS), an open source CFD model of fire driven fluid flow. The code solves numerically a form of the Navier-Stokes equations appropriate for low speed, thermally-driven flows, coupling them with submodels which account for approximate descriptions of the combustion reactions and the soot formation, with emphasis on convective and radiative heat transport from. Turbulence is described with the Smagorinsky form of Large Eddy simulation (LES): the eddies that account for most of the mixing are large enough to be calculated with reasonable accuracy from the equations of fluid dynamics, while the small scale eddy motions are accounted for with a sub grid scale model (McGrattan et al., 2010a).

It is possible to represent the geometry of the system, define boundary conditions as well as to add sprinklers and smoke detectors to the simulation, simply by describing their position and describing their properties in the Cartesian coordinate system. All the geometrical entities are divided into rectilinear grid cells, and their description is achieved by defining their coordinates in the computational domain in a text input file: thus, its application to the description of complex geometries, such as tunnel scenarios with curved walls obstacles and by-passes, becomes extremely cumbersome.

To overcome this limitation, the BlenderFDS open-source graphical interface for FDS has been developed Gissi (2012); 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 and is available to other further refinements or modifications. Compared to the use of FDS on its own, the tool provides a quick representation and an immediate evaluation of the system geometry, hence strongly reducing the time required to set up the case geometry as well as the possibility of quickly generate the computational grid.

3. Results and Discussion

The model predictions obtained using the Fire Dynamics Simulator (FDS) were compared with experimental data related to pool fires in confined areas. First, a series of room fire scenarios was simulated to test the model performances in reproducing the accident dynamics in a small and quite simple environment; then, a series of tunnel fire experiments, carried out at both laboratory and real scale was reproduced.

3.1 The Steckler Room Fire Tests

The first set of simulations reproduced a series of room fire tests conducted by Steckler et al. (1982) to study the flow induced by steady state methane burner fires in enclosures under the characteristic conditions of the developing fire. Temperature and air velocity measurements were taken at the door opening and in a corner of a 2.8 x 2.8 x 2.18 m^3 test room with adiabatic walls for experiments with different heat release rates (HRR), door widths and fire locations. A sketch of the enclosure domain with the computational grid and a typical representation of the fire and smoke plume is shown in Figure 1. As shown in Figure 2, measured temperature and velocity profiles were correctly reproduced by the model for different fire tests configurations.
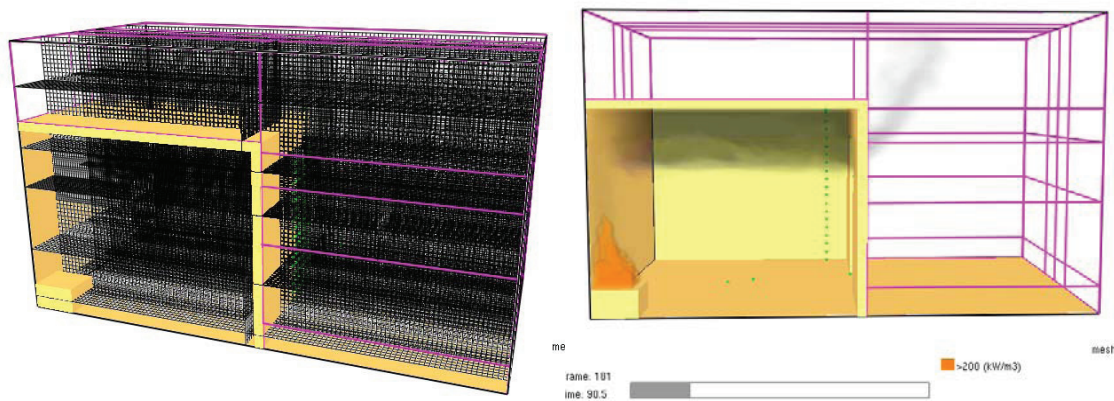

Figure 1: Representation of the room for fire tests: computational grid (left) and smoke plume dispersion in a test with the fire located in the room corner (right).
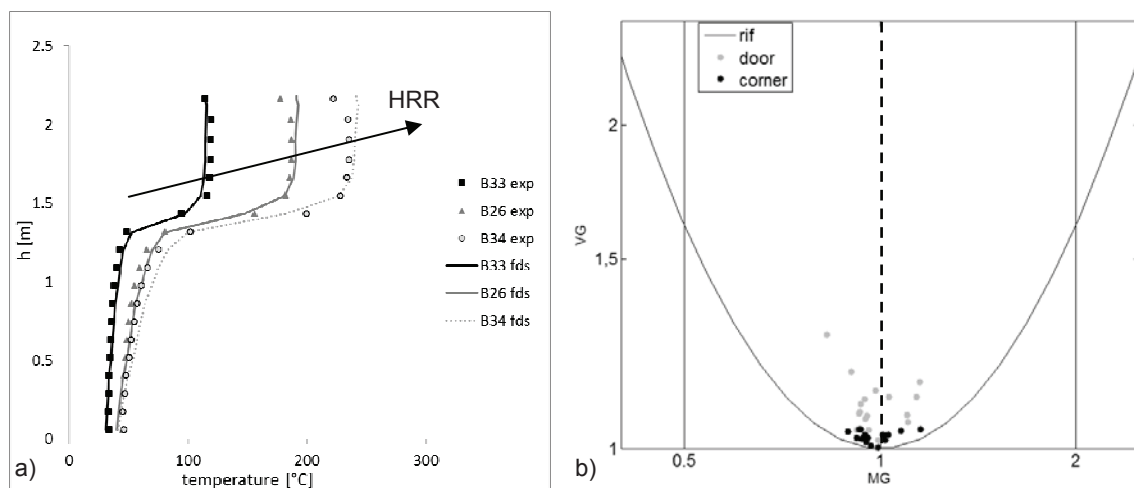

Figure 2: a) Comparison between experimental (exp) and model (fds) results for thermal profiles obtained in the room corner, for fire tests with increasing HRR (B33 = 31.6 kW, B26 = 62.9 kW, B34 = 103.6 kW); b) Plot of geometric mean bias (MG) versus geometric variance (VG) for door and corner temperature values related to different Steckler fire experiments. The vertical lines at MG=0.5 and MG=2 represent a factor of two over prediction and under prediction, respectively; the parabola indicates minimum VG for a value of MG due to systematic bias.

In particular, the air and smoke flow pattern generated by the fire in the compartment were well described by the model, as evidenced by the accurate prediction of the sharp temperature boundary (layer interface) due to thermal stratification between the relatively hot upper layer and cooler lower layer (Figure 2a). Considering all the fire tests modeled, a statistical performance indicator was used to evaluate the performance of this modeling approach against the available experimental dataset, as suggested by Hanna et al. (1993). Geometric mean bias (MG) and geometric mean variance (VG) were computed for door and corner thermal profiles; for a modeling approach to perfectly match the experimental data, the values of MG and VG should be equal to 1. With reference to Figure 2b, the parabola represents the minimum possible value of VG corresponding to a particular MG due to systematic bias. All points must lie within or above the parabola. This plot confirms the good model predictions, particularly for data that are collected in the room corner, where less temperature fluctuations were evidenced during the experiments.

A grid sensitivity study was also conducted and helped verify that the criterion described by Ma and Quintiere (2003) can be considered appropriate for the definition of the cell size in proximity of the fire (cubical, 5 cm cells) , while at distance from the fire source it is possible to conveniently increase the grid dimension, thus reducing the computational time without lack of accuracy of the predictions.

3.2 The Apte Tunnel Fire Tests

Three experiments were performed by Apte et al. (1991) to study 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 uniformly 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 ignited in each test.

For the numerical analysis, a square size source term, with a surface equivalent to that of the liquid n-octane pool was considered; the estimated HRR of the fire (ranging from 2 to 2.4 MW according to the experiment) was 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 were modeled, after a series of tests, coherently with the Quintiere criterion as cubes with sides of 0.1 m, while at distance the grid was conveniently stretched (McGrattan et al., 2010b), resulting in a computational domain with more than 200 kcells (400k 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 sources were activated once a fully developed turbulent flow was achieved in the tunnel.

A preliminary comparison between simulations representing half of the computational domain (with a mirror condition on the tunnel axis) and full 3D simulations showed results with minimal differences, encouraging to choose to simulate only half of the domain, hence reducing the overall computational efforts. This analysis is not pointless, since in an LES model there is no time-averaging built into the equations, and there is no time-averaged symmetric solution. Putting a symmetry boundary along the centerline of a fire plume could change its dynamics entirely, so it is usually not recommended. However, for this kind of problems, considering also that the pool fire is described with an approximate source term, the symmetry assumption can be conveniently adopted (Figure 3a).

It was found that the definition of the tunnel wall properties has a paramount importance in tunnel fires simulation; by using the adiabatic wall approximation, that proved to be accurate for the room fire tests, the predicted temperatures into the tunnel were too high compared to the experimental data, while the isothermal wall assumption, that can be reasonable for the floor and for the walls far from the flame, resulted in an underestimation of the temperatures particularly within the smoke hot layer.
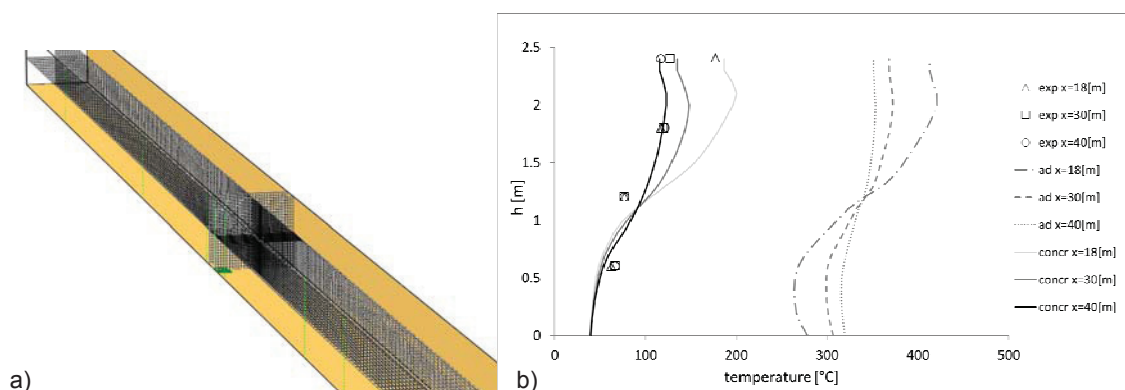

Figure 3: a) detail of the computational domain of the Apte tunnel fire; b) Temperature profiles for different wall characteristics (ad=adiabatic, concr=concrete) compared with experimental results (U=0.5 m/s).

Figure 4: Estimated effect of the forced ventilation on the smoke backlayering for the Apte tunnel fire tests (from top to bottom, U=0.5 m/s, U=0.85, and U=2.0 m/s).

After several tests, good results were obtained by modeling the wall with the physical characteristics of concrete (in terms of thermal conductivity, specific heat and emissivity); as reported in Figure 3b, a fairly good agreement between the model predictions and the experimental data was found with this assumption for the boundary conditions. This behavior can be ascribed to the increased contribution of the heat exchange through the walls due to the elongated geometry of the tunnel.

As Apte et al. (1991) carried out experiments at different airflows, the effect of the forced ventilation on the smoke dispersion into the tunnel was also modeled (Figure 4). An increased value of the ventilation velocity results in better dilution of the combustion products, lower temperatures at the ceiling and reduced backlayering; at low airflows (U=0.5 m/s) a strong backflow of hot smoke was determined upwind the pool fire, while a marked smoke destratification (with hot combustion products that mix together with fresh air and spread in the whole section of the tunnel) was observed in the downwind region. A further series of simulations with different ventilation velocity input, allowed to identify the threshold value for the absence of smoke backflow: this critical velocity, an useful design parameter, was identified as 2.2 m/s.

3.3 Tunnel fire tests at lab-scale

A series of experimental runs was recently performed in a small-scale road tunnel by Maschio et al. (2010); the tunnel curved shape and the fire source features were realized with the aim to reproduce at the laboratory scale full-scale tunnel fires (Vianello et al., 2012). For this reason, a 6 m long tunnel, with an internal radius of 0.15 m, was built reproducing a real highway road tunnel; the heat release rates of the small-scale fires were chosen according to the Froude similarity criterion.

As previously discussed, Blender was used to build the tunnel geometry, as well as to analyze different meshing strategies before to run FDS simulations. In particular, a pool fire scenario mimicking a real-scale fire with an approximate HRR of 4 MW was modeled; this means that, at laboratory scale, only 12 mL of fuel were burnt in a pool with a diameter of 62 mm, so a computational grid with cubic cells of 1 cm, uniformly distributed in the domain was selected, thus resulting in a mesh of about 720 kcells. A sketch of the tunnel geometry, with the representation of temperature and velocity monitors (symbols), is shown in Figure 5a. 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. During the experiments, Vianello et al. (2012) observed very irregular thermal profiles on the flame section, strongly influenced by flame fluctuations; moreover, they 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.
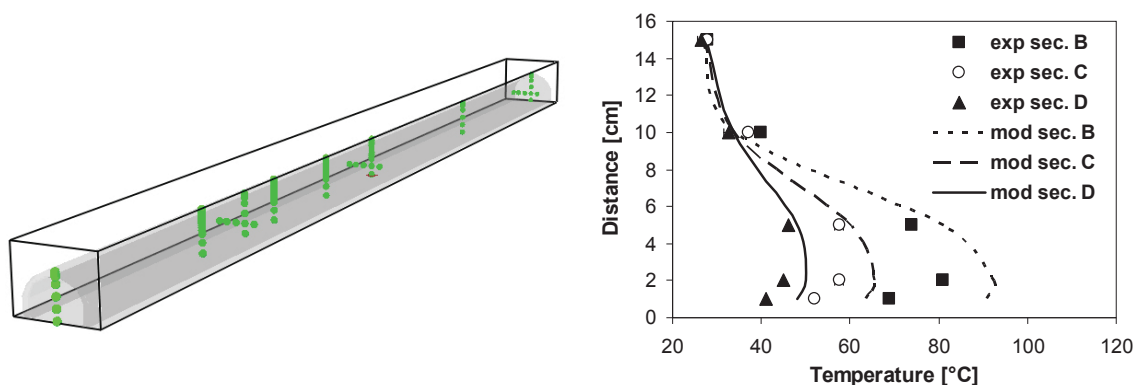

Figure 5: a) Sketch of the computational domain which represents the investigated lab-scale tunnel for fire tests; b) Comparison between model predictions and experimental temperatures measured at different distances from the tunnel ceiling for the 62 mm pool fire scenario.

Due to the lack of information about the recorded wind velocities, a sensitivity analysis was performed by testing different wind conditions. All the simulations were carried out until steady-state profiles for temperatures and velocities were obtained; then, predicted temperatures were compared with the average steady-state values measured by the thermocouples located at different distances from the tunnel ceiling. It was found that a satisfactory agreement between model predictions and experimental data can be obtained only if a "no wind" condition is used for the CFD simulations, thus allowing the hot plume produced by the flame to spread symmetrically into the small-scale tunnel. Figure 5b compares model trends with the experimental data collected at different axial distances from the fire section (section B = 0.6 m, section C = 1.2 m, section D = 1.9 m). Considering the uncertainties related to the lab-scale runs, the model represented fairly good the investigated experiment, however further simulations will be performed to achieve a better understanding of these fire tests, trying to reproduce also the event dynamics.

4. Conclusions

The severity of the fire accidents in congested environments such as tunnels in recent years has led to an increased attention towards the study of the phenomenon and its consequences.
In this work the CFD code FDS was used to model different experimental datasets related to congested environments and tunnels; for this kind of environments, it was found that it is essential to correctly define the boundary conditions of the computational domain. The use of both Blender and FDS allowed to reproduce in detail the domain geometry, thus improving the possibility to better reproduce the experimental runs. 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. All this issues encourage the use of computational fluid dynamics as a convenient tool for a better understanding of fires accidents, with applications for the development of fire protection systems and for designing ventilation systems for both ordinary and emergency situations.

Acknowledgments

References

Apte V.B., Green A.R., Kent J.H., 1991, Pool Fire Plume Flow in a Large-Scale Wind Tunnel, Fire Safety Science Proceeding of the 3rd International Symposium, 425-434.

Beard A., Carvel R., 2005, The handbook of tunnel fire safety, Thomas Telford Publishing, London, UK.

Chow W.K., 1996, Simulation of tunnel fires using a zone model, Tunneling and Underground Space Technology 11, 221-236.

Gissi E., 2012, Installing BlenderFDS <code.google.com/p/blenderFDS>, accessed 20.03.2012.

Hanna S.R., Chang J.C., Strimaitis D.G., 1993, Hazardous gas model evaluation with field observation, Atmospheric Environ. 27, 2265-2285.

McGrattan K., Hostikka S., Floyd J., Baum H., Rehm R., Mell W., McDermott R., 2010a, Fire Dynamics Simulator (Version 5). Technical Reference Guide, NIST Special Publication 1018-15, US Government Printing Office, Washington, USA.

McGrattan K., McDermott R., Hostikka S., Floyd J., 2010b, Fire Dynamics Simulator (Version 5). User's Guide, NIST Special Publication 1019-5, US Government Printing Office, Washington, USA.

Grant G.B., Jagger S.F., Lea C.J., 1998, Fires in tunnels, Phil. Trans. Royal Soc. 356, 2873-2906.

Ma T.G., Quintiere J.G., 2003, Numerical simulation of axi-symmetric fire plumes: accuracy and limitations, Fire Safety Journal 38, 467-492.

Maschio G., Vianello C., Palazzi E., Fabiano B., 2010, Experimental study on thermal and toxic hazards connected to different accident scenarios in road tunnels, Chemical Engineering Transactions 19, 207-212.

Ortolani C., 2007, Cases of accidental combustion, Maggioli Editore, Milano, Italy (in Italian).

Pontiggia M., Landucci G., Busini V., Derudi M., Alba M., Scaioni M., Bonvicini S., Cozzani V., Rota R., 2011, CFD model simulation of LPG dispersion in urban areas, Atmospheric Environ. 45, 3913-3923.

Steckler K.D, Quintiere J.G., Rinkinen W.J., 1982, Flow induced by fire in a compartment, National Bureau of Standards, Washington, USA.

Vianello C., Fabiano B., Palazzi E., Maschio G., 2012, Experimental study on thermal and toxic hazards connected to fire scenarios in road tunnels, J. Loss Prevention Process Industries 25, 718-729.