Coupled numerical simulation of fire in tunnel

Matteo Pachera[1], Francesco Pesavento[2], Bernhard A. Schrefler[3], Dariusz Gawin[4], Arkadiusz Witek[5]

[1] *Dept. of Industrial Engineering, University of Padova*
Via Gradenigo 6/a - 35131 Padova, Italy
e-mail: matteo.pachera.mp@gmail.com

[2,3] *Dept. of Civil, Environmental and Architectural Engineering*
Via F. Marzolo 9, 35131 Padova, Italy
e-mail: francesco.pesavento@dicea.unipd.it; bernhard.schrefler@dicea.unipd.it

[4,5] *Dept. of Building Physics and Building Materials, Technical University of Łódź*
Al. Politechniki 6, 93-590 Łódź, Poland
e-mail: gawindar@p.lodz.pl

Abstract

In this work a coupling strategy for the analysis of tunnel under fire is presented. This strategy consists in a "one-way" coupling between a tool considering the computational fluid dynamics and radiation with a model treating concrete as a multiphase porous material exposed to high temperature. This global approach allows taking into account in a realistic manner the behaviour of the "system tunnel", formed of the fluid and the solid domain (i.e. the concrete structures), from the fire onset, its development and propagation to the response of the structure. The thermal loads as well as the moisture exchange between the structure surface and the environment are calculated by means of the computational fluid dynamics. These set of data are passed in an automatic way to the numerical tool implementing a model based on Multiphase Porous Media Mechanics. Thanks to this strategy the structural verification is no longer based on the standard fire curves commonly used in the engineering practice, but it is directly related to a realistic fire scenario. To show the capability of this strategy some numerical simulations of a fire in the Brenner Base Tunnel, under construction between Italy and Austria, is presented. The numerical simulations show the effects of a more realistic distribution of the thermal loads with respect to the ones obtained by using the standard fire curves. Moreover, it is possible to highlight how the localized thermal load generates a non-uniform pressure rise in the material, which results in an increase of the structure stress state and of the spalling risk. Spalling is likely the most dangerous collapse mechanism for a concrete structure. This coupling approach still represents a "one way" strategy, i.e. realized without considering explicitly the exchange of boundary conditions from the structure to the fluid. This results in an approximation, but from physical point of view the current form of the solid-fluid coupling is considered sufficiently accurate in this first phase of the research.

Keywords: fire in tunnel, coupled strategy, CFD, radiation, concrete, multiphase porous material

1. Introduction

For the assessment and verification of the safety level of tunnel structures exposed to fire, several curves reproducing the temperature evolution induced by a fire are defined in the European standards. This approach allows for designing safe structures without linking the possible fire scenario to the structural verification. It is important to improve such approach in order to provide thermal loads for the structural analysis which are directly related to the fire scenario. For the simulation and the modelling of a fire, several approaches are available, the most accurate but computationally expensive one is the computational fluid dynamics (CFD). This allows to solve numerically the Navier-Stokes equations and to predict the thermal loads acting on the structure. Then, these ones can be used in combination with the mechanical loads for the structural verification. On the other hand, for a proper modelling of concrete it is necessary to consider its porous nature, i.e. a porous material in which the inner voids are filled partly with liquid water and partly with a gaseous mixture of vapour and dry air. This means that also heat and mass transfer must be taken into account as described in [1,2].

2. A coupled tool for the simulation of fire in tunnel

In order to provide a realistic set of boundary conditions to perform the structural analysis, it is necessary to model the effects of the fire. The flow field induced by a fire can be analysed starting from the Navier-Stokes equations, which are always solved numerically by CFD codes for practical problems. In this work, for the specific case of the fire scenario description, the code Fire Dynamic Simulator (FDS) is used.

The total heat flux on a solid surface results from the balance between incoming and outcoming fluxes, considering both the radiative and the convective heat fluxes. In FDS the radiative heat flux is calculated solving the radiation transport equation (RTE), by means of a finite volumes approach. The convective heat transfer is calculated simply through the Newton law, in which the convective heat transfer coefficient is assessed on the base of some simplified correlations.

The water vapour density can be evaluated by tracking the species: its density near the surface of the structure can be used as boundary condition.

The structural analysis is carried out with a home-made FEM code, Comes-HTC, which was specifically developed for the analysis of concrete structure at high temperature. The mathematical model used for the analysis of concrete behaviour in fire conditions is described in detail in [1,2]. Concrete is modelled as a multi-phase porous material in which pores are filled partly with liquid water and partly with a gas phase. The model considers the physic-mechanical interactions between phases, the exchange of mass, energy and linear momentum between phases and their components, the most important chemical reactions (e.g. dehydration), thermo-chemical and thermo-mechanical degradation processes that can take place in concrete under such severe conditions, various mass and energy transport mechanisms (among others: convection, advection diffusion on different forms, etc.).

Here, a "one way" coupling strategy for solving the fluid and solid domain in a tunnel under fire is presented. This implies that the results from the FDS simulations are used to compute the boundary conditions of the structural analysis in Comes-HTC. The one way coupling neglects the effects of the solid domain (i.e. the concrete structure) on the fluid. Of course this is an approximation but it can be considered as acceptable and a step forward, since in FDS a simple pure conductive model is embedded to calculate the wall temperature.

The water vapour mass exchange between the domains is controlled by the water density in the fluid close to the wall and by its value on the structure surface: the first one is evaluated in FDS, while the second one in the Comes-HTC code.

The incident heat flux, radiative plus convective, is calculated in FDS and then is imposed on the concrete surface as boundary condition, while outcoming flux is evaluated basing on the wall temperature calculated using Comes-HTC.

$$q_n = q_{FDS,in} - q_{HTC,out} = q_{FDS,in} - \left(\varepsilon\sigma T^4_{w,HTC} + hT_{w,HTC}\right) \quad (1)$$

where $q_{FDS,in}$ is the incident flux on the vault surface, h is the convective heat exchange coefficient and $T_{w,HTC}$ is the wall temperature. The incoming heat flux in FDS is calculated by means of the so-called adiabatic surface temperature AST [3].

Both the heat and water mass transfer are controlled by the conditions on the wall evaluated in Comes-HTC and in the fluid near the wall evaluated in FDS. Therefore, Robin-type boundary conditions are used on the boundaries. For a more detailed description of the coupling strategy adopted, the reader can refer to [3].

3. Numerical application

The coupled tool briefly described in the previous section was applied to a real case: the base tunnel under construction between Italy and Austria, known as Brenner Base Tunnel. It is the central part of the Berlin–Palermo corridor which runs for 2,200 km from North to South and is also known as the TEN-1 axis. The length of the tunnel is of 55 km (excluding the local network in Innsbruck). In the simulation, the fire source is a rail coach, 20.8 m long and 3.2 m wide, whose heat release rate curve has been estimated based on experimental data. The fire has a peak of 50 MW after 15 minutes and lasts 60 minutes, the fire place has been placed at 3.0 m high and the gap between the fire and the ceiling is about 4.2 m. The roof of the rail coach has not been included in the model because this can act as screen for the radiation and reduce the thermal load on the vault. Due to the lack of information about the ventilation strategy, the tunnel is assumed to be naturally ventilated, this scenario is not realistic, however this is the most severe for the structure. The main results for the fluid and the solid domain (i.e. the concrete vault) are shown in Figures 1-3. See [3] for further details.

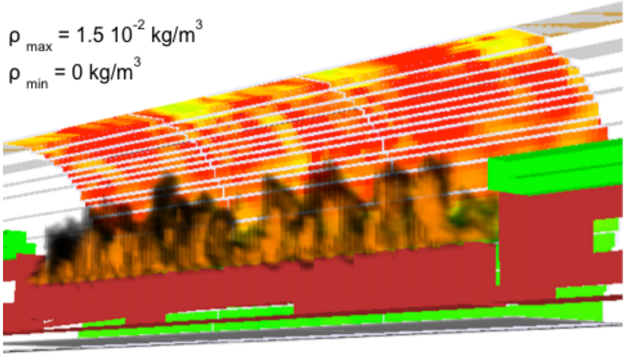

Figure 1. Adiabatic surface temperature distribution near the fire after 15 min.

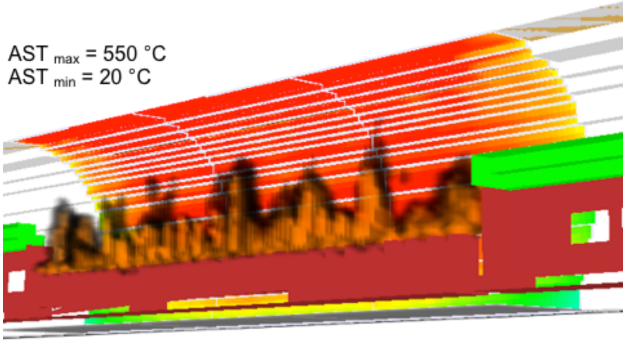

Figure 2: Water vapour density distribution near the fire after 15 min.

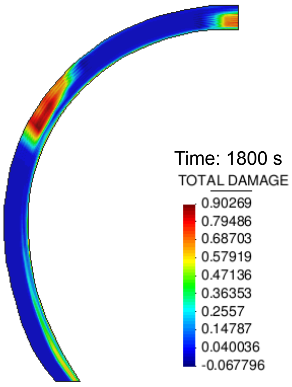

Figure 3: Total damage distribution in the concrete vault after 15 min.

References

[1] Gawin, D., Pesavento, F., Schrefler, B. A., Modelling of hygro-thermal behaviour of concrete at high temperature with thermo-chemical and mechanical material degradation, *Comput. Methods Appl. Mech. Engng,* 192(13), pp. 1731-1771, 2003.

[2] Schrefler, B.A., Codina, R., Pesavento, F., Principe, J., Thermal coupling of fluid flow and structural response of a tunnel induced by fire, *Int. J. Num. Meth. Engng*, 87(1-5), pp. 361-385, 2011.

[3] Pachera, M., *Numerical simulations of fires in road and rail tunnels with structural and fluid dynamic analysis*, PhD Thesis-University of Padova, 2017.