

WATER MIST FIRE PROTECTION – RESEARCH ON A NUMERICAL TOOLBOX FOR SYSTEMS EFFECTIVENESS ASSESSMENT

Luca Iannantuoni*, Giovanni Manzini*

Politecnico di Milano - Department of Energy, via Lambruschini, 4, 20156 Milano (Italy),
Ph. +39 02 23993874, e-mail giovanni.manzini@polimi.it

ABSTRACT

There is a considerable interest in the potential of water mist systems for new fire safety applications due to the great effectiveness that, in some cases, the strategies using these systems have. The heat and mass transfer capabilities effectiveness of water mist droplets clusters allows faster heat absorption and water evaporation and therefore they can be very effective for temperature mitigation, fire control and suppression. But, their important feature to remain suspended for long periods and to act as a total flooding agent can be compromised by the air velocity field imposed by ventilation.

The trade-off between natural/ forced ventilation and water mist systems characteristics must be investigated experimentally, but the current tendency is to use few and very significant experimental results in order to improve the development and validate numerical models capable to simulate the complex dynamics of such fire scenarios.

The research program described is focused on the development first stage of a numerical tool to help engineers in the work of elaborating and optimizing effective fire protection strategies that use these systems; it is based on Computational Fluid Dynamics (CFD) but it is not be limited exclusively to such methodology. In fact, because of the high computational cost of 3D CFD simulations, its domain is limited to a section of the scenario, large enough to capture the most important phenomena, while the boundary conditions of the CFD model domain are obtained coupling it with dynamic fast running numerical tools (1D and 3D), capable of reproducing the rest of the environment and the effect of operating fans, located far from the fire source. So the primary aim of the model is to integrate “large scale” models (a 1D tool that is developed using Python, while a 3D fast running one that was developed in the past) with a CFD sub-models engine (OpenFOAM framework) capable to allow the simulation of main phenomena in fire scenarios with water mist injections and strong ventilation.

Keywords: water mist, ventilation, numerical simulation, large scale modeling, CFD.

INTRODUCTION

Our past studies were focused mainly on verifying the capability of longitudinal and transversal fire ventilation strategies to prevent the spread of smoke along the tunnels. To do this, in order to acquire all the needed elements to clarify the complex phenomena related to the fire scenarios object of our interest, we did a bibliographical research to select and assess mathematical tools for the preliminary analysis and focusing mainly on the critical velocity determination (the minimum air velocity needed to prevent smoke back-layering), useful on the evaluation of smoke confinement resulting from ventilation system activation. The subsequent phases, concerning the appliance of computational codes, included a first set of simulations computed by 1D/ one-dimension (SES [1]) and 3D/ three-dimension tools (ECART [2], FDS [3]). The aim of these simulations was to estimate the effectiveness of different fire protection strategies (with the activation of only ventilation systems or both ventilation and deluge water mist systems) to mitigate the consequences of fires with a focus mainly on the smoke propagation and air temperature mitigation. The analyses were focused on driver-less underground metro lines (single and double track subway), rail tunnels (single bore) and urban road tunnels (double bore) evaluating different airflows, ventilation strategies and different geometrical dimensions of tunnels, stacks and ventilation chambers that were considered. For example, many different ventilation strategy were analyzed and compared for underground metro lines: extraction from only one shaft, push-pull from two or more shafts across the tunnel fire. Different airflows given for each ventilation shaft, different location of the shafts, effects of tunnel jet fans and tunnel semi-transversal ventilation (longitudinal flow + transversal flow) were also analyzed for typical scenarios.

In particular, the results of our analysis showed that values of temperatures and combustion products concentrations along tunnels, corridors, etc. met the safety criteria, using just ventilation systems, only in some cases, and that for many fire scenarios, in particular for higher design fires (e.g. in some cases $HRR > 10$ MW), only those strategies

that used both ventilation systems and water mist deluge systems would be very effective. This results convinced us to consider very interesting and very challenging a more accurate numerical analysis of the problem, but considering only as final stage the development and validation of a multiphase combustion model capable to simulate also fire control and suppression by water sprays.

The research work summarized here is focused on the first stage of development, verification and validation of a numerical tool capable to simulate fire scenarios in order to perform water mist protection strategies effectiveness assessment. In particular our effort is to take into account strategies applied in tunnel fire scenarios, hence considering strongly ventilated compartments, to verify their effectiveness in terms of safety criteria chosen to evaluate the maximum egress time available for people. The effectiveness of the water mist control and suppression systems derive from a synergy with the ventilation systems which, for long tunnel, is not simply treatable with CFD analysis.

The interest in this task raise from the increasing number of applications in road and subway tunnels and from the uncertainty associated mainly with the interaction between water mist droplets, hot smoke and ventilation flow. Indeed, these phenomena are very important to maintain adequate safety levels for the occupants and the fire brigades in terms of temperature, visibility and concentration of combustion products, and so their evaluation is of great importance to estimate the consequences of a strategy against fire. A second important aspect is the interaction among combustion, which is not treated because is beyond our present purpose.

Besides, we highlight that the experimental activities on real scale tunnel fire scenario are very expensive and that parametric studies are necessary to perform a good effectiveness assessment, because of the great number of independent variables affecting the scenario dynamics.

We first evaluate the problem of simulating the water mist spray in a lagrangian framework, and we give evidence of the limitation of such approach for large domain with a high number of injectors. In fact the computational requirement of a lagrangian-tracking approach in order to achieve meaningful average information on the field is very high and so we are now implementing an eulerian-eulerian model.

In this first stage of our project, the activity has focused more on aspects of modeling and tool development and we have chosen the OpenFOAM CFD toolbox [4] to simulate the multiphase domain (turbulent buoyancy driven combustion, droplets heat, mass and momentum transfer, ...) [5], and we have implemented a Python based - 1D network (www.python.org) model capable to simulate the entire tunnel system in order to give the proper boundary condition to the CFD problem.

NUMERICAL TOOL

Modeling

We are developing our CFD model using OpenFOAM, which uses finite volume methods to solve systems of partial differential equations ascribed on any 3D unstructured mesh of polyhedral cells. OpenFOAM is supplied with numerous pre-configured solvers, utilities and libraries and so it can be used like any typical simulation package. It is open in terms of source code and its structure and hierarchical design makes the solvers, utilities and libraries fully extensible. The built-in libraries comprise a Lagrangian particle tracking library which provides tracking and coupling with the Eulerian phase, allowing for simple implementation of spray and other multi-phase modeling. The standard OpenFOAM release includes a complete set of atomization, breakup, collision, and evaporation models.

We started applying the discrete-particle method via Lagrangian tracking with OpenFOAM for which particles are assumed to take up a negligible volume in the continuous phase and particle-particle interactions are ignored, but a two-way coupling of course was used in order to take into account the interaction with forced ventilation. We assumed a Rosin-Rammler (Weibull) distribution of droplets diameters at injection in the domain for the prediction of the droplet sizes that result from atomization, whose diameters reduces just due to evaporation (governed by Ranz-Marshall correlation).

Many simulations were done to evaluate grid dependence and the number of tracking-particles focusing on the fluid motion in an almost isothermal condition, and results has shown a strong dependence of solution with the grid (the particle trajectories are calculated from mean continuous phase velocities), and, for instance, more the grid is fine, more particles tend to be close to each other and the spray cone reduces, as observed also by others [6]. A fewer mass of water is then dispersed in the whole domain, reducing significantly the ability to model this peculiarity of water-mist sprays, characterized by a great dispersion (used in fact as competitor of total flooding agents). Small droplets sufficiently far from the spray region will be neglected by the Lagrangian procedure, but they might

significantly affect the flow and temperatures. The method for coupled analysis of dispersed multiphase flow is the particle-source-in-cell (PSI) algorithm [7] (Fig. 1).

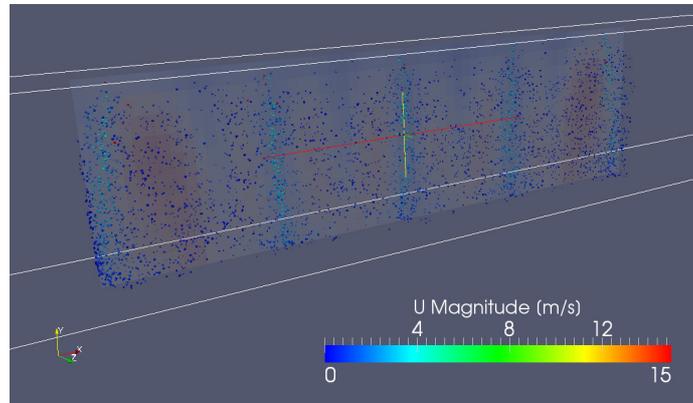


Figure 1. CFD tool: snapshot from an example case with five water mist nozzle operating in a tunnel section (nozzle water pressure: 100 bar, orifice droplet velocity magnitude: 100 m/s, nozzle water flow: 18 l/min). Only some of the 200000 tracking parcels in system are shown.

Among grid dependence, another criticality is the necessity of an averaging of individual particle properties to derive the intermediate spray structure, so a great number of particles need to be tracked in order to obtain a statistically meaningful average, but this number is strongly limited due to the available computational resources.

Combustion is modeled as a buoyancy driven turbulent diffusion combustion with a one-equation (Smagorinsky) LES model (Fig. 2) and we are considering to adopt the same scheme proposed for the fireFoam application recently distributed as open source project.

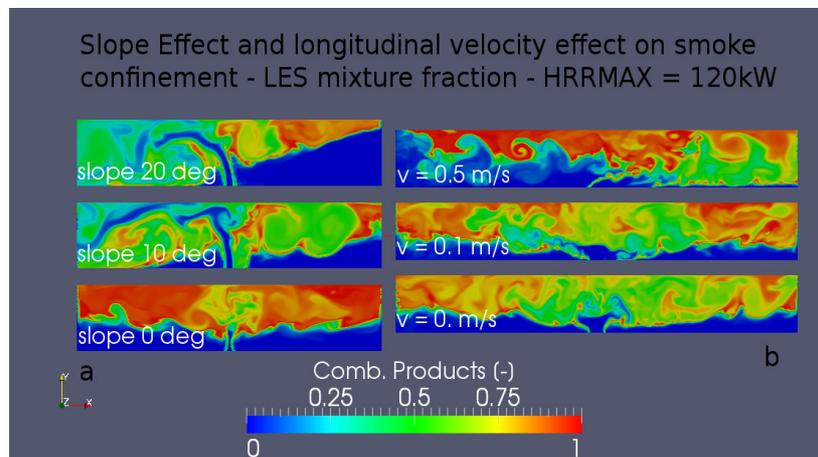


Figure 2. CFD tool: example of LES turbulent diffusion flame combustion results.

Multiscale approach

Through a multiscale approach, we believe it is possible to treat a complex system as a tunnel fire scenario is. In fact, coupling a lumped parameter network model and a CFD model it is possible to dramatically reduce the computational cost and determining the boundary conditions of the CFD detailed analysis with a network model that simulates the complex system that has incorporated the “detailed analysis” of CFD [8] (Fig. 3). Tunnels, stations, ramps and stairs for access, ventilation ducts, can be considered as the branches of a circuit connected by nodes. In this way the topology can be immediately represented in matrix form. We assume each branch representing a section of the system having uniform linear distribution of pressure and volume forces (whether due by fans, piston effects of moving vehicles, friction effects and buoyancy forces). Our model solves the conservation of mass with a balance

of flow rates on each nodes and the conservation of momentum for each branch based on the Bernoulli equation. It assumes perfect mixing at nodes and with the assumption of unidirectional heat transfer to the wall it determines temperatures at nodes once the flow field is solved.

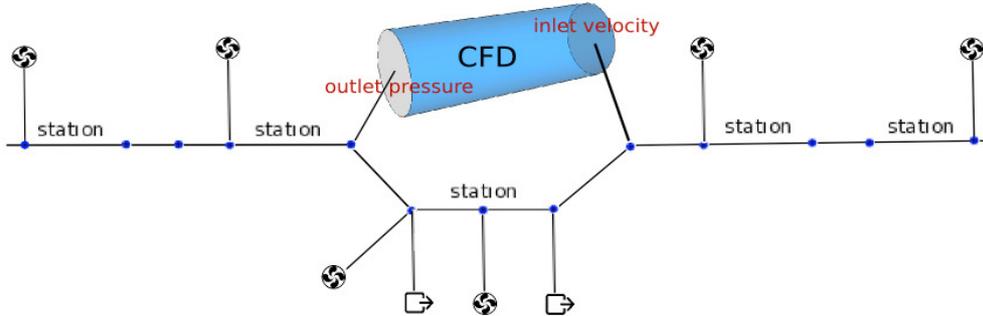


Figure 3. Multiscale approach scheme for subway scenario analysis.

EXPERIMENTAL ACTIVITIES

Spray Characterization

We are characterizing the sprays by laser measurements of droplets diameters and velocity (PDA - Phase Doppler Anemometry, Fig. 4). Such experiments allow us to obtain values to determine the average diameters and average speeds in different positions of the injection cone. In fact, using this data, instead of the ones supplied by manufacturers we would better reconstruct the diameters and velocities distribution of droplets close to the nozzle, which is a fundamental input for simulations.

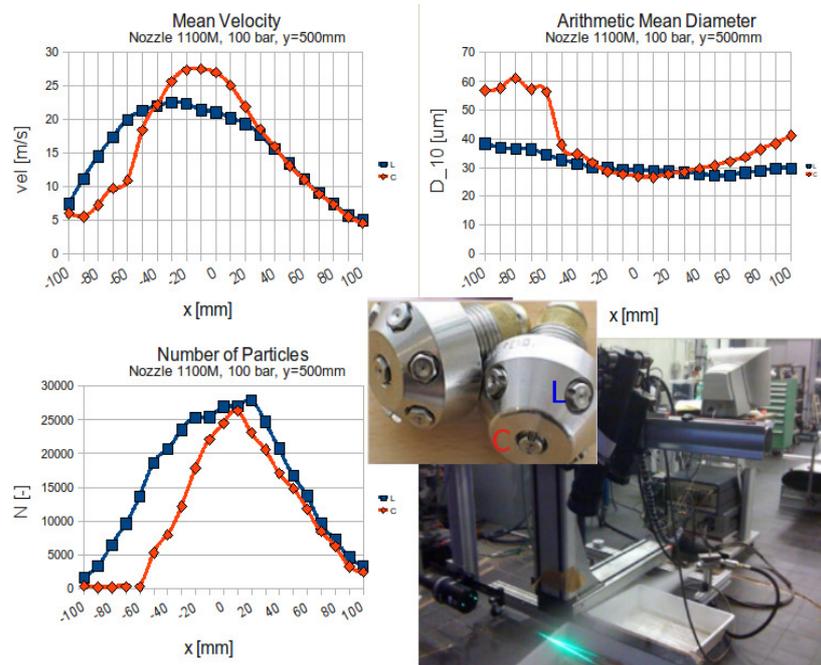


Figure 4. Example of measurement results for droplets velocity and diameter distribution for a central orifice (C) and a lateral one (L) for a common water mist nozzle.

Spray distribution

We did experiments oriented to compare the results obtained from simulations in terms of spray dynamics to the real distribution of droplets at soil. These findings were important to evaluate also the minimum computational costs

expected for the adoption of the Lagrangian spray model, without considering break-up and coalescence, and without any atomization model, to achieve sufficient accuracy (Fig. 5).

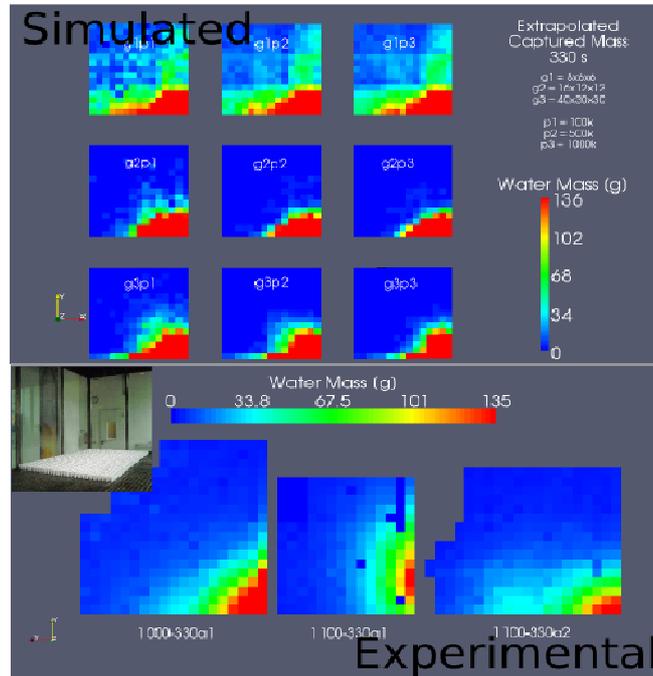


Figure 5. Example of spray precipitation at floor: comparison between experiment results and numerical simulations ones to assess lagrangian procedure parameters in terms of number of parcels and grid sensitivity analysis.

Jet Fire

The experimental activity that is planned now is regarding the validation of the evaporation model (we don't address now suppression effects). We try to do this measuring air/ smoke temperatures near a non premixed jet fire flame in a compartment in presence of a water mist discharge, with the intent to not affect significantly the combustion.

Conclusions

During this first stage of our project we approached water mist spray modeling with eulerian-lagrangian procedure in order to be able to perform effectiveness assessment of fire protection strategies in tunnel fire. This method was applied with the Lagrangian particle tracking present in the OpenFOAM code. The results we obtained, preeminently regarding spray dynamics, evidenced the encountered limitations of that spray approach in particularly dense spray, because of the comparison with experimental data for precipitation drops to the floor in cold environments. So, we are working now on the development of an Eulerian-Eulerian model (initially for monodisperse type and then polydisperse type of spray) for the simulation of transport and evaporation [9], [10], which must be then integrated with the turbulent combustion one, which instead has already given encouraging results, at least in qualitative terms. We have planned to validate the CFD model mainly on experiments provided by the temperatures measured during the jet fire experiments. Then, when the CFD tool will have been validated for simple cases, it will be coupled with the 1D network tool to simulate fire tunnel scenarios. We are also interested in considering also the integration with another 3D tool (fast running type numerical model) to have a more uniform and effective approach to the complex analysis.

REFERENCES

- [1] N/A, *SES User's Manual*, v4.1, U.S. Department of Transportation, 2002.
- [2] Parozzi, F., *ECART User's Manual*, CESI Ricerca, 2005.
- [3] McGrattan, K., *Fire Dynamics Simulator (Version 4) - Technical Reference Guide*, NIST Special Publication 1018, 2006.

- [4] N/A, *OpenFOAM 1.6 User Guide*, OpenCFD Ltd, 2000 - 2009.
- [5] Nmira, F., Consalvi, J.L., Kaiss, A., Fernandez-Pello, A.C., Porterie, B., "A numerical study of water mist mitigation of tunnel fires", *Fire safety journal*, vol. 44, n. 2, pp. 198 ÷ 211, 2009.
- [6] Sirignano, W. A., *Fluid dynamics and transport of droplets and sprays*, Cambridge University Press, 2005.
- [7] Crowne, C.T., Sharma, M.P., Stock, D.E., "The particle-source-in-cell method for gas and droplet flow", *ASME Journal on fluids engineering*, vol. 3, n. 99, pp. 325 ÷ 332, 1977.
- [8] Colella, F., Rein, G., Borchiellini, R., Carvel, R., Torero, J.L., Verda, V., "Calculation and design of tunnel ventilation systems using a two-scale modeling approach", *Building and Environment*, Building and Environment 44: 2357 ÷ 2367, June 2009.
- [9] Massot, M., "Eulerian multi-fluid models for polydisperse evaporating sprays", in Marchisio, D.L., Fox, R.O., *Multi-Phase Reacting Flows: Modeling and Simulation*, CISM Courses and Lectures, vol. 492, Springer, Wien, chapter III, pp. 79 ÷ 123, 2007.
- [10] de Chaisemartin, S., Laurent, F., Massot, M., Reveillon, J., "Evaluation of Eulerian multi-fluid versus Lagrangian methods for the ejection of polydisperse evaporating sprays by vortices", *Journal of Computational Physics*, submitted for publication, available on HAL, <http://hal.archives-ouvertes.fr/hal-00169721/>, 2009.