Modelling of spray injection from water mist fire suppression systems

S. Tonini[§,*], A. Theodorakakos[†], M. Gavaises[‡] and G.E. Cossali[§]
[§]Department of Industrial Engineering, University of Bergamo, Italy
[†]Fluid Research Co., Athens, Greece
[‡]School of Engineering and Mathematical Sciences, The City University London, UK

Abstract
The paper aims to describe the numerical methodology implemented to predict the flow inside a water mist injector used for fire control spray applications and the formation of the liquid lamella emerging from its discharge hole, using single and multi-phase Eulerian flow models. These provide the necessary information required by the liquid film atomisation model, which can be used to determine the spray drop characteristics from this type of injection systems, as function of the geometric and operating conditions selected.

Introduction

The water mist is defined as a system where 99% of the drops have a diameter smaller than 1 mm [1]. This type of injector is capable to assure control, suppression or extinguishment of the flame by using a smaller quantity of water compared to traditional sprinkler systems. Due to the limited number of studies published in the open literature, the characteristics of the flow inside the water mist systems and their effect on spray formation and subsequent mixing are relatively unknown [2]. The large majority of the computational codes simulating sprays for fire protection applications imposes, as boundary conditions, data obtained by previous studies, available in open literature, or derived from simplified models that usually neglect many complex phenomena taking place inside the injector [3]. On the other hand, recent studies suggest that complex two-phase flow processes may take place inside the discharge hole of such injectors, evidencing the crucial role of the in-nozzle flow in characterising the external spray [4, 5]. For the category of injectors to which water mist system belongs, a liquid lamella is formed inside the outer part of the nozzle, due to the swirl motion produced by the internal flow geometry, inducing the opening of the liquid cone and the formation of a recirculation zone that affects the liquid primary break-up generating the drops of the external spray [4, 6, 7]. This supports the idea of a strong relationship between the possible phase transition inside the nozzle, due to local pressure fall, and the external spray formation.

Mathematical Modelling

The flow development inside the injector is simulated employing a so called ‘two-step’ numerical methodology in the GFS CFD code [8]. In the ‘first step’ single-phase calculations are performed, based on the Eulerian methodology, assuming that the injection hole is filled with liquid, while in the ‘second step’ it is assumed that the liquid-gas interface is located in the area where the net balance of the incoming and exiting from the hole flow becomes zero [8]. The nozzle flow model accounts for the tracking of liquid-gas interface surfaces using the ‘volume of fluid’ (VOF) methodology [9], imposing as boundary conditions the results from the previous single-phase 3-D simulations. The internal nozzle flow is investigated in a water mist injector with discharge hole exit diameter equal to 0.7 mm. Three typical values of injection pressure, equal to 60, 70 and 80 bar, have been imposed as uniform boundary conditions at the injector inlet, while atmospheric back pressure is selected as initial conditions in the external region.

Results and Discussion

The 3-D single-phase CFD model has been used to predict the flow distribution and the swirl formation process inside the water mist nozzle. The liquid from the cylindrical conduct enters the inclined flow passages that give an angular momentum to the fluid before converging in the conical region. Figure 1 shows the internal recirculation of the flow field in the conical slot of the nozzle, just above the discharge hole exit. A gradual increase of the swirl velocity is taking place in the conical slot, forcing the main bulk of liquid entering to the hole to rotate. This will induce the opening of the conical liquid film once the fluid exits the discharge hole and it develops in the external region, and it suggests that two-phase phenomena may occur inside the nozzle. The results from the 3-D single-phase model evidence that the characteristics of the internal flow on a plane perpendicular to the nozzle axis, located at 2.6 mm above the discharge hole exit in the conical slot of the nozzle,

* Corresponding author: simona.tonini@unibg.it

present a quasi axisymmetric behaviour, then 2-D axisymmetric simulations can be performed. The necessary boundary conditions are calculated as radial profiles of mean flow field characteristics on that plane and they are used for the successive two-phase flow simulations, which predict the formation of the liquid lamella emerging from the discharge nozzle hole as a result of the action of the centrifugal forces associated with this swirling motion. Figure 1(b) shows the liquid volume concentration, on the left, and the velocity flow field distribution, on the right, in the discharge hole and the external region at a distance of about one hole diameter from the nozzle exit, for the three injection pressures selected. The results show that the film thickness is about one third of the discharge hole radius at the exit almost independent of the injection pressure. The characteristics of the liquid film emerging from the nozzle will be used as input conditions for the atomisation model, which predicts the spray drop generation and it represents the main target of future investigation.

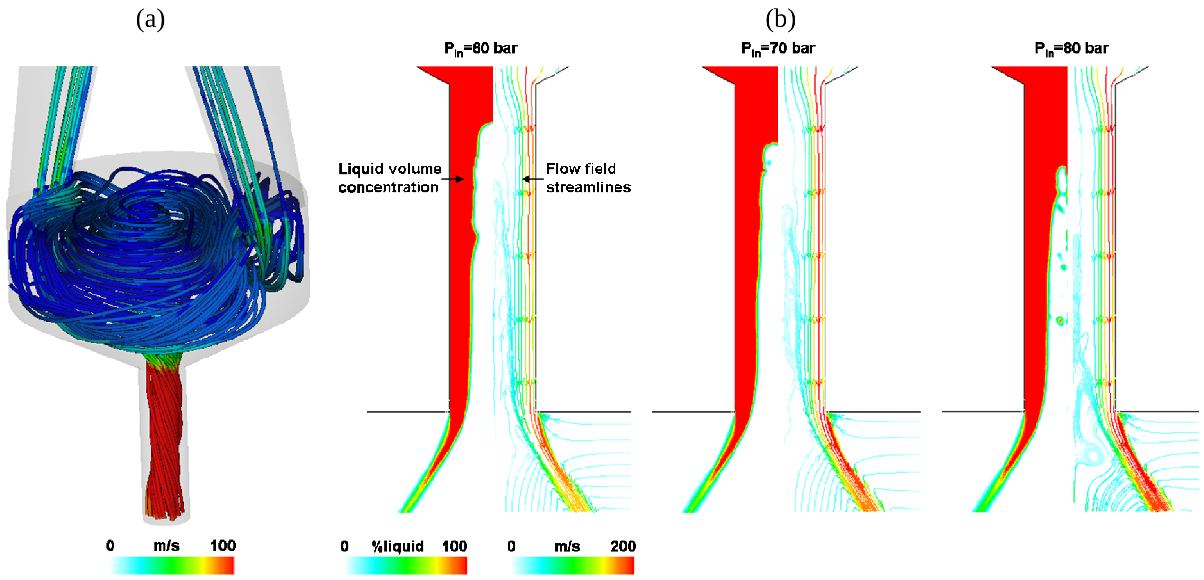

Figure 1. (a) Sketch of the flow field internal recirculation in the injection hole with streamlines coloured according to the liquid total velocity, P_{in}=80 bar; (b) effect of injection pressure on the liquid volume concentration (left) and velocity flow field (right) distributions on the plane along the injector axis.

Acknowledgement

The authors would like to deeply acknowledge Mr Demetris Papoulias from City University London for his assistance in the implementation of the internal nozzle flow numerical methodology into the CFD code used in the present investigation, and Bettati Antincendio s.r.l. for the details of the injector nozzle geometry. This work has been partially financed by the Italian research program PRIN2007.

References

[1] NFPA, *National Fire Protection Association* Quincy, MA, USA (2000).
[2] Santangelo, P.E., Tartarini, P., Pulvirenti, B., and Valdiserri, P., *Proc. 11th International Conference on Liquid Atomization and Spray Systems ICLASS 2009*, Vail, CO, USA, 26-30 July 2009.
[3] Santangelo, P.E., Ren, N., Tartarini, P., and Marshall, A.W., *25th UIT National Heat Transfer Conference*, Trieste, 18-20 June 2007.
[4] Chinn, J.J., and Yule, A.J., *Proceedings of ICLASS-'97*: 868-875 (1997).
[5] Das, S., and VanBrocklin, P.G., *13th International Multidimensional Engine Modelling User's Group Meeting*, Detroit, MI, 1-6 March 2003.
[6] Cousin, J., Ren, W.M., and Nally, S., *SAE Paper 980499* (1998).
[7] Dumouchel, C., Bloor, M.I.G., Dombrowski, N., Ingham, D.B., and Ledoux, M., *Chemical Engineering Science* 48(1): 81-87 (1993).
[8] Gavaises, M., and Arcoumanis, C., *International Journal of Engine Research* 2(2): 95-117 (2001).
[9] Ubbink, O., *Numerical prediction of two fluid systems with sharp interfaces, Imperial College, University of London*, 1997.