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

Fire suppression by water mist systems is a relatively recent subject of engineering investigation, although the physical basis that support this technology have been studied since long time [1]. The water mist is defined as a system where 99% of the drops have a diameter smaller than 1 mm [2]. 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. The water mist system performances depend mainly on the success in generating and distributing micrometric droplets in the fire. The growing interest of the scientific community on these fire protection systems has driven the development of detailed tools and procedures capable to evaluate their characteristics and performances [3, 4].

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; moreover experimental evidence of the spray structure very near to the nozzle hole exit is limited to visualization rather than quantitative information [4]. Consequently the development of numerical modelling able to predict the internal flow and its link with the spray generation has been largely promoted within the scientific community. 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 [5]. 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 [6, 7].

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 [6, 8, 9]. 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.

Based on previous experience on the modelling of sprays for automotive applications, the easiest approach to predict the formation and development of water mist sprays is to employ the conventional Eulerian-Lagrangian approximation, according to which the spray is represented by a discrete number of computational parcels [10]. According to this methodology, the droplet initial conditions (size and velocity) represent one of the most important factors affecting the following spray predictions [11]. The injection rate can be estimated by injection system flow models, while arbitrary assumptions are typically employed for the initial size of the injected parcels. Usually blobs representing the liquid film formed at the exit of the water mist discharge hole are injected and then a liquid sheet atomization model is employed in order to predict the mean size of the formed droplets [12]. The initial blob size is assumed to be equal to the liquid film thickness at the nozzle exit, while the injection velocity is estimated from mass conservation considerations. These information could be arbitrarily imposed, according to previous studies or physical assumptions, or could be estimated by the investigation of the internal nozzle flow and its link with the emerging liquid from the injector nozzle [13].

One method proposed in the open literature for the estimation of the nozzle flow exit characteristics consists in the application of a multi-dimensional single-phase Eulerian model that predicts the development of the swirl velocity inside the injection nozzle as a function of the internal nozzle geometry and a two-phase model, based

* Corresponding author: simona.tonini@unibg.it

on the 'volume of fluid' (VOF) methodology, which estimates the gas-liquid surface interface inside the injector, thus providing better estimates of the two-phase flow characteristics emerging from its discharge hole [14].

Successively the liquid film thickness and the spatially averaged mean axial and swirl velocity component, estimated at the nozzle hole exit during the injection period by the previous Eulerian models, can be used as inputs to the liquid film motion model [11], based on the Lagrangian approach, in the vicinity of the injection hole until its disintegration into droplets. Liquid film primary break-up models can be applied to estimate the atomization time and length scales and to predict the size distribution of the resulting droplets [15]. The liquid jet at the injector exit, subject to internal oscillations and surface instabilities, deforms forming ligaments that, under the action of capillary forces, break into smaller droplets of different sizes (primary atomisation). Primary atomisation has been extensively studied both experimentally and numerically [16-18], proposing, as possible mechanisms to explain it, the upstream turbulence [19] and the surface instabilities, treated by linear and viscous stability theory [20, 21]. Due to the complexity of these phenomena, it is nowadays still difficult to predict atomisation and its nature is the subject of debates among the researchers in the field [6]. These models can be incorporated into spray CFD models which predict the droplet size and velocity at distances further downstream of the injection nozzle. Refer to [11] for a detailed description of this numerical methodology to sprays injected from pressure swirl atomisers for GDI engine applications, which present several analogies with water mist spray systems.

The objective of this study is to describe the numerical methodology employed to predict the flow development inside a water mist injector and the characteristics of the liquid lamella emerging from the discharge hole, which provide the necessary injection conditions for successive spray models. In the next sections of the paper the models used for the purposes of the present investigation are briefly presented, followed by the discussion of the main results obtained and the summary of the most important conclusions.

Mathematical modelling

In this section, the mathematical models used to estimate the liquid injection process from a water mist injector are briefly described. The flow development inside the injector is simulated employing a so called 'two-step' numerical methodology in the GFS CFD code [11]. 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 [11]. The nozzle flow model is based on the two-phase extension of a previous developed CFD code, which accounts for the tracking of liquid-gas interface surfaces using the 'volume of fluid' (VOF) methodology [22]. The models numerically solve the full Navier-Stokes equations describing the motion of the fluids. The time averaged forms of the continuity, momentum and conservation equations for the scalar variables are numerically solved using collocated Cartesian velocity components on a Cartesian non-uniform, curvilinear, non-orthogonal numerical grid. The discretisation method is based on the finite volume approach and the pressure correction method is according to the PISO algorithm [23]; the spatial discretisation scheme used here is the second order hybrid, while the time dicretisation is based on a fully implicit Crank-Nicolson scheme. More details about the relevant equations and their numerical form can be found in [22, 24].

The internal nozzle flow is investigated in a water mist injector with discharge hole exit diameter equal to 0.7 mm. The numerical grid used for the 'first step' 3-D single phase calculations is shown in Figure 1(a), consisted of approximately 650,000 pyramidal and hexahedral cells, with the flow exit boundary extended well downstream from the hole exit in order to allow more accurate boundary conditions. The figure enlightens the injector inlet plane and axis, the inclined flow passages that induce the swirling motion to the liquid fluid, the discharge hole and the external region. A plane perpendicular to the injector axis, at about 2.6 mm above the hole exit, is also evidenced since it is used to define the boundary inlet conditions for the successive 2-D axisymmetric simulations, since it delimits the region below which the flow behaviour can be assumed as quasi-axisymmetric. Injection pressure is assumed constant for the whole injection period, then the simulation is stopped when the steady-state solution is calculated. Three injection pressures, 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. From the solution of the single-phase 3-D simulation the axial, radial and swirl velocity components of the fluid in the discharge hole can be estimated with good accuracy.

These information provide the initial conditions to the 'second step', which simulates the liquid-gas interface using the VOF methodology. Since this approach is computationally more expensive than the previous one, a 2-D axisymmetric computational domain, selected on a plane along the nozzle axis, has been used instead of a fully 3-D one, in order to reduce the computational time, which otherwise would be too long for practical calculations to be performed. The region where the flow can be assumed axisymmetric needs to be identified in the 3-D geometry and the corresponding plane perpendicular to the injector axis is selected to define the boundary inlet conditions for the 2-D axisymmetric simulations. The numerical grid used for the 2-D solution comprises approximately 10,000 computational triangular cells with a grid spacing inside the discharge hole of

about 17 µm, as shown in Figure 1(b). Mesh independency tests have been performed implementing one and two levels of automatic grid local refinement, according to [24], while approximately 3,000 time steps of 0.1 µs were required to achieve a steady-state solution.

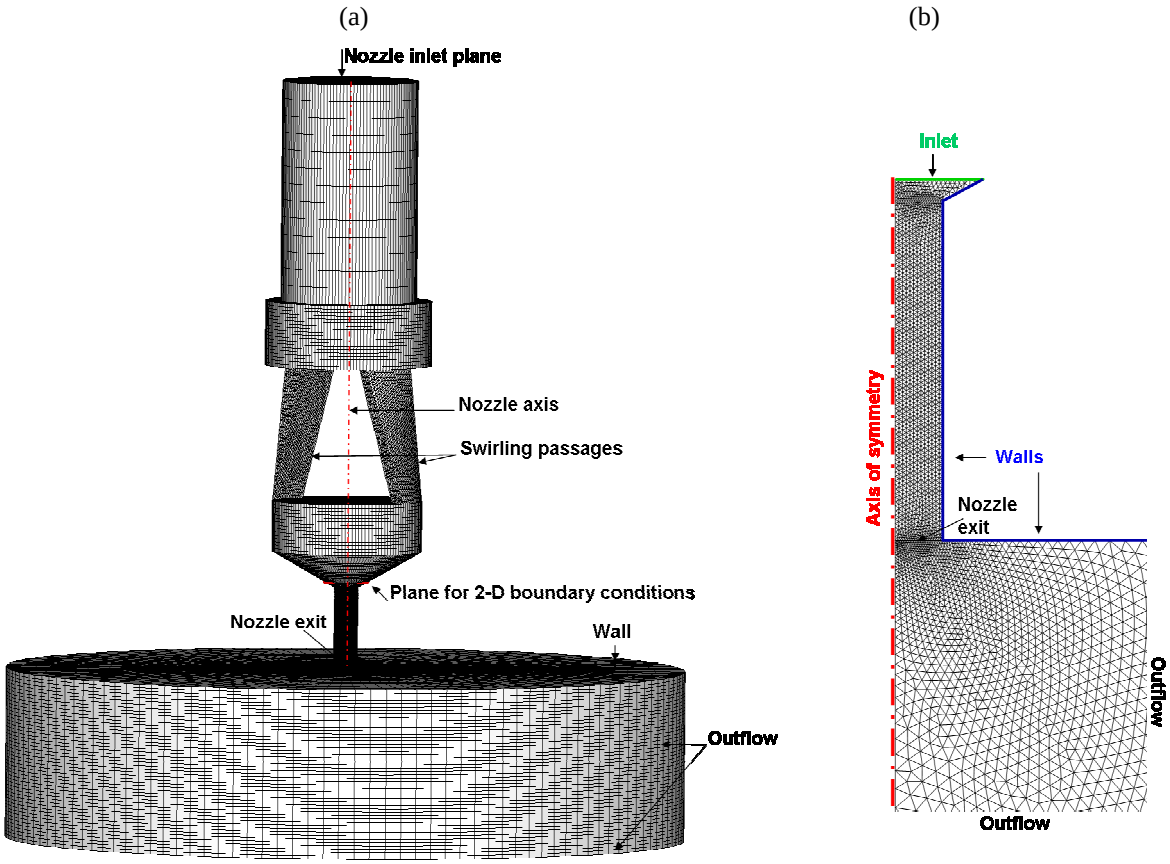

Figure 1. (a) Numerical grid for the 3-D single-phase Eulerian simulations. The plane selected for the definition of the boundary conditions for the 2-D calculations is also shown. (b) Numerical grid for the 2-D axisymmetric two-phase flow simulations.

Results and Discussion

In this section the computational results obtained with the single and two-phase flow models previously described, referring to the internal nozzle flow and the near nozzle liquid film structure until its disintegration into droplets, are presented and discussed.

3-D single-phase internal nozzle flow simulations

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, where constant and uniform injection boundary pressure is imposed according to the test case selected, enters the inclined flow passages that give an angular momentum to the fluid before converging in the conical region with a discharge hole of 0.7 mm diameter and 2.4 mm length. Figure 2(a) shows the internal recirculation of the flow field in the conical slot of the nozzle, just above the discharge hole exit, for the case with injection pressure equal to 80 bar. 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 process occurs together with the increase of the total velocity as the liquid enters the smaller discharge hole. The characteristics of the fluid on a plane along the injector axis are evidenced in Figure 2(b), which shows the distributions of the pressure field and of the velocity streamlines in the nozzle conical region. The results show the pressure gradually dropping within the conical slot and becoming equal to the back pressure in the area of the discharge hole occupied by the liquid film, while a low pressure region is developing at the centre of the injection hole, where the flow field velocity reaches its maximum value. The single-phase calculations provide evidence of the swirling motion inside the nozzle, which will induce the opening of the conical liquid film once the fluid exits the discharge hole and it develops in the external region, and moreover it suggests that two-phase phenomena may occur inside the nozzle, due to the depression generated inside the hole.

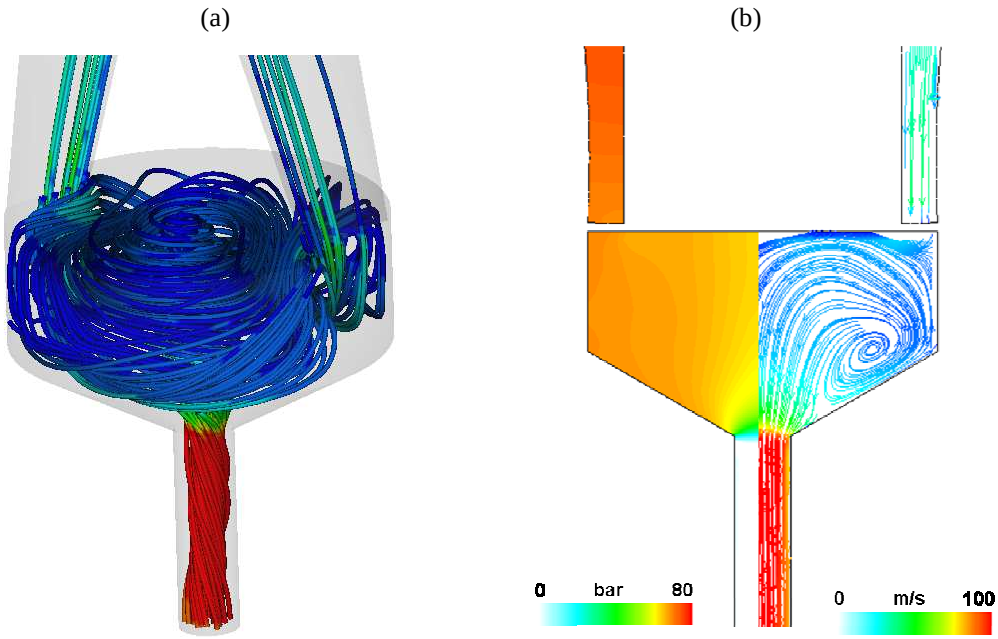

Figure 2. (a) Sketch of the flow field internal recirculation in the injection hole with the streamlines coloured according to the liquid total velocity; (b) pressure field (left) and flow velocity streamlines (right) distributions on a plane inside the injection hole along the injector axis; P_{in}=80 bar.

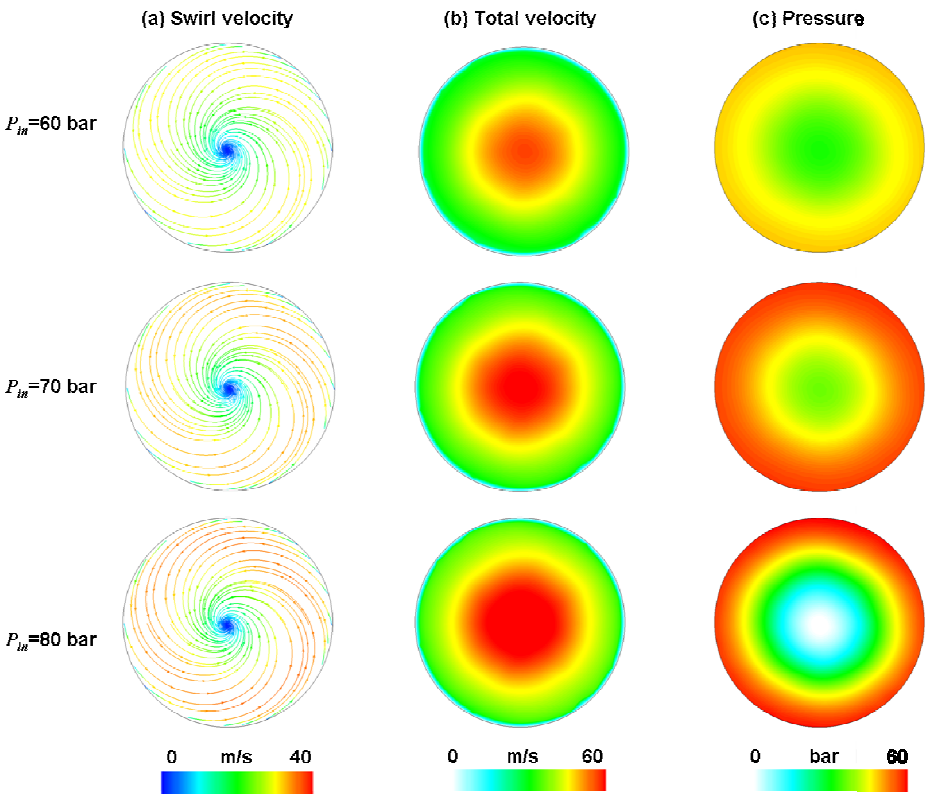

Figure 3. (a) Streamlines coloured according to swirl velocity, (b) total velocity and (c) pressure flow field distributions on the plane selected for the 2-D boundary conditions, located at 2.6 mm from the discharge hole exit, perpendicular to the nozzle axis.

On the other hand the single-phase model alone cannot predict the formation of the liquid lamella at the nozzle exit, as suggested by experimental investigation on this injector [4], then a more accurate two-phase flow model must be employed. Due to the remarkably more expensive computational time requested by the VOF model implemented for such calculations, 2-D axisymmetric simulations should be preferred. This requires the definition of the boundary conditions, imposed as input to the model, in the region where the flow field can be assumed as axysimmetric. 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, present an axisymmetric behaviour, as shown in Figure 3, which illustrates the swirl velocity, total velocity and pressure distributions for the three injection pressures selected for the present investigation. The images enlighten the increase of the flow field velocity (swirl and total) with the injection pressure, which is combined with the corresponding enhancement of the depression in the central recirculation zone of the conical slot. The total velocity increases along the radial direction approaching the nozzle axis.

Since the 2-D axisymmetric simulation requires a radial profile of the velocity components on the selected plane, mean values of the flow field characteristics have been calculated. Figure 4 show the axial, radial and swirl mean velocity profiles along the radial distance from the nozzle axis, on the plane chosen for the 2-D boundary conditions. The results enlighten the decrease of the axial velocity along the radial distance, together with its increase with injection pressure, as expected. The radial velocity presents a peak at about half distance from the nozzle axis, while the swirl velocity at about 80% distance, almost independently on the injection pressure, and they both become null in the centre of the plane and at the nozzle wall. All the velocity components increase with injection pressure.

These velocity profiles have been used for the successive 2-D axisymmetric two-phase flow simulations.

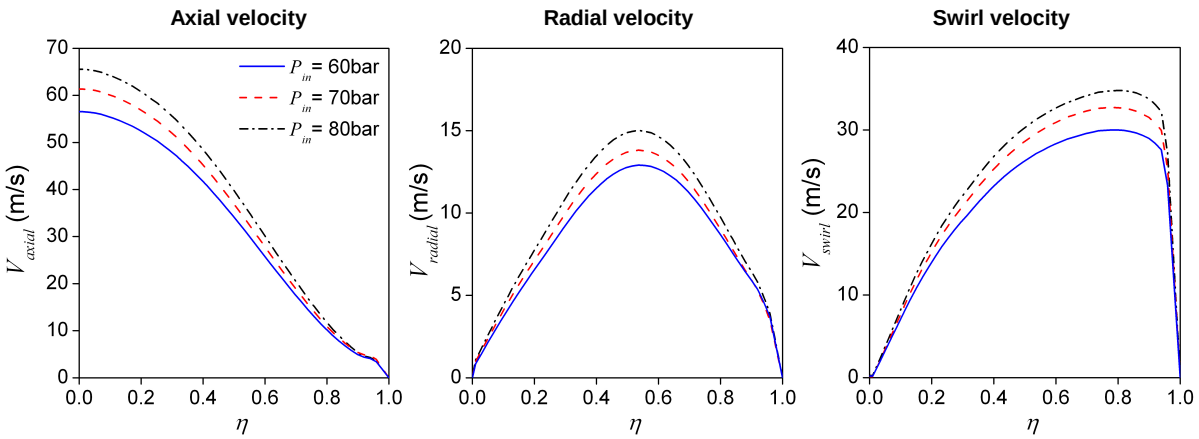

Figure 4. Effect of injection pressure on axial, radial and swirl mean velocity profiles on the plane selected for the 2-D boundary conditions, as function of the radial distance from the nozzle axis.

2D two-phase nozzle flow simulations

The two-phase flow model, based the VOF methodology and implemented in the GFS CFD code, has been used to perform the 2-D axisymmetric simulations predicting the formation of the liquid lamella emerging from the discharge nozzle hole. The computational mesh used for these calculations is illustrated in Figure 1(b), while the automatic local refinement is activated, refining the grid at the liquid/gas interface. The radial velocity profiles shown in Figure 4 are imposed as boundary conditions at the inlet, while outflow conditions are assumed at the outlet boundaries and atmospheric pressure as initial flow conditions. The model predicts a gradual increase of the swirl taking place in the conical slot, forcing the main bulk of liquid in the nozzle to rotate and thus to gradually move towards the walls of the discharge hole, as a result of the action of the centrifugal forces associated with this swirling motion. Parallel to the swirl process, a low pressure recirculation zone is gradually developing at the centre of the injection hole. This results in the formation of a liquid lamella attached to the walls of the discharge hole and emerging from the nozzle, while air is entrapped in the central low pressure region. Exiting from the nozzle the lamella opens due to the centrifugal forces. The developed film gradually stabilizes and converges to quasi steady-state characteristics.

Figure 5 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, which is about one third of the discharge hole radius at the exit, is almost independent of the injection pressure. This can be explained by the fact that the swirl-to-axial velocity ratio, which controls the liquid film thickness, might not depend on the

injection pressure, but mainly on the design of the inclined flow passages, which impose the swirling motion to the liquid entering into the nozzle. The velocity flow field distributions enlighten the recirculation of the air entrapped in the central region, while the liquid film accelerated as it exits from the nozzle, proportionally to the injection pressure.

Figure 6 summarizes these findings showing the predicted film thickness as well as the spatially averaged mean axial and the swirl velocity components of the liquid emerging from the injection hole. These information will be used as input conditions for the liquid sheet atomisation model, implemented in the GFS spray code, which will provide the spray cone angle and drop size distribution as resulting from the liquid primary break-up, and which represents the target of the future investigation.

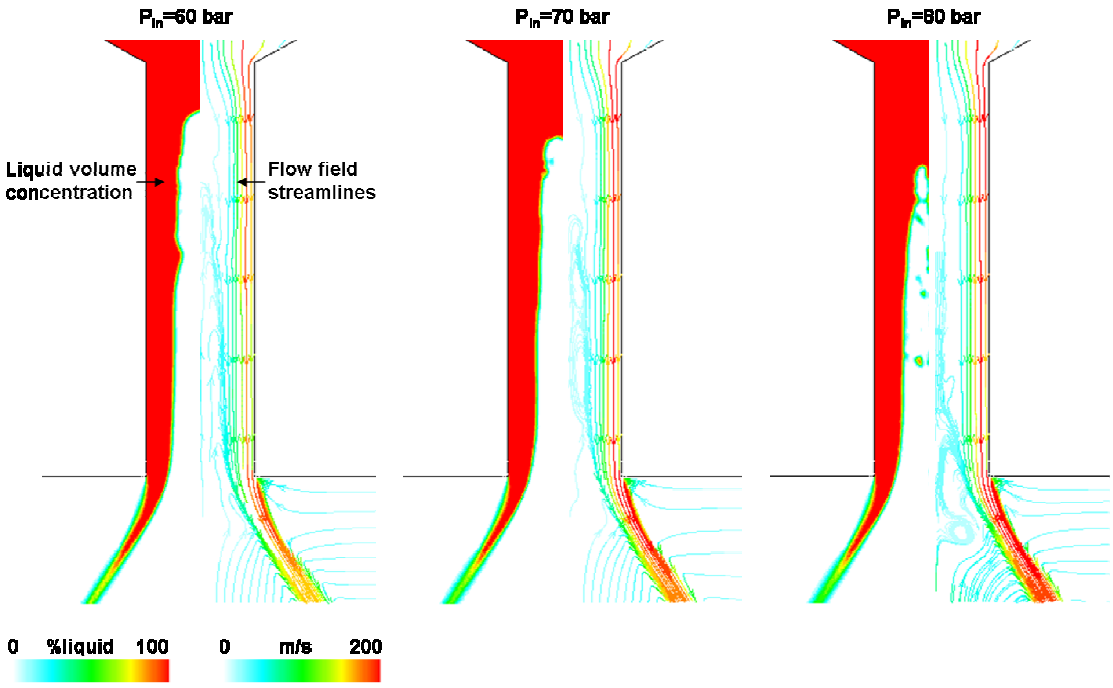

Figure 5. Effect of injection pressure on the liquid volume concentration (left) and velocity flow field (right) distributions inside the injection hole on a plane along the injector axis.

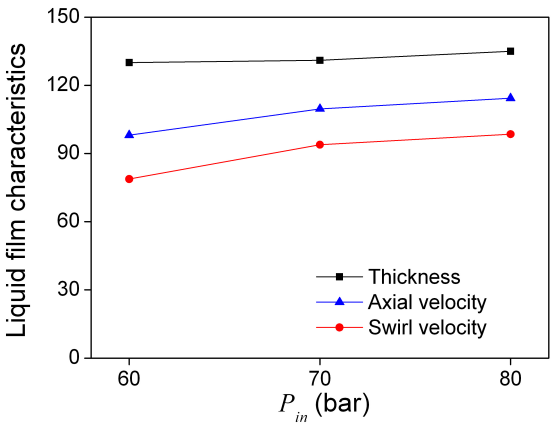

Figure 6. Effect of injection pressure on the liquid film characteristics at the nozzle exit: thickness, mean axial and swirl velocities.

Conclusions

The numerical methodology used to predict the liquid film emerging form the nozzle of a water mist injector is illustrated, describing the 'two-step' procedure followed to calculate the swirl motion generated by the internal nozzle geometry, according to the 3-D single-phase steady-state CFD model, and the characteristics of the liquid lamella emerging from the discharge hole, following the 2-D axisymmetric VOF model. The effect of injection pressure on the internal nozzle flow field and on the development of the external liquid film is investigated

selecting three typical values for the injection pressure, assumed constant and uniform along the injector inlet plane.

The results from the present investigation suggest that complex two-phase flow phenomena occur in the central zone of the conical slot of the nozzle, where the air is entrapped from the external region, due to the swirling motion of the liquid generated by the inclined flow passages located just above. The liquid is forced to be attached to the discharge hole walls, forming a liquid lamella that, due to the centrifugal forces, opens as it exits the nozzle. The effect of injection pressure on this process is evident on the internal flow field and liquid lamella velocity components, while the film thickness results to be almost independent on injection pressure, since its value seems to be mainly determined by the geometric details of the nozzle, instead of its operating conditions.

The characteristics of the liquid film emerging from the nozzle (thickness, mean axial and swirl velocities) will be used as input conditions for the liquid sheet atomisation model, which predicts the spray drop generation and it represents the main target of future investigation.

Acknowledgments

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] Grant, G., Brenton, J., and Drysdale, D., *Progress in Energy and Combustion Science* 26(2): 79-130 (2000).
[2] NFPA, *National Fire Protection Association* Quincy, MA, USA (2000).
[3] Adiga, K.C., and Hatcher, R.F., *Fire Safety Journal* 42(2): 150-160 (2007).
[4] 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.
[5] Santangelo, P.E., Ren, N., Tartarini, P., and Marshall, A.W., *25th UIT National Heat Transfer Conference*, Trieste, 18-20 June 2007.
[6] Chinn, J.J., and Yule, A.J., *Proceedings of ICLASS-'97*: 868-875 (1997).
[7] Das, S., and VanBrocklin, P.G., *13th International Multidimensional Engine Modelling User's Group Meeting*, Detroit, MI, 1-6 March 2003.
[8] Cousin, J., Ren, W.M., and Nally, S., *SAE Paper 980499* (1998).
[9] Dumouchel, C., Bloor, M.I.G., Dombrowski, N., Ingham, D.B., and Ledoux, M., *Chemical Engineering Science* 48(1): 81-87 (1993).
[10]Dukowicz, J.K., *Journal of Computational Physics* 35(2): 229-253 (1980).
[11]Gavaises, M., and Arcoumanis, C., *International Journal of Engine Research* 2(2): 95-117 (2001).
[12]Raju, M.S., *International Conference on Computational and Experimental Engineering and Sciences*, Chennai, India, 1-6 December 2005.
[13]Arcoumanis, C., Gavaises, M., Argueyrolles, B., and Galzin, F., *International Congress and Exposition*, Detroit, Michigan, 1-4 March 1999.
[14]Arcoumanis, C., and Gavaises, M., *SAE Paper 2000-01-1044* (2000).
[15]Santangelo, P.E., Ren, N., Tartarini, P., and Marshall, A.W., *22nd Conference on Liquid Automization on Spray Systems*, Como, Italy, 8-10 September 2008.
[16]Crapper, G.D., Dombrowski, N., and Pyott, G.A.D., *Proceedings of the Royal Society of London. Series A, Mathematical and Physical Sciences*: 209-224 (1975).
[17]Marmottant, P., and Villermaux, E., *Journal of fluid mechanics* 498: 73-111 (2004).
[18]Reitz, R.D., and Bracco, F.V., *Physics of Fluids* 25: 1730 (1982).
[19]Faeth, G.M., Hsiang, L.P., and Wu, P.K., *International Journal of Multiphase Flow* 21: 99-127 (1995).
[20]Chandrasekhar, S., *Hydrodynamic and hydromagnetic stability*, 1961, Clarendon Press, Oxford.
[21]Yecko, P., Zaleski, S., and Fullana, J.M., *Physics of Fluids* 14: 4115 (2002).
[22]Ubbink, O., *Numerical prediction of two fluid systems with sharp interfaces, Imperial College, University of London*, 1997.
[23]Issa, R.I., *Journal of Computational Physics* 62: 40-65 (1986).
[24]Theodorakakos, A., and Bergeles, G., *International Journal for Numerical Methods in Fluids* 45(4): 421-439 (2004).