ILASS 2008
Sep. 8-10, 2008, Como Lake, Italy

Paper ID ILASS08-12-3

IMPROVED MODELING OF LIQUID JETS IN CROSSSFLOW

Bellofiore A.[1], Di Martino P.[2], Lanzuolo G.[1], Ragucci R.[3]

1. Dipartimento di Ingegneria Chimica - Università Federico II, Naples – ITALY, gianluca.lanzuolo@unina.it,
2. Avio Group, Pomigliano d'Arco, Naples – ITALY, pasquale.dimartino@aviogroup.com
3. Istituto di Ricerche sulla Combustione - C.N.R., Naples – ITALY, ragucci@irc.cnr.it

ABSTRACT

This paper deals with the elaboration and implementation of a numerical model aimed to describe the behavior of a liquid jet, as it issues from a plain nozzle into a square channel and is suddenly exposed to a crossing airflow. The mathematical description of the process is formulated by assuming a two-dimensional curvilinear coordinate system. The guideline of this work has been the development of a simple model able to capture the essential physical phenomena involved in the crossflow atomization process, with no or minimal resort to empirically tuned parameters. Conceptual models available in literature were exploited for the different sub-mechanisms acting in the process and adopt and/or adapt them in order to describe as much as possible in a physically sound way the jet evolution. Mass reduction due to atomization was tracked with an adapted version of the Boundary Layer Stripping model. Jet cross section deformation due to the air drag force, was described in the framework of a modified version of the Taylor Analogy Breakup (TAB) model accounting both for the cylindrical geometry of the elementary volume of the jet and for the dependence of aerodynamic and capillary forces on jet cross-section deformation. Comparison of the modeled liquid jet trajectories were made both with leading edge and centerline measured trajectories, since, while the former are common reference data in the literature, the latter are making more sense from a physical point of view

INTRODUCTION

New technologies in the gas-turbine engine field are raising a growing interest in the injection of liquid fuel transversely to high-density airstream as an effective alternative to realize significant achievements of engine performances (mainly in terms of fuel economy and system robustness) along with a reasonable reduction of pollutants emission.

As matter of facts, engines currently under development rely on some form of staging of combustion process to achieve these improvements. The zonalization of the combustion chambers require the availability of injection and premixing devices that are capable of injecting a well mixed and, possibly, vaporized, air/fuel mixture in an as much as possible working condition-independent way. A general conceptual scheme of such devices is essentially a swirled duct (with a double swirl flow co-rotating or not) in which the fuel is injected in order to disperse and vaporize it as fast as possible. In this framework, the optimization of the fuel atomization and mixing processes represents one of the challenges to be faced.

On the other hand the demand for higher efficiency of power production and smaller engines pushes the increase of both pressure and temperatures of the inlet airflow along with an increase in the gas temperature at the turbine. This is cause of a decrease of the ignition delay of the air/fuel mixture that, in turn, can favor both the occurrence of flashback phenomena and the increase of some pollutants (e.g. NOx, CO and HC). In this case too a possible answer rely on the optimization of the mixture preparation process in the inlet duct of the engine.

Single and multipoint injection systems of the fuel present some remarkable advantages that make them a suitable answer to these requests. In fact, they represent a robust solution in reason of the simplicity of the atomizer (essentially a plain nozzle) and of the fuel feeding system. They can also be optimized either on the ground of a time-modulated strategy or by modulating the number and the position of the nozzles active in dependence of the working condition. There is a rather large corpus of knowledge available in literature on jet injection in cross-flow and it appear that the status of knowledge is at a satisfying level at least for the phenomenological description of the process. Unfortunately, the status of knowledge is mainly of empirical nature and in reason of the complex nature of the process a satisfying mathematical and numerical description is not yet available. Availability of efficient modeling tools represent a key factor in the efficient an economical development of new devices by allowing the determination of the most promising configurations suitable for further development and prototyping.

This paper contribute to the development of such modeling tools by presenting a further advancement of a jet bending and breakup model already presented by the same research group [1]. In the followings a brief description of the possible modeling strategies available in literature are presented with emphasis on the application to the gas-turbine engines applications and in correlation with the here proposed model. In particular, new implementations, concerning the deformation and heat transfer sub-models, are presented.

MODELING OF LIQUID JETS IN A GASEOUS CROSSFLOW

Possible approaches to the modeling of bending, deformation and breakup of a liquid jet subject to the interaction with a crossing airflow can be mainly classified as lagrangian, eulerian and lagrangian/eulerian.

The first class of models is basically based on the seminal work of Reitz [2]. They rely on the basic assumption that the liquid column can be modeled as a set of isolated drops issuing from the nozzle. The temporal evolution of each drop is assumed to be representative of the whole jet dynamics. This approach has been successfully used in case of liquid injected into quiescent or co-flowing gas, particularly when the high level of injection pressure actually prevents the existence of a liquid core even few diameters after the nozzle outlet [3, 4]. The main advantages of this approach are the ease of formulation of momentum balance equations, since the control volume coincides with the blob and only inertia and drag forces have to be accounted for.

The discrete blob method has been also used in the case of crossflow atomization [5, 6, 7]. The major drawback of this application can be attributed to its difficulties in accounting for cohesive viscous forces responsible of the higher resistance of a liquid column with respect to a set of isolated droplets to the airflow induced bending. This problem is more important in the premixing ducts of a gas turbine engine than in a diesel engine chamber. In fact, in this latter case the higher momentum of the liquid jets induces a sudden breakup of the liquid column and the rapid formation of a cloud of droplets that can be well modeled using the discrete blob approach. The injection conditions in typical gas turbine premixing ducts are quite different from diesel injection, because of the much lower kinetic energy level of liquid. As a consequence it can be assumed that jet behavior is not the one defined as atomization regime but, at least in the conditions of reference for this paper, the peculiarity of orthogonal interaction between liquid jet and high-momentum (i.e. pressurized, high-velocity) gas stream results in a sort of second wind–induced regime [8] whose characteristics differ from either extremely controlled or fully developed atomization regimes. An indirect confirmation of this assertion comes from the observation that the jet penetration increases nearly linearly with the injection velocity differently from the full atomization regime where a substantial independence of the penetration with respect to the injection velocity can be observed.

An eulerian approach to the modeling problem has been attempted by several researchers [9, 10, 11]. The basic approach has been the description of the spray bending by accounting for the drag force exerted on the jet cross section and the modeling of the liquid atomization and spray breakup by invoking either internal disturbances leading to jet disruption [10] or the effect of the aerodynamic stress [9,10,12]. In spite of the relative numerous number of models proposed none of them proved to be practically usable in a predictive and quantitative modeling activity. Some more refined models, taking advantage of numerical tools aimed to the description of the liquid-gas interface behavior, have been proposed [13-16].

Of particular relevance is the model proposed by Mashayek and coworkers that exploited a series expansion of the solution of the motion equation of the liquid-gas interface, originally developed by Gonor and Zolotova [17], allowing for reconstructing small to moderate deformations of the interface in axisymmetric conditions.

In spite of their relevant computational demand these models are generally limited to relative low Weber numbers and do not account for the liquid removal from the liquid column due to the airflow shearing effect. For these reasons they can be hardly used to simulate the jet behavior in the typical conditions of a gas turbine premixing duct.

An alternative approach is the one pursued, among the others, by [6, 16, 5] that is based on a lagrangian/eulerian computational scheme. For instance Ashgritz proposed to combine some simple models, which calculate the jet shape and trajectory, with an improved Lagrangian droplet tracking scheme which accounts for the secondary breakup of the droplets. In this way a computationally affordable solution could be obtained [16].

The approach followed in the present paper follows the above reported guidelines. In addition the description of the liquid jet trajectory does not rely on some semi-empirical correlation but it is computed by numerically solving the force balance equations along a curvilinear coordinate coincident with the jet axis. The advantage in this case is that the model is not restricted, in line of principle, to the canonical orthogonal intersection between the liquid jet and the air flow. In addition the air velocity profile can be arbitrarily assigned making the model more suitable for use in practical devices modeling.

More details will be given in the following paragraph but, in essence, the physical model presented here represents an enhancement of a previous model, described in details in earlier work [1]. The model describes the liquid jet as a liquid column bent, deformed and atomized by the drag force of a transverse airflow. The momentum exchange between phases responsible for the bending is described in an eulerian frame of reference, also accounting for the continuity of the liquid medium and for the heat transfer between the jet and the surrounding airflow, whereas the mass removal and the jet deformation are modeled within a lagrangian frame by resorting to purposely adapted sub-models. Major improvements were made in the sub-model describing the liquid jet cross-section deformation and by introducing a balance equation accounting for the heat exchange between the jet and the air crossflow. This approach, in spite of its simplicity, will be proved to be effective in reproducing the jet trajectory with a relatively low computational effort and a satisfactory degree of agreement with a large set of experimental data built up in several experimental campaign made by the same research group in high pressure and high temperature conditions significant to the gas-turbine aeropropulsion engines [18-21].

The jet describing model will then be included in a CFD code and the results on the jet trajectory and airflow velocities will be compared, in some relevant cases, to the ones obtained by using particle image velocimetry (PIV) [22].

MODEL DESCRIPTION

Transport equations

Momentum balance equations used to determine the jet trajectory, have been formulated in an eulerian frame of reference. The adoption of a curvilinear coordinate system $\xi\eta\zeta$ allows a better description of the physical problem. Keeping η everywhere tangent and equiverse to the jet trajectory, the problem is one-dimensional if liquid velocity *V*, diameter *D* and deformation *k* are assumed to be uniform in directions normal to the jet trajectory.

In this case: $\frac{\partial}{\partial\xi} = \frac{\partial}{\partial\zeta} = 0$ and the transport equation for a generic variable ϕ along the curvilinear coordinate η can be written as:

$$\frac{\partial}{\partial\eta}(\rho V\phi) = \frac{\partial}{\partial\eta}\left(\frac{\Gamma}{J} q_{22} \frac{\partial\phi}{\partial\eta}\right) + S(\eta) \tag{1}$$

where V is the so-called contravariant velocity along η direction:

$$V = A_{21} \cdot u + A_{22} \cdot v + A_{23} \cdot w \tag{2}$$

and

$$q_{22} = A_{21}^2 + A_{22}^2 + A_{23}^2 \tag{3}$$

In the present case the flow is two-dimensional (in the cartesian space) that is $w = 0$ and $A_{23} = 0$, therefore the contra-variant velocity and the area become:

$$V = A_{21} \cdot u + A_{22} \cdot v \tag{4}$$

and

$$q_{22} = A_{21}^2 + A_{22}^2 \tag{5}$$

In Eq (1) Γ is the transport property of ϕ, *J* is the jacobian of the coordinate transformation from cartesian to curvilinear and $S(\eta)$ is the source term.

Equation (1) can be integrated over a finite control volume and transformed into a difference equation by applying the Gauss theorem. Being *N* and *S* the north and south face of the control volume:

$$\begin{aligned}&(\rho V\phi)_N - (\rho V\phi)_S = \\ &= \left(\frac{\Gamma}{J} q_{22} \frac{\partial\phi}{\partial\eta}\right)_N - \left(\frac{\Gamma}{J} q_{22} \frac{\partial\phi}{\partial\eta}\right)_S + S_\phi\end{aligned} \tag{6}$$

Where the source term has the following form

$$S_\phi = \int_{\Delta\eta} J \cdot S(\eta) \cdot d\eta \tag{7}$$

It can be assumed that:

1) the fluid that moves inside the stream tube is a liquid, which density is a constant: $\rho = \rho_L$.
2) solving the transport equation for the momentum in *x* and *y* direction the diffusion term Γ is equal to dynamic viscosity $\mu = \mu_L$.
3) q_{22} is the square of the area of the section of the liquid jet, $q_{22} = A^2$.
4) The jacobian of the coordinates transformation is equal to the volume of elementary cell, $J = \Omega$.

With these assumptions eq. (6) becomes:

$$\begin{aligned}&(\rho_L V\phi)_N - (\rho_L V\phi)_S = \\ &= \left(\frac{\mu_L}{\Omega} A^2 \frac{\partial\phi}{\partial\eta}\right)_N - \left(\frac{\mu_L}{\Omega} A^2 \frac{\partial\phi}{\partial\eta}\right)_S + S_\phi\end{aligned} \tag{8}$$

The area of the section is defined on the boundary of the cell, while the volume Ω of the cell is defined in the centre of the same cell, therefore in the above equation an averaged value between two adjacent volumes has to be used:

$$\Omega_N = \frac{\Omega_i + \Omega_{i+1}}{2} = \frac{A_i(\delta_i + \delta_{i+1})}{2} \tag{9}$$

and

$$\Omega_S = \frac{\Omega_i + \Omega_{i-1}}{2} = \frac{A_i(\delta_i + \delta_{i-1})}{2} \tag{10}$$

where δ_i is the height of the generic control volume. Assuming a constant δ for all the cells eq. (8) can be rewritten by substituting the expressions for the cell volume (9) and (10) and considering that the term $\rho_L V$ is the mass flow rate across the control volume face *m*:

$$\begin{aligned}&m_i\phi_N - m_{i-1}\phi_S = \\ &= \mu_L A_i \frac{(\phi_{i+1} - \phi_i)}{\delta} - \mu_L A_{i-1} \frac{(\phi_i - \phi_{i-1})}{\delta} + S_\phi\end{aligned} \tag{11}$$

The equation (11) could be solved iteratively but to numerically stabilise the calculation and to get a convergence an upwind scheme for the convective terms was adopted assuming that on each face of the control volume the variable ϕ is equal to the upstream value (respect to fluid stream):

$$\begin{cases}\phi_N = \phi_i \\ \phi_S = \phi_{i-1}\end{cases} \tag{12}$$

In the model presented here three transport equations were solved for u and v components of the velocity and for the enthalpy.

In the case of the equations of transport of the *u* and *v* velocities, the source terms are given by the *x* and *y* components of the external aerodynamic force on the liquid jet:

$$\begin{aligned}F_u &= \frac{1}{2} C_D \rho_G |u_\infty - u_i|(u_\infty - u_i) \frac{(D_i + D_{i-1})}{2} \delta \cdot \cos\theta_i \\ F_v &= -\frac{1}{2} C_D \rho_G |v_i| v_i \frac{(D_i + D_{i-1})}{2} \delta \cdot \sin\theta_i\end{aligned} \tag{13}$$

where θ_i is the local angle of the axis of the liquid jet respect to the normal to the cross-flow and C_D is a drag coefficient taking in account the actual efficiency of the momentum exchange between the airflow and the liquid jet.

For the equation of transport of the enthalpy, the source term represents the heat exchange between the air stream and the liquid surface and it is given by:

$$Q = \bar{h}_c A(T_\infty - T_S) \quad (14)$$

Where $\bar{h}_c$ is the forced convection heat transfer coefficient between the liquid surface and the airflow. It is evaluated from the correlations based on the experimental tests on isolated cylinders in cross flow made by Hilpert [23] as:

$$\bar{h}_c = \frac{C\,\mathrm{Re}_c^n\,\Lambda_g}{D} \quad (15)$$

Where Re_c is the Reynolds number based on the cylinder diameter D, Λ_g is the thermal conductivity of the air and the constants C and n are functions of Re_c.

A critical task in the model elaboration has been the definition of effective sub-models for jet cross section deformation and primary atomization (i.e. mass removal from the liquid column). Furthermore, the determination of the drag coefficient C_D also required some care. All these steps present an intrinsic difficulty being generally associated to complex mathematical descriptions and scarcity of simplified models validated in condition significant to the ones here considered. It has been decided to attempt the exploitation of conceptual models available in literature for the different sub-mechanisms acting in the process and adopt and/or adapt them in order to describe as much as possible in a physically sound way the jet evolution. An effort has been made to reduce to a minimum arbitrary assumptions implied in these sub-models to give to the results the maximum degree of generality. In the followings the chosen models are presented along with a brief justification on the motivation of their selection.

Primary atomization of the jet

Several models are available in literature to describe the fragmentation of liquid drops and jets, but none of those was specifically thought for crossflow atomization. As a consequence, even if for secondary drop atomization the use of one of the standard models, developed for the co-flow case appears to be reasonable, less persuasive is to make the same assumption for the primary jet breakup modeling, since in this latter case a dominant role is played by the interaction of the liquid jet with the orthogonal air flow. In this process both the energy transfer and the interface evolution are characterized by a strong mutual interaction and develop with a more complex three-dimensional pattern than typical cases for which classical models have been built up. The peculiarity of cross-flow atomization must be searched in the geometry of the jet, stressed and bended in the airflow direction, and in the strong deformation of its cross-section, which produces a pressure distribution around the liquid jet completely different from that observable around a jet injected in still air. For instance one of the main problems connected to the adaptation of such models to cross-flow problems is the inability to capture the effects of the strong wake settling downstream the liquid column. The wake affects the jet cross-section geometry and, as a consequence, the mechanism of drop and ligament stripping from the liquid surface.

As matter of facts, in the typical conditions encountered in gas turbine premixing ducts the Weber numbers of the gaseous flow are such that the primary atomization process is dominated by liquid stripping mechanisms promoted by the air drag action and by the onset of oscillative perturbations on the interface. In this sense the mas removal of the jet is likely to be well reproduced by a stripping mechanism. Two major models have been developed to predict the amount of mass removed from a liquid jet due to these mechanisms. The first is the Kelvin-Helmholtz model [24-26], which deduces the mass removal rate from wavelength and growth rate of the fastest growing surface wave, calculated on the basis of a linearized stability analysis.

The alternative model, which has been adopted in the present paper, is the Boundary Layer Stripping (BLS) model [27,28]. The basic assumption is that the mass removed from the jet coincides with the flow rate in the liquid boundary layer at the separation point, taken for simplicity as the equatorial plane normal to the airflow. Extreme ease of implementation and absence of parameters to be experimentally tuned are the major benefits of this model. On the other hand the BLS model shows some unrealistic features, as it is based on the hypothesis of circular cross–section. Moreover, it does not account for the influence of surface tension on the atomization process.

The continuity equation can be obtained from Eq. (1) by setting $\phi = 1$. In this case the source term accounts for the mass removal from the jet due to atomization. By integrating over the finite control volume the Boundary Layer Stripping model provides the following expression for the mass shedding rate [27-29]

$$\dot{m} = \Delta\sqrt{8\pi d \rho_L \mu_L \left(\frac{\rho_G}{\rho_L}\right)^{1/3}\left(\frac{\mu_G}{\mu_L}\right)^{1/3} V_G} \quad (16)$$

This expression with the coefficient 8 is obtained for a double boundary layer on a cylinder (in the case of a spherical droplet the coefficient would be 6). In fact, the expression of the velocity outside the boundary layer is: $U_x = 2\,U_\infty \sin(2x/D)$ for the cylinder instead of $U_x = 1.5\,U_\infty \sin(2x/D)$ which is valid for a sphere.

Jet cross section deformation

A fundamental problem to be faced in the description of jet evolution is the choice of the most appropriate model describing the deformation of the jet cross section due to the action of the air drag force. Here the literature reference that appears to be easily applicable is the Taylor-Analogy-Breakup (TAB) model [30] that is largely used in the modeling of secondary atomization process of spherical droplets. It has been already suggested [31] that the deformation of the liquid jet cross-section before the break-up occurrence has significant analogies with the secondary breakup of a spherical drop and that, in the typical condition of a Jet-A1 jet injected in the premixing duct of a gas turbine, the jet breakup is likely to occur after a significant liquid removal by means of a shear atomization process. In these conditions the deformation of the liqid column can be modeled by means of a mathematical model analogous to the one underpinning the TAB model. On the other hand some modification to this model were required to adapt it to the different geometry and to the peculiar conditions considered in this work. In particular, the TAB sub-model was modified to take into account the cylindrical geometry of the elementary volume of the jet. Moreover, the dependence of aerodynamic force and capillary force (due to liquid surface tension) on jet cross-section deformation were taken into account, unlike the original TAB model.

In this paper both a formulation elaborated following the same procedure suggested in [30] and a modification based on the more recent work of Park and coworkers [32] were considered.

In the first formulation the cross section deformation parameter can be expressed as:

$$k = 1 + We\frac{C_F}{C_k C_b}\left[1 - \exp\left(-\frac{t}{t_d}\right)\left(\cos\omega t + \frac{1}{\omega t_d} sen\omega t\right)\right] \quad (17)$$

where:

$$We = \frac{\rho_G U_R^{\,2} r_0}{\sigma} \quad (18)$$

$$t_d = \frac{2\rho_L r_0^{\,2}}{C_d \mu_L} \quad (19)$$

$$\omega^2 = \frac{C_k \sigma}{\rho_L r_0^{\,3}} - \frac{1}{t_d^{\,2}} \quad (20)$$

This is essentially the same formulation of [30] and the parameters are the same of O'Rourke and Amsden substituting the radius of the jet to the radius of the droplets. The constants C_d and C_b have the same value of the spherical case (5 and 0.5, respectively) while the constant C_F is equal, for the cylindrical case, to C_d/π. Finally, the value of C_k is, in the cylindrical case equal to 6 [33].

The second formulation of the jet section deformation is based on the work of Park et al. [32] that proposed an improvement of the TAB model that takes in account the change of the aerodynamic force in consequence of the deformation assuming that the it is proportional to the surface of the deformed cross section. Under this hypothesis the aerodynamic force is multiplied by a factor $a^2 = r_0^2\left(1 + 0.5\,y\right)^2$ with respect to the original TAB value. In this case the force balance equation can be only solved by numerical means (for instance using a standard Runge-Kutta scheme). In addition the value of C_F is determined as 4/19 if it is assumed that the critical Weber number (i.e. at $y = 1$) is equal to 6 analogously to [30]

It must be stressed here that TAB model suffers from the strong limitation that the hypothesis of elliptical shape, assumed for the cross-sectional deformation of the liquid column, appears to be quite unrealistic for the fluid-dynamical conditions of gas turbine premixers. On the other hand, the more reasonable physical model of half-lenticular deformation, again based on an idea by Taylor, has as a drawback the assumption of instantaneous transition from circular to deformed shape, which makes it suitable for describing the behavior of liquid drops or columns exposed to a shock wave and apparently less appropriate for modeling the fast but gradual flattening of a liquid jet exposed to a steady cross-flow. Another significant limitation of the TAB model is that it takes in account only the fundamental mode of oscillation and neglects higher order modes. While this mode is the one with longer characteristic time, making the TAB model suitable for the description of drops and jet break-up at low relative velocities (i.e. low We numbers) it is very likely that at higher We numbers higher order modes can be excited modifying in an unpredictable way the section deformation.

During the definition of the best jet model, it has been also evaluated the possibility of taking in account the effect of surface tension on the jet deformation in different ways. In the following section the results of this procedure will be given.

Drag coefficient determination

The correct determination of the drag forces acting on the jet requires the assumption of a suitable value of the drag coefficient. This problem has been faced, in the literature, in different ways starting either from experimental or theoretical point of view. Adelberg [34] reports a constant value of 1.2 for fully developed turbulent flow over a circular cylinder. From a simplified point of view the drag coefficient is dependent on fluid-dynamic conditions, i.e. the Reynolds number, and on the degree of flattening of the jet cross-section. In literature such a kind of correlation for elliptic geometry was investigated either for viscous flow with very small Reynolds number [22,23], or for oscillating flow [24], or to design elliptical airfoils with zero or little angle of attack between the airflow and the major axis [25]. In all those cases the drag coefficient evaluation follows from the numerical calculation of the gas flow field. A similar procedure was implemented by Nguyen and Karagozian [17], resulting in a constant value of about 1.2÷1.4 for high Reynolds number and low subsonic crossflow over an elliptical cylinder. Unfortunately no dependence on the degree of deformation was provided. Wu et al. [13] assessed a C_D value of about 1.7 on the basis of a regression over experimental data collected for different injected liquids which is fairly a high value.

In the present paper it has been chosen to keep the drag coefficient value as the one of an inviscid rigid cylinder computed as a function of the local Reynolds value (computed using the relative velocity and the actual jet diameter values at each jet position). In this way the variation of the drag force due to the variation of the drag coefficient has been mainly accounted for by taking in account the jet cross-section enlargement (due to the jet cross section flattening) and the progressive change of the relative velocity component in the direction of airflow (due to the jet deceleration and bending). The formulation used was the one based on experimental data of Wieselberger and reported in the book of Schlichting:

$$\mathrm{Re} = \frac{\rho_g \cdot U_{rel} \cdot D}{\mu_g} \quad (21)$$

where ρ_g and μ_g are the density and viscosity of the air stream and U_{rel} is the relative velocity of the cylinder with respect to the air strema.

JET TRAJECTORY MODEL VALIDATION

The final aim of the work was the implementation of the jet model in a CFD code in order to simulate the mutual influence of the liquid jet, acting as an obstacle, and the airflow with its dragging action. To this aim a preliminary validation of the liquid jet model had to be made. Validation has been made referring to an experimental set of about 300 jet trajectories relative to different test conditions, at ambient and elevated temperature and at high air density, with two different nozzle sizes and using either water or Jet-A1. The number of cases in each subset, relative to a fixed pressure, temperature, liquid and nozzle size, is reported in Table 1. In each subset the test cases differ each other for the air and/or liquid velocities. For each case, starting from an ensemble of 1000 shadowgraphic images, the average value of the jet trajectory has been obtained by using an automated statistical procedure. A detailed discussion of the procedure and of the experimental set-up is given in [18-21].

		P (MPa)	1.0	1.6	2.0	2.0
		T (K)	300	600	300	600
Liquid	d_0 (mm)					
Water	0.3	RMS	0.097			0.144
		#	25			35
	0.5	RMS	0.159		0.134	0.110
		#	57		14	24
Jet-A1	0.3	RMS	0.0839	0.0668		0.114
		#	12	28		28
	0.5	RMS	0.100		0.0846	
		#	59		9	

Table I Condition of test for the liquid jet trajectory model. The RMS errors are relative to the model selected for implementation in the CFD code.

Several combinations of the different sub-models available have been tested. The best agreement with experimental data was found when the dependence of the aerodynamic external force term in the TAB equation on the jet cross section deformation was assumed to be linear, as well as assuming that surface tension term is directly proportional to the length of the elliptical perimeter of the deformed jet cross section.

The approach of Park et al. [32], which assumed an increase of the aerodynamic force term with the square of jet deformation, was found to give a poorer agreement with experimental data and in general an excess in jet flattening and bending particularly in the Jet-A1 high temperature cases. Since these are the condition of interest for the present work it was decided to not use this submodel.

The RMS errors computed for the different subsets of experimental conditions are reported in Table I for the finally selected liquid jet model. It must be stressed that the RMS errors of the computed trajectories with respect to the experimentally determined ones is of the same order of magnitude of those computed using the empirical correlations available in literature like those in [21, 35-37]. The proposed model is, in addition, not necessarily limited to the canonical case of nearly uniform air velocity profile and orthogonal intersection of the liquid jet and airflow. For these reasons it can be used also in more complex geometries unlike the available empirical models. Of course the model has to be considered validated only for the range of conditions where it has been validated but there is no limitation, in line of principle, to its application to a wider range of operating conditions.

IMPLEMENTATION OF THE LIQUID JET MODEL IN THE CFD CODE

It has been chosen to attempt the implementation of the liquid jet model in an in house developed CFD code (BODY3D). This has allowed for an easier adaptation and a faster test procedure. The BODY3D is a finite volumes package allowing the simulation of 3D incompressible, heterogeneous, reacting flows in an arbitrary geometry with structured meshes. The coupling between the liquid jet model and the CFD code has been made possible by preparing a meshing procedure that, on the ground of the results of the liquid jet model, generates a mesh with a set of inactive cells in correspondence of the liquid jet position. By imposing appropriate bondary condition it is possible to simulate the interaction of the liquid column with the airflow. On the other hand, the liquid removal from the jet (by means of the BLS submodel) generates droplet injection points along the liquid jet trajectory. Liquid mass, average droplet size and initial velocities of the droplets have been determined on the ground of the liquid velocity and mass flow rate in the boundary layer of the liquid column (generated in the liquid jet model). The droplets detached from the jet can be tracked by means of a Lagrangian tracking algorithm accounting for the mutual interaction of the droplet with the surrounding airflow including the heat transfer and evaporation process.

COMPARISON OF CFD RESULTS WITH PIV MEASUREMENTS

The CFD results will be compared with the PIV measurements in some selected cases in order to gain a first insight of the numerical model potentials and of possible improvements.

The experimental results used to validate the CFD model have been presented elsewhere in the same conference [38]. Three conditions, relative to injections of Jet-A1, are considered:

a T=300 K, P=10 bar, V_G=25 m/s, V_L= 40 m/s (q = 176);
b T=600 K, P=20 bar, V_G=25 m/s, V_L= 40 m/s (q = 176);
c T=300 K, P=10 bar, V_G=44 m/s, V_L= 36 m/s (q = 46).

The first two conditions are relative to the same q value and exhibit a similar liquid trajectory putting in evidence the role of the air temperature on the jet evolution, that is accounted for by the significantly higher We number. The third conditions exploit a case with significantly lower q value and comparable We values. In this case the jet is significantly more bent than in the first case but it is not expected that the atomization process changes significantly with respect to the first case.

In fig. 1 the airflow velocity patterns, as they have computed using the CFD code for the three conditions above mentioned, are reported. They can be, in a first approximation, compared with drop velocity pattern measured using a PIV technique and reported in [22]. This comparison is, obviously, only partly significant considering that in the present case the velocity maps are those of the air flow while in the [22] they refer to the droplets. In any case, the numerical patterns appear to be coherent with the experimental results.

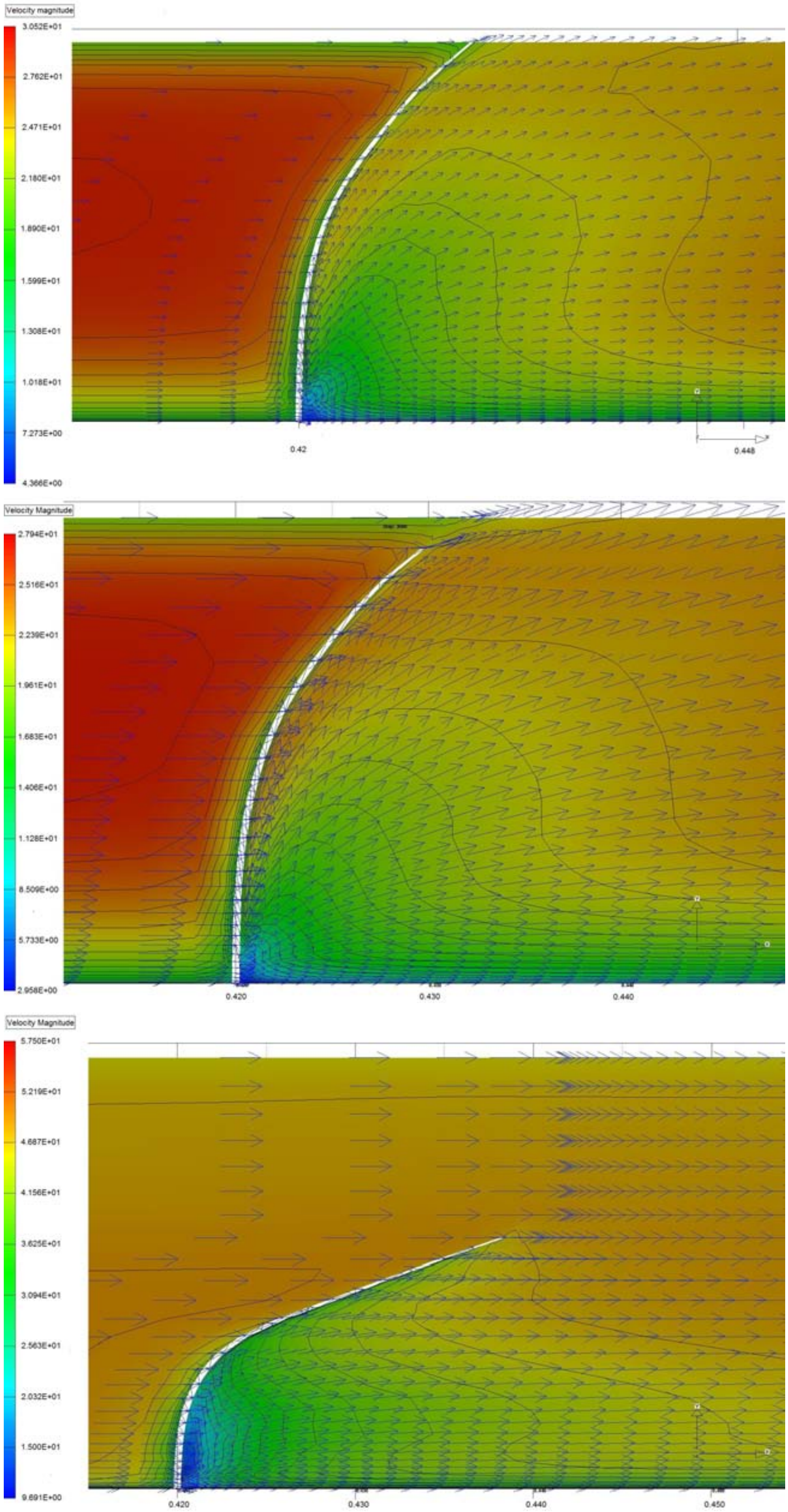

Fig 1 Velocity Magnitude patterns obtained by using the model for the three test cases considered. This results have to be compared with the experimental pattern reported in [22].

A more significant validation of the numerical results has been made possible by a post-processing procedure applied to the output of the droplets lagrangian tracking module.

The results of the post-processing procedure are reported in Fig. 2 by using the green lines. By comparison the corresponding results obtained by PIV are reported using red lines. In the figure velocity components along the x and z axis at three x positions are reported for the three case here considered. It can be observed from the diagrams that the progressive reduction of the z components (derived from the residual liquid velocities) with increasing x position and the increase of the velocity components along x are well reconstructed. The general agreement between experimental data and model results is fairly acceptable and the trends of the profiles are kept.

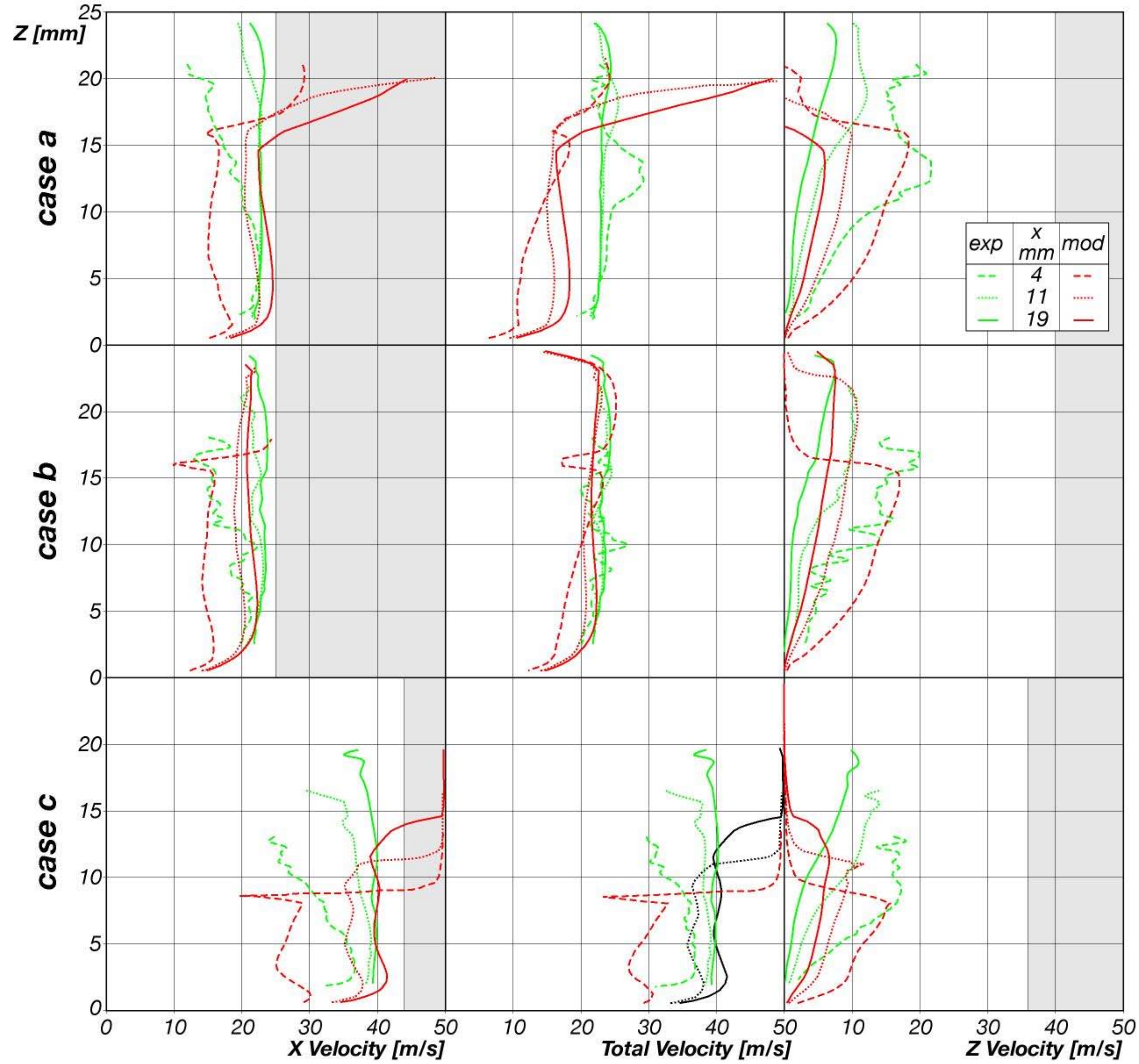

Fig. 2 Components along x and z of the droplets velocity profiles at three selected x positions and for the three cases. The model results are reported using red lines while the experimental ones [22] are reported using green lines.

CONCLUSIONS

In this paper an improved version of a numerical model aimed at decribing the behavior of liquid jets injected in an air crossflow at high temperature and pressure in conditions significant to gas-turbine engines has been preented. A hybrid scheme based on an eulerian-lagrangian computational scheme has been exploited and proved to be effective in reconstructing the liquid jet trajectory and to give promising results concerning the gaseous and liquid velocities.

The close comparison with a large set of experimental results and with few seected cases significant to the target conditions allowed for the vaidation of the procedure.

Further development of the liquid jet submodel will be pursued in order to better describe the cross section deformation rewriting the force balance in a more accurate way. In addtion a recursive procedure to couple the CFD code and the liquid jet submodel.

ACKNOWLEDGEMENT

This work has been partly supported by the European Community in the framework of the TLC contract, Contract N°AST4-CT-2005-012326.

REFERENCES

1. Ragucci, R., Bellofiore, A., Di Martino, P., Cavaliere, A.: Modeling the injection of liquids in crossflowing airstreams, 10th International Conference on Liquid Atomization and Spray Systems (ICLASS 2006), F4-03-247, 2006.
2. Reitz, R.D.: Modelling Atomization Processes in High Pressure Vaporizing Sprays, Atomization and Spray Technology, Vol. 3, pp. 309-337, 1987.
3. Reitz, R.D. and Diwakar, R.: Structure of High-Pressure Fuel Sprays, SAE Paper 870598, 1987.
4. Tanner, F.X.: Liquid Jet Atomization and Droplet Breakup Modeling of Non-Evaporating Diesel Fuel Sprays, SAE 1997 Transactions: Journal of Engines, Vol. 106, pp.127–140, 1998.
5. Rachner, M., Becker, J., Hassa, C., Doerr, T.: Modelling of the Atomization of a Plain Liquid Fuel Jet in Crossflow, Aerospace Science and Technology, Vol. 6, pp. 495-506, 2002.
6. Madabhushi, R.K.: A Model for Numerical Simulation of Breakup of a Liquid Jet in Crossflow, Atomization and Sprays, Vol. 13, pp. 413-424, 2003.
7. Khosla, S., Crocker, D.S.: CFD Modeling of the Atomization of Plain Jets in Cross Flow for Gas Turbine Applications, IGTI Turbo Expo: Combustion & Fuels, GT2004-54269, Vienna, Austria, June 2004.
8. Faeth, G.M.: Structure and Atomization Properties of Dense Turbulent Sprays, Proc. 23rd Int. Symposium on Combustion, pp. 1345-1352, 1990.
9. Clark, B.J.: Breakup of a Liquid Jet In a Transverse Flow of Gas, NASA Technical Note - TN D2424, 1964.
10. Kitamura, Y., Takahashi, T.: Stability of a Liquid Jet in Air Flow Normal to the Jet Axis, Journal of Chemical Engineering of Japan, Vol. 9, No. 4, pp. 282–286, 1976.
11. Heister, S.D., Nguyen, T.T., Karagozian, A.R.: Modeling of Liquid Jets Injected Transversely into a Supersonic Crossflow, AIAA Journal, Vol. 27, pp. 1727-1734, 1989.,
12. Nguyen, T.T., Karagozian, A.R.: Liquid Fuel Jet in Subsonic Crossflow, Journal of Propulsion and Power, Vol. 8, pp. 21-29, 1992.
13. Ryan, M.J.: CFD prediction of the trajectory of a liquid jet in a non-uniform air crossflow, Computers & Fluids vol. 35, 463–476, 2006.
14. Tsau, F., Elghobashi, S., Sirignano, W.A.: Prediction of a Liquid Jet in a Gaseous Crossflow, AIAA Paper 90-2067, 1990.
15. Ibrahim, E.A., Yang, H.Q. and Przekwas, A.J.: Modeling of spray droplets deformation and breakup, J. Propulsion 9, 651-654, 1993.
16. Mashayek, A., Jafari, A., Ashgriz, N.: A Model for Deformation of Liquid Jets and Droplets Subjected to Gaseous Flows, 20th Annual Conference on Liquid Atomization and Spray Systems of ILASS Americas, Paper 3B1, 2007.
17. Gonor, A.L., Zolotova, N.V.: Deceleration and Deformation of a Liquid Drop in Gas Stream, Izvestiya Akademii Nauk SSSR, Mekhanika Zhidkosti I Gaza, Vol. 2, pp. 58-69, 1981
18. Ragucci, R., Bellofiore, A., Cavaliere, A.: Statistical Evaluation of Dynamics and Coherence Breakdown of Kerosene and Water Jets in Crossflow, Proc. 19th Annual Conference on Liquid Atomization and Spray Systems (ILASS-Europe ’04), pp. 44-49, 2004.
19. Bellofiore, A., Cavaliere, A., Ragucci, R.: Air Density Effect on the Atomization of Liquid Jets in Crossflow, Combustion Science and Technology, 179, pp. 319-342, 2007.
20. Ragucci, R., Bellofiore, A., Cavaliere, A.: Trajectory and Momentum Coherence Breakdown of a Liquid Jet in High-Density Air Crossflow, Atom. Sprays vol. 17 (1), 47-70, 2007.
21. Ragucci, R., Bellofiore, A., Cavaliere, A.: Breakup and Breakdown of Bent Kerosene Jets in Gas Turbine Conditions, Proc. Comb. Inst., vol. 31, 2231-2238, 2007
22. Bellofiore, A., Cavaliere, A., Ragucci, R.: PIV Characterization of Sprays Generated by Crossflow Injection in High-Density Airflow, Proc. 22nd Annual Conference on Liquid Atomization and Spray Systems (ILASS-Europe ’08), 2008.
23. Kreith, F.: “Principles of Heat Transfer”, Donnelly Publishing Corporation, 1973.
24. Reitz, R.D.: Modelling Atomization Processes in High Pressure Vaporizing Sprays, Atomization and Spray Technology, 3 (1987) 309-337.Vol. 3, pp. 309-337, 1987.
25. Reitz, R.D., Bracco, F.V.: Mechanism of Atomization of a Liquid Jet, Physics of Fluids, Vol. 25, pp. 1730-1742, 1982.
26. Reitz, R.D. and Diwakar, R.: Structure of High-Pressure Fuel Sprays, SAE Paper 870598, 1987.
27. Ranger, A.A. and Nicholls, J.A.: Aerodynamic Shattering of Liquid Drops, AIAA Journal, Vol. 7, pp. 285-290, 1969.
28. Delplanque, J.P. and Sirignano, W.A.: Numerical Study of Transient Vaporization of an Oxygen Droplet at sub- and super-Critical Conditions, Atomization and Sprays, Vol. 4, pp. 325-349, 1994.
29. Bellofiore, A., Cavaliere, A., Ragucci, R.: Atomization and Bending of Coherent Jets in Crossflow, Proc. Joint Meeting of the Italian and Greek Sections of the Combustion Institute, p. 79, 2004.

30. O'Rourke, P.J. and Amsden, A.A.: The Tab Method for Numerical Calculation of Spray Droplet Breakup, SAE paper 872089, 1987.

31. Mazallon, J., Dai, Z., Faeth, G.M.: Primary breakup of nonturbulent round liquid jets in gas crossflows, Atomization and Sprays vol. 9, 291–311, 1999.

32. Park, J.-H., Yoon, Y., Hwang, S.-S., Improved TAB model for the prediction of spray droplet deformation and breakup, Atomization and sprays, Vol. 12, pp. 387-401, 2002.

33. Lamb, H.: Hydrodynamics, Dover Publications, 1932.

34. Adelberg, M.: Breakup Rate and Penetration of a Liquid Jet in a Gas Stream, AIAA Journal, Vol. 5, pp. 1408-1415, 1967.

35. Wu, P.K., Kirkendall, K.A., Fuller, R.P., Breakup Processes of Liquid Jets in Subsonic Crossflows, Journal of Propulsion and Power, Vol. 13, pp. 64-73, 1997.

36. Becker, J. and Hassa, C., Breakup and Atomization of a Kerosene Jet in Crossflow at Elevated Pressure, Atomization and Sprays, Vol. 11, pp. 46-67, 2002.

37. Stenzler, J.N., Lee, J.G., Santavicca, D.A. and Lee, W., Penetration of Liquid Jets in a Crossflow, Atomization and Sprays, vol. 16, p. 1, 2006.