Atomization and Bending of Coherent Jets in Crossflow

[1]A. Bellofiore, [1]A. Cavaliere, [2]R. Ragucci

1 Dipartimento di Ingegneria Chimica - Università Federico II, Naples - ITALY
2 Istituto di Ricerche sulla Combustione - C.N.R., Naples - ITALY

INTRODUCTION

The study of the interaction between a liquid jet and a gas crossflow is still far from a complete qualitative description of the fluid-dynamical process, and even more far from a satisfying modeling. The problem appears to be complex, due to the simultaneous presence of a liquid column, subject to the stresses induced by the air cross flow, bended and likely deformed by drag forces and pressure drop between windward and leeward side, and liquid fragments stripped from jet surface. The continuous stripping of small ligaments and drops reduces the coherent liquid jet's cross-section and so, starting at a certain distance from the nozzle outlet, jet undergoes coherence breakdown phenomena. Liquid particles, generated by the jet breakup, are easily dragged by the airflow surrounding the spray. In addition, the droplets can undergo to a secondary breakup process producing even smaller droplets. Inertia of the liquid jet, aerodynamic shear of the gas phase, surface tension and atomization are the main factors playing a rule in this process. Other effects that could be taken in account are the convection inside liquid phase due to drag force and pressure drop and the energy transfer taking place at all the turbulence scales. In particular the liquid turbulence and cavitation phenomena inside the nozzle are supposed to have a primary effect on the onset of the wave instabilities that propagate along the liquid surface and, enhanced by the interaction with the gas current, promote both the aerodynamic stripping of small fragments from the jet and the final breakdown of the liquid column.

Multiple influences of airflow characteristics (temperature, pressure and velocity field) and injection modes represent a challenging task to be faced in developing effective modeling tools to be used in a predictive way in design and testing activities. The experimental analysis of liquid jet in crossflow has highlighted the possibility to study the influence of the several classes of variables, such as design and operating parameters and liquid properties, by means of few adimensional numbers. The trajectory run by the liquid jet seems to be strongly influenced by the value of the liquid-to-air momentum ratio, usually named q number and defined as $q=\left(\rho_L v_L^2\right)/\left(\rho_G v_G^2\right)$. Schetz et al.[1], in an experimental study on the rise and propagation of instability waves on the surface of several liquids injected in a supersonic crossflow, state that also the dynamical behavior of the liquid column was dependent on the value of q. For low values they observed a violent oscillation of the whole jet, while at higher values at least the initial portion of the column seemed to be quite steady. Chen et al.[2] dealt with the modeling of the trajectory of a water jet in a low-pressure / high-speed airflow. The strategy they adopted was to discriminate three different zones of the spray, namely a liquid core region before the occurrence of the coherence breakdown, then a transition region characterized by the presence of large fragments produced by the jet disruption, and finally the fine spray zone. These classification was exploited to develop an empirical model, which calculates the trajectory of the windward profile of the spray by means of the product of three exponential terms, each of them dominating in one of the spray regions. As well as similar empirical approaches, those models find a strong limitation when one attempts to generalize the use of their predictions to different facilities or liquids. Basing on a nonlinear regression of experimental data collected from kerosene jets in a high-pressure air stream, Becker and Hassa[3-5] proposed a simple analytical correlation to predict the jet penetration as a function of the q number, of the downstream distance from the nozzle and of the injection diameter. The shape of the jet evolution was modeled by adopting a logarithmic dependence on the downstream distance. Such a behavior cannot be derived by any known theoretical approach based on the momentum balance equations, being the result of choices aimed to achieve the best fitting of the experimental data. As regards the effect of the liquid-to-air momentum ratio, the authors found a power-law dependence, with an exponent varying from 0.36 to 0.42 depending on the q range. It has been generally accepted that the jet penetration is related to the square root of q, that is the liquid-to-air velocity ratio and the square root of the density ratio, a parameter found to be relevant also in the study of the aerodynamic breakup of drops[6]. The employ of data regression techniques makes it difficult to obtain correlation with a precise physical meaning, such as the exponents found by Becker and Hassa, or the 0.44 proposed analogously in the above quoted work by Chen et al. However it can be thought that those exponents are not too far from the more meaningful value of 0.5. Schetz and Padhye[7] used this value to assess the penetration height of a water jet in a subsonic crossflow. More recently Wu et al.[8] proposed a model based on the analytical integration of the momentum balance equation written for the downstream velocity component of a finite element of the liquid column, tracked in a lagrangian way. The mass reduction of the element, due to the atomization process, is neglected, as well as the cross section deformation. The transverse component of the velocity is supposed to be constant, despite of the blocking effect of the air. The drag force exerted by the airflow is assessed by means of an average friction factor, supposed constant and empirically set to 1.696. An analogous model was developed by Ragucci et al.[9] and takes into account the effect of the atomization by hypothesizing a reduction of the diameter of the element, assumed to be linear with the transverse direction. In this case the model requires the experimental assessment of the drag factor and the jet breakdown height. The final analytical expression is more complex than Wu's, but it seems to give better results in the prediction of the penetration of water jets in a high pressure air current, probably because the higher air density makes not negligible the atomization process. Hence an even better agreement can be expected with fuel jets, being in this case lower the surface tension and then stronger the mass removal from the liquid column.

As regards the atomization process, in literature several models have been proposed for the fragmentation of liquid drop and jets, but none of those found was specific for crossflow atomization. The peculiarity of this case mainly consists of the geometry of the jet, stressed and bended in the airflow direction, and also the strong deformation its cross section is supposed to undergo, which takes to a pressure distribution around the liquid jet completely different from a typical diesel jet. For instance one of the main problems connected to the adaptation of such models to crossflow problems is the incapability to capture the effects of the strong wake settling downstream the liquid column. The wake is presumed to affect the jet cross section and, as a consequence, the mechanism of drop and ligament stripping from the liquid surface, and so there is no chance to take into account those effects by adopting models developed for drop disintegration or jet atomization in quiescent or co-flowing air. A fundamental support must come from the experimental observation, aiming to achieve a description, as detailed as possible, of the complex phenomenology of jets bended, stressed, deformed and atomized by the transverse airflow, and to clarify the underneath physical mechanisms. Unfortunately the optical diagnostic techniques, both traditional and laser, are still unable to investigate with sufficient resolution the near field of a spray, because of the high density of liquid fragments that pose serious difficulties to consolidated techniques such as Phase Doppler Particle Dynamics Anemometry (PDPDA) or Particle Image Velocimetry (PIV). In addition the study of the coherent jet dynamics is even stiffer because the weak scattering signal emitted by the jet surface is completely overcome by the surrounding droplets, having a much larger liquid-gas interface. In conclusion up to date the efforts did not give significant results in the understanding of the jet cross section deformation, or rather there is not even an experimental evidence whether such a deformation really occurs or not. Therefore at the state of the art there is a gap between the large amount of simple models, available for the prediction of the liquid jet trajectory under crossflow condition and summarized above, and the development of numerical models implemented in CFD codes, incorporating modules for jet dynamics and for spray generation and evolution in turbulent airflow. The gap consists of the lack of comprehension, and hence mathematical description, of the three-dimensional behavior of a jet and of the atomization mechanisms. In other words, the module for jet dynamics usually implemented in CFD codes is nothing more than a simplistic model for one-dimensional jet trajectory, with an additional TAB-based description[10] of the flattening of the jet cross section due to the aerodynamic pressure. This kind of approach has been recently followed by Rachner et al.[11]. Their model is able to take in account the liquid jet dynamics and breakdown by means a lagrangian tracking of isolated liquid drops emitted from the nozzle and simulating the behavior of the whole column. The shear breakup responsible of the atomization of the jet was predicted by the Boundary Layer Stripping model, firstly developed by Ranger and Nicholls[12]. The model also predicts the secondary atomization and evaporation of large fragments and droplets. Madabhushi[13] proposed a similar approach, the main difference being the use of a wave breakup model for the atomization process.

This paper aims to the development of a jet dynamics module to be implemented in a CFD code. A first requirement is the possibility to use that module in different contexts, mostly different liquids. That target can be achieved by pursuing a jet description as much as possible independent on arbitrary parameters, usually required by integral models, which could limit its field of application. The model is based on the momentum balance equations written on a finite thin slice of the liquid column jet. Although a lagrangian approach has been followed, the model considers also the forces necessary to maintain the column coherence, forces here identified as the inertia (normal) and viscous strains (tangential) acting between contiguous slices. In addition a simple atomization model, based on the Boundary Layer Stripping model, was adopted to provide a first assessment of the effect of the liquid mass removal on the jet dynamics. The model has been validated against experimental data obtained from the injection of kerosene and water in a high–pressure / high–temperature channel reproducing conditions typical of LPP gas turbines.

DESCRIPTION OF THE MODEL

Starting from the lagrangian approach introduced by Reitz[14], the general trend followed in the description of jets has been to represent the liquid column as a set of isolated drops emitted from the nozzle outlet. The temporal evolution of each drop is assumed to be representative of the whole jet dynamics[11,13]. In that way it appears to be difficult to model any interactions between the drop and the other parts of the liquid jet, de facto losing the continuity of the column. This paper aims to the definition of a modeling approach that first of all does not neglect the integrity of the liquid column from the injection up to the breakdown point. That should represent a first step in the construction of a physically consistent model of the full crossflow atomization problem, by means of a sequential enlargement combination of the separated effects contributions. At first this model have been provided with a simple atomization model to coarsely assess the effect of the mass removal on the jet trajectory. The resulting model here presented is based on the lagrangian tracking of discrete elements of the liquid jet. Since the cross section deformation was neglected, each element is cylindrical, with finite constant height δ and diameter D variable along the trajectory due to the mass removal. A first difference with the Reitz' approach is the use of a non uniform temporal discretization, due to the need for an easier evaluation of the inner forces responsible of the continuity of the column, as explained in the following.

A Cartesian system is chosen with the origin in the geometrical center of the nozzle outlet, under the hypothesis of two-dimensional steady problem and taking *x* and *z* axes respectively oriented along the gas flow and initial velocity of the jet. The momentum balance and the continuity equations are written on the finite cylindrical element. The unknown quantities are the diameter and the two components of the velocity for each discrete element.

The continuity equation takes into account the liquid mass variation due to the atomization occurring during the time $dt_i = \delta / v_i$, where v_i is the liquid velocity in the *z* direction:

$$\frac{\rho_L \frac{\pi D_{i+1}^2 \delta}{4} - \rho_L \frac{\pi D_i^2 \delta}{4}}{\delta / v_i} = -\dot{m} \tag{1}$$

where ρ_L is the liquid density, D_i the diameter of the circular cross section, and $\dot{m}$ is the mass removal rate and will be formulated later by choosing an atomization model.
As regards the momentum balance equations, the momentum flux changes between two contiguous sections because of either the aerodynamic interaction with the airflow and the momentum exchange with the bordering parts of liquid. Along the *x* direction that momentum exchange can be expressed in terms of viscous tangential stresses:

$$\frac{\rho_L u_{i+1} \frac{\pi D_{i+1}^2 \delta}{4} - \rho_L u_i \frac{\pi D_i^2 \delta}{4}}{\delta / v_i} = c_{Dx} \cdot \frac{1}{2} \rho_G \left| u_\infty - u_i \right|^2 D_i \delta + \left(\mu_L \frac{u_{i+1} - u_i}{\delta} - \mu_L \frac{u_i - u_{i-1}}{\delta} \right) \frac{\pi D_i^2}{4} \tag{2}$$

u_i represents the liquid velocity along *x*, c_{Dx} is the friction factor referred to the liquid element area $D_i\delta$ normal to the *x* direction, ρ_G the air density, u_∞ the undisturbed airflow velocity and μ_L is the liquid viscosity.
The momentum balance equation in the *z* direction contains the inertial forces between elements traveling at different velocities:

$$\frac{\rho_L v_{i+1} \frac{\pi D_{i+1}^2 \delta}{4} - \rho_L v_i \frac{\pi D_i^2 \delta}{4}}{\delta / v_i} = -c_{Dz} \cdot \frac{1}{2} \rho_G v_i^2 \cdot \pi D_i \delta + \left(\frac{1}{2} \rho_L v_{i+1}^2 - \frac{1}{2} \rho_L v_{i-1}^2 \right) \frac{\pi D_i^2}{4} \tag{3}$$

In that equation c_{Dz} is the friction factor referred to the area $\pi D_i \delta$ wetted by the airflow.
In order to close the problem an atomization model has to be chosen. The Boundary Layer Stripping model was firstly developed by Ranger and Nicholls[12] for describing the atomization of liquid drops exposed to airflow, and it is based on an idea attributable to Taylor. Delplanque and Sirignano[15] used this model to simulate the aerodynamic stripping a liquid oxygen droplet undergoes in supercritical non-quiescent environment due to the reduced surface tension. The strong hypotheses required by this model limit its adequacy to the crossflow case, but at the same time those conceptual assumptions turn loose the model validity from the experimental assessment of empirical parameters. On the other hand it must be stressed that this model does not take into account in any way the influence of the Weber number on the atomization process, influence that has been largely demonstrated, so that the use of such a model appears to be necessarily not the ultimate choice. A satisfying atomization model cannot be thought without considering a strong dependence on the interaction between aerodynamic and capillary forces. The model assumes that the mass removed from the liquid jet by the aerodynamic stripping coincides with the mass flowing out of the liquid boundary layer, settled around the column, through the *equatorial layer* defined as the layer normal to u_∞ and containing the geometrical center of the liquid column circular cross section. The stripping point should be more correctly placed in correspondence of the boundary layer separation point, but the error occurring by placing the stripping point at the equator of the circular section is not larger than the error that would occur by attempting to assess the position of the separation point with the available models[15], validated for solid spheres. Furthermore at the equatorial layer there is a strong simplification of the equations presented later.
Basic hypotheses of the model are then the absence of jet deformation and furthermore the steadiness and incompressibility of the airflow. A double boundary layer is supposed to exist and a coordinate system is taken with the curvilinear coordinate *x*, tangential to the boundary layer, and its normal axis *y*, as shown in the figure 1 (y_L has opposite orientation). *U* and *V* are respectively the

x and y components of velocity in the boundary layers.
With respect to the quoted works, the model has been here adapted to cylindrical geometry.

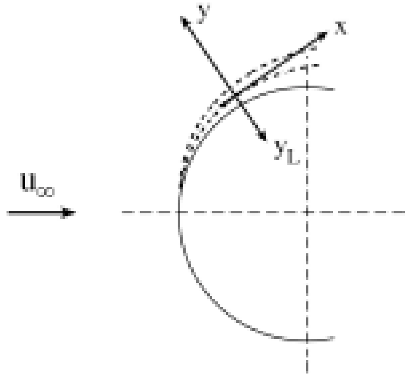

Fig.1 The double boundary layer and the Cartesian coordinates.

The continuity equation, under the above recalled hypotheses, takes to

$$\frac{\partial U}{\partial x}+\frac{\partial V}{\partial y}=0 \tag{4}$$

both for the gas and liquid boundary layer. For the gas phase the momentum balance along x can be formulated as[16]:

$$U\frac{\partial U}{\partial x}+V\frac{\partial U}{\partial y}=-\frac{1}{\rho_G}\frac{\partial p}{\partial x}+\nu_G\frac{\partial^2 U}{\partial y^2} \tag{5}$$

The pressure gradients across the y direction can be neglected, so that[16]

$$\frac{\partial p}{\partial x}\approx\frac{dp}{dx}=\frac{dp_e}{dx}=-\rho_G U_e\frac{dU_e}{dx}$$

The last result comes out from the Bernoulli equation written outside the boundary layer. U_e depends only on x. That balance is as usual integrated along y between 0 and ∞, obtaining the von Karman integral equation. The equation can be further elaborated and the result is

$$\frac{\partial}{\partial x}\int_0^\infty U(U_e-U)dy+\frac{dU_e}{dx}\int_0^\infty (U_e-U)dy=\nu_G\left.\frac{\partial U}{\partial y}\right|_{y=0} \tag{6}$$

Similarly for the liquid phase the momentum balance in the x direction, integrated across y, is

$$\frac{\partial}{\partial x}\int_0^\infty U_L^2 dy=-\frac{1}{\rho_L}\frac{dp}{dx}\delta_L-\nu_L\left.\frac{\partial U_L}{\partial y}\right|_{y=0} \tag{7}$$

By assuming the continuity of the tangential stresses across the interface between the boundary layers, another equation is obtained:

$$-\mu_L\left.\frac{\partial U_L}{\partial y_L}\right|_{y=0}=\mu_G\left.\frac{\partial U}{\partial y}\right|_{y=0} \tag{8}$$

The equations 6 to 8 can be solved by assuming velocity profiles expressed in function of a number of parameters equal to the number of available equations. Following the suggestion by Taylor

$$U_L=AU_e\exp\left(-\frac{y_L}{\alpha_L\sqrt{x}}\right) \tag{9}$$

$$U = U_e \left[1 - (1 - A)\exp\left(-\frac{y}{\alpha_G \sqrt{x}}\right)\right] \tag{10}$$

The parameter *A* has a physical meaning, representing the velocity of the interface between the boundary layers, adimensionalized to U_e. Outside the boundary layer the approximation $\mathrm{Re} \to \infty$ can be considered valid, and so the theory of potential flow around a circular section can be used to obtain

$$U_e = U_e(x) = \frac{3}{2} U_\infty \sin\left(\frac{2x}{D}\right) \tag{11}$$

The problem is now to assess the liquid flow, in the boundary layer, through the equatorial layer, where $x \approx \frac{\pi D}{4}$ and so the following assumption cab be made:

$$\frac{dU_e}{dx} \approx 0 \text{ and } U_e = \frac{3}{2} U_\infty$$

The equation 8 takes to

$$A = \frac{\frac{\mu_G}{\alpha_G}}{\frac{\mu_G}{\alpha_G} + \frac{\mu_L}{\alpha_L}} \tag{12}$$

while from the momentum balances it is possible to derive

$$\alpha_G = \sqrt{\frac{8}{3} \frac{\nu_G}{(1 + A) U_\infty}} \tag{13}$$

$$\alpha_L = \sqrt{\frac{8}{3} \frac{\nu_L}{A U_\infty}} \tag{14}$$

The interface velocity can be supposed to be negligible with respect to the undisturbed airflow velocity[12]. By substituting equations 13 and 14 in 12 and assuming $A << 1$, the parameter *A* can be evaluated as

$$A = \left(\frac{\rho_G}{\rho_L}\right)^{1/3} \left(\frac{\mu_G}{\mu_L}\right)^{1/3} \tag{15}$$

Referring to a discrete cylindrical element, the liquid mass flows out at the stripping point through two areas of height equal to δ and deepness equal to the boundary layer thickness, so that

$$\dot{m} = 2\delta \int_0^\infty \rho_L U_L \, dy_L = 2\delta \int_0^\infty \rho_L U_e A \exp\left(-\frac{y_L}{\alpha_L \sqrt{x}}\right) dy_L = 2\delta \rho_L A U_e \alpha_L \sqrt{x} \tag{16}$$

Substituting the values of *A* and α_L the final expression of the mass removal rate is

$$\dot{m}_i = \delta_i \sqrt{6\pi D_i \rho_L \mu_L \left(\frac{\rho_G}{\rho_L}\right)^{1/3} \left(\frac{\mu_G}{\mu_L}\right)^{1/3} U_\infty} \tag{17}$$

With these formulation the equation 1 to 3 can be solved for each discrete element, that is to say that for each step *i* a system of three equations, non linear and coupled, must be solved to find the values of D_{i+1}, u_{i+1} and v_{i+1}. An iterative method has been used to this purpose, so that a first attempt assessment of the unknown quantities is required. That is possible by preliminary solving a simplified problem formulated by assuming that in the accumulation term of the momentum balance equations the value of D does not change, and that allows uncoupling the equations and solving them by evaluating separately the unknown quantities. The integration stops when the We_G drops below a value set to 12 in analogy with liquid drops. It can be assumed at that point the jet no longer undergoes aerodynamic stripping.

RESULTS AND DISCUSSION

Experimental data, collected from kerosene and water jets in a high-pressure air crossflow and presented elsewhere[17], were used to validate the results of the model described above. In the following this model will be referred to as ABC model. Figure 2 reports a comparison between the model (green line) and experimental data (orange dots). The measured trajectory was truncated nearly in correspondence of the experimentally assessed breakdown height. In addition the model proposed by Wu et al.[8] (blue line) and two different forms of the model by Becker and Hassa[3-5] were compared to the ABC model and the data. As before mentioned, the model by Wu et al. was developed to fit the measurements on water jets in relatively low-pressure conditions. The two versions of the modeled by Becker and Hassa, validated on kerosene sprays in a high-pressure airflow, target different q ranges: the former was proposed for q values up to 40, whereas the latter is meant to fit better liquid jets with q lower than 12, as indicated in the figure captions.

The first couple of figures, i.e. 2.W-1 and 2.K-1, reports an example of water and kerosene jet trajectory at low q values. In that case the bending of the jet appears to be so strong that it can reach not even the middle height of the channel. Figures 2.W-2 and 2.K-3 present higher q conditions, so that both the liquids show a higher penetration.

As regard the literature models, figure 2 highlights a general aspect of their behavior. The model by Wu et al. reveals its attitude to reproduce sprays that oppose a tougher resistance to the aerodynamic drag, resulting in a weaker bending. This behavior is due to the choice to neglect the velocity decrease along the transverse direction, the jet cross section deformation and, most of all, the reduction of the jet inertia due to the atomization, so that fits well jets of liquids with an higher surface tension, such as water, injected in low-density airflows. For the former group of reference data chosen here, that is water jets in high-density airflow reported in the left column, the model behaves quite good, although not as well as for the data Wu et al. used to validate it. The right column, relative to kerosene, denotes a worse capacity of prediction of the model by Wu et al. On the contrary it must be underscored that the models by Becker and Hassa, and mainly the version for larger q values, works in a satisfying way for kerosene jet, predicting well the marked curvature observed starting from about 2/3 of the penetration height and indicating the liquid column is already there strongly stressed, maybe curled and grown thinner so that its resistance to the drag is reduced even before the occurrence of the ultimate breakdown. Since those models have been obtained from a non-linear regression of kerosene experimental data, they appear to be unable to fit water jets, as indicated by figures 2.W-1 and 2.W-2. In that case the influence of the atomization on the jet inertia is less tough than for kerosene, and so in particular in figure 2.W-2 a bad agreement is evident.

The ABC proposes as an initial step to develop a model capable to overcome the lack of generalization intrinsic of the other integral models. Figures 2.W-1 and 2.W-2 report a good overall agreement of the model with the data for water, even though it is not able to capture the local curvature of the experimental trajectories. In particular near the nozzle the model seems to overestimate the drag. The slope of the experimental dot remains higher up to about 2/3 of the penetration height. After that the bending increases and the experimental slope becomes soon lower than the model's. This behavior is imputed to a loss of resistance of the liquid column, which could be due either to an increasing mass removal by liquid stripping or to an effect of the deformation, curling and consequent local thinning of the jet. The comparison of ABC model with data for kerosene (figures 2.K-1 and 2,K-2) indicates that the initial drag is better assessed than in case of water, but here the model is not capable to predict the stronger bending of the jet. Again, this lack of agreement could be due to a tougher effect of the atomization on the jet trajectory. That would mean the BLS model fails underestimating the mass removal rate from low surface tension jets.

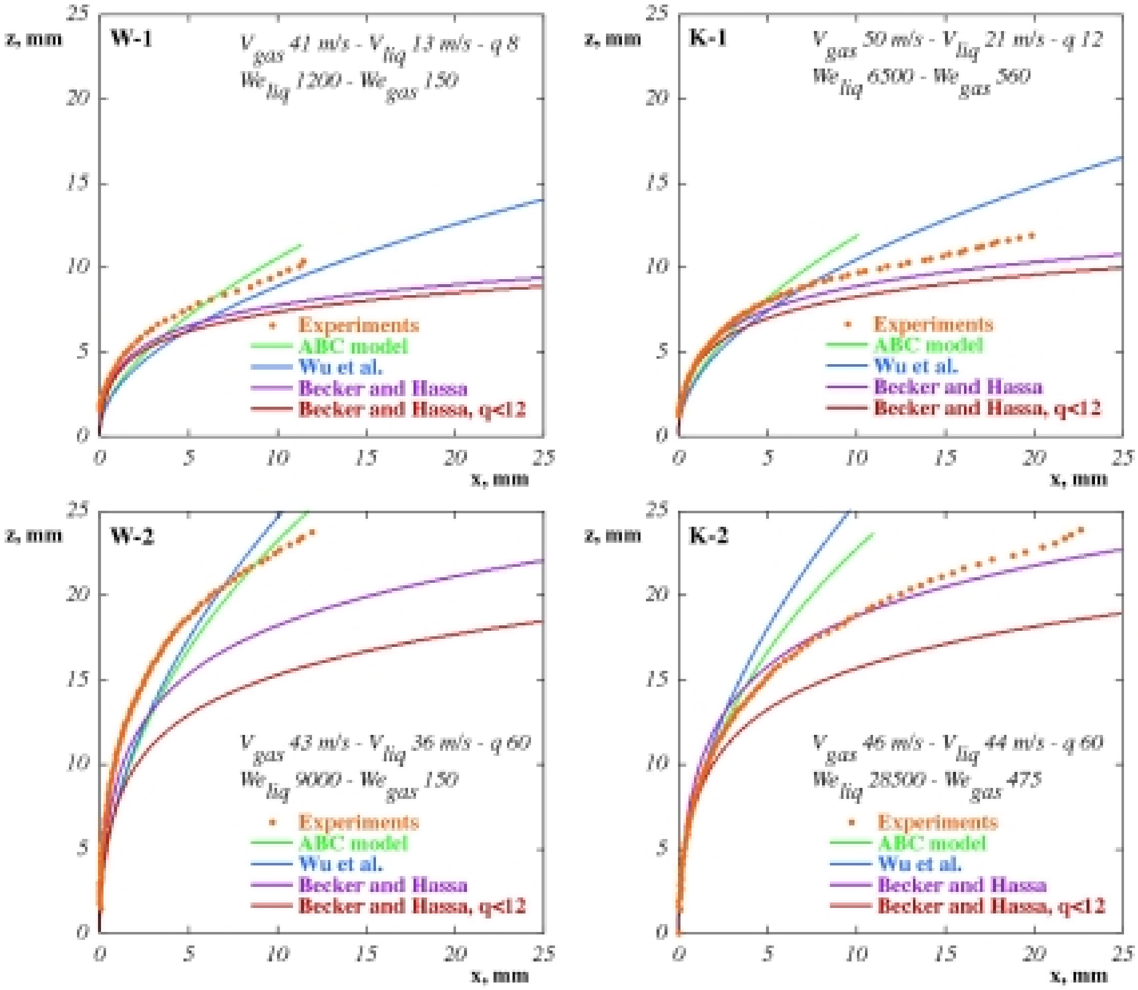

Fig. 2 Comparison between experimental spray windward profiles and trajectories predicted by the ABC model and other model from literature

Figure 3 reports the values of the residual average error (RAE) against the q number and We_G. This parameter is defined as

$$RAE = \frac{\sum (Z_{ic} - Z_{is})^2}{\sum (Z_{ic} - Z_{is})^2 + \sum (Z_{ic} - \overline{Z_s})^2} \tag{19}$$

where Z_{ic} and Z_{is} are the ordinate values of correlation points and experimental points, while $\overline{Z_s}$ is the average value of the experimental point ordinates. The residual average error is commonly used as an indicator of the agreement between experimental data and model. Figure 3.1, gives information about the behavior of the RAE as a function the liquid-to-air momentum ratio for water jets. Apart from the second version of the model by Becker and Hassa, completely failing predictions, the other three models appear to fit quite well the trajectory of water jets for elevated q values, where the bending is not too strong. On the contrary at low q numbers the RAE increases slightly, mainly for the model by Becker and Hassa that as expected overestimates the bending of low-energy water jets. The ABC model shows a good agreement with data across the whole explored q range. An analogous plot is presented in figure 3.2 for kerosene. In that case the main difference is that the first version of the model by Becker and Hassa fits very well the experimental profiles, especially at higher q values. The model by Wu et al., working fine with water, reveals its substantial inadequacy to be generalized to liquids with higher surface tension. The ABC model appears to be the most flexible, giving good overall results for both water and kerosene.

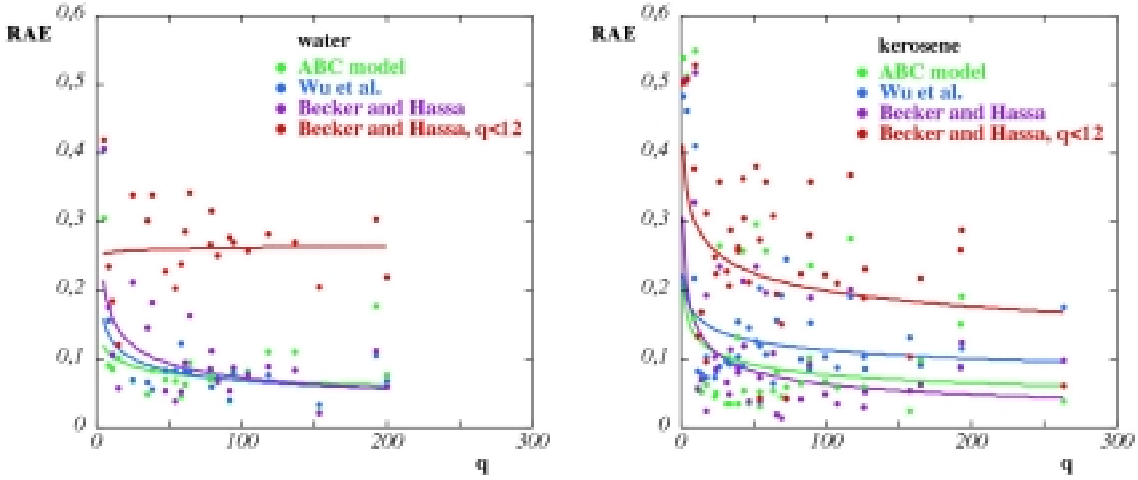

Fig. 3 Residual Average Error (RAE) for water (left column) and kerosene jets (right column), plotted as a function of the q *number (upper row) and* We_G *(lower row).*

CONCLUSIONS

The employ of a modeling approach based on a physically consistent description of the crossflow atomization mechanisms gave reasonably good results in terms of global behavior of the liquid jet, being the model flexible enough to fit well both water and kerosene jets, and so marking a line with the integral models available in literature. Nevertheless it must be noted that for both liquids the ABC model is still far from a satisfying description of the local jet trajectory. The inability to reproduce accurately the slope profile seems to be attributable mainly to the fact that the BLS model, chosen as an “entry level” approach to the atomization problem, substantially fails to capture the actual amount of liquid stripped away by the airflow. As a consequence the trajectories predicted by the ABC model show a weaker bending with respect to experiments. In addition this lack of fit could be partly due to the absence, in the model, of a description of the deformation of jet cross section, which probably undergoes local spinning and thinning because of both varicose and asymmetric instabilities enhanced by the shear. These phenomena could be responsible for the supposed reduction of the column resistance to drag even before the occurrence of the jet breakdown. Nevertheless a physical understanding of those mechanisms still misses, so that at the moment a mathematical model is forced to neglect them, unless taking into account by resorting to a lot of experimentally assessed parameters. Finally an indication is given that the friction factor is overestimated by the value suggested by Wu et al., mostly for water jets. The use of a large value of the friction factor was justified by the necessity to make the modeled jet bend enough to globally fit the behavior of the experimental trajectory, balancing the absence of an effective atomization model. The future adoption of an adequate model for aerodynamic stripping should at the same time improve the agreement with experiments both near the nozzle, where the value of c_{Dx} would be reduced, and far from the nozzle, where the mass removal due to the atomization seems to strongly affect the trajectory.

REFERENCES

1. Schetz J. A., Kush E. A. Jr., Joshi P. B.: *AIAA Journal*, **18**:774 (1980)
2. Chen T. H., Smith C. R., Schommer D. G.: *AIAA* Paper 93-0453 (1993)

3. Becker J., Hassa C.: *Atomization and Sprays*, **11:**49 (2002)
4. Becker J., Hassa C.: *ILASS-Europe.* Touluse, France, July 5-7, (1999)
5. Becker J., Hassa C.: *ICLASS 2000 Proceedings,* Pasadena, CA, USA, July, pp. 942-949 (2000)
6. Pilch M., Erdman C. A.: *Int. Journal of Multiphase Flow*, **13**:741 (1987)
7. Schetz J. A., Padhye A.: *AIAA Journal*, **15**:1385 (1977)
8. Wu P.-K., Kirkendall K. A., Fuller R. P., Nejad A. S.: *J. Prop. and Power*, **13:**64 (1997)
9. Ragucci R., Cavaliere A.: *ILASS-Europe.* Zaragoza, Spain, September, (2002)
10. O'Rourke P. J., Amsden A. A.: *AIAA Paper* 872089 (1987)
11. Rachner M., Becker J., Hassa C., Doerr T.: *Aerospace Science and Technology*, **6**:495 (2002)
12. Ranger A. A., Nicholls J. A.: *AIAA Journal*, **7**:285 (1969)
13. Madabhushi R. K.: *Atomization and Sprays*, **13:**413 (2003)
14. Reitz R. D.: *Atomisation and Spray Technology*, **3**:309 (1987)
15. Delplanque J. P., Siringano W. A.: *Atomization and Spray*, **4:**325 (1994)**.**
16. Tritton D. J.: *Physical Fluid Dynamics*, Oxford Science Publications (1988)
17. Bellofiore A., Cavaliere A., Della Valle V., Ragucci R.: *Joint Meeting of the Greek and Italian Sections of the Combustion Institute*, Corfu, Greece, June, n. 69 (2004)