Momentum Coherence Breakdown of Bending Atomizing Liquid Jet

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

1. Istituto di Ricerche sulla Combustione - C.N.R., Napoli, Italy
2. Dip. Ingegneria Chimica - Università degli Studi di Napoli Federico II, Italy

Modeling of liquid jets in air cross flow is a very complex problem due to the complex interactions occurring between the atomizing liquid jet and the surrounding air flow. In this paper two simplified models are presented and their results are compared with experimental data obtained in a high-pressure channel at different air and jet velocities. The first one is an integral model based on the momentum balance equations written on a finite thin slice of the liquid column jet. Effectiveness of this model is closely related to parameters tuning based on comparison with available experimental data. The second one is a numerical model based on the same simplified momentum balance equations but written on a single non-spherical particle with the task of simulating the jet. It takes into account transport equations for continuous phase, standard $\kappa-\varepsilon$ turbulent model, coupling between the phases and is solved by a commercially CFD code. Experimental data are obtained in a fully optical accessible tunnel in which the jet is injected perpendicularly to the air cross flow; a CCD digital camera collected shadowgraphs of the sprays at each light pulse generated by a low-pressure xenon flash lamp. The experimental trajectories have been compared with those obtained by using the described models to verify their agreement in dependence of the process parameters.

1. Introduction

The study of the interaction between a liquid jet and a gas cross flow 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 deformed by drag forces and pressure drop between windward and leeward side, and liquid fragments stripped from jet surface. 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 also undergoes breakup and breakdown phenomena. The first one leads to the loss of jet continuity and to the formation of ligaments and, eventually, of droplets. The lack of coherence of the jet results in a substantial breakdown of the jet due to the fact that liquid particles, formed in the jet break-up, are easily dragged by the airflow surrounding the spray. In addition, the droplets can undergo to a secondary breakup process producing even smaller droplets.
Experimental analyses [1, 2] have been interpreted by invoking the existence of two main atomization mechanisms in dependence of the value of the ratio of jet to airflow momenta, expressed by means of the "q" parameter $\left[q=\left(\rho_L v_L^2\right)/\left(\rho_G v_G^2\right)\right]$. The first mechanism is the column breakup: Here the jet breakup is due to the onset and growth of waves on the windward side of the jet. In this case the jet appears to be already bended by the airflow

before the occurrence of the breakup even if a significant change in the jet trajectory could be observed in correspondence of the rupture point. This mechanism is characteristic of lower q values when the airflow has enough energy to amplify jet instabilities before the alignment of the jet to airflow direction generates a quasi-coaxial condition where the effect of airflow induced stresses is less relevant. On the other hand, at higher q values the jet breakup occurs mainly through the surface breakup mechanism. In this case liquid ligaments, eventually leading to the formation of droplets, are detached from the jet surface due to the action of the airflow eroding the jet. Jet breakup can take place, in this case, when the jet becomes thin enough that the combined action of aerodynamic stresses and surface tension causes the rupture of the liquid column. The discriminating parameter in this case is the Weber number relative to the airflow ($We_G = \rho_G v^2 D / \sigma$, where v is the relative normal velocity at the jet interface that is usually assumed equal to the airflow velocity) [3]. At We_G values high enough the aerodynamic stresses become strong enough to induce the stripping of liquid from the liquid column. At lower We_G values the only effect of airflow is the amplification of jet instabilities and a moderate bending of the jet. It is clear from these considerations the relevance of the jet size and of the surface tension of the liquid.

This picture is simplistic due to the omissions of several effects occurring during jet injection. For instance, the jet atomization due to the liquid velocity and to the ambiance pressure (typical of high pressure jets injected in quiescent atmosphere) can play a relevant role in the phenomenology of jet breakup. In this case the parameters that seem to be relevant to the jet breakup are the Weber number relative to the liquid column ($We_L = \rho_L v^2 D / \sigma$, in this case v is the relative velocity of the liquid column with respect to the surrounding air which is usually assumed equal to the initial injection velocity) and the ratio of the air to liquid densities $M = \rho_G / \rho_L$ [3], even though a satisfactory description of the process is far from being attained.

In conclusion, 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 have to be taken in account are the viscosity, the convection inside liquid phase due to drag force and pressure drop, the energy transfer taking place at all turbulence scales and cavitation phenomena inside the nozzle.

This very complex phenomenology prevented from the implementation of satisfactory predictive models of the global jet behavior in cross flow. At present time many efforts are focused on the implementation of models able to predict global properties of jets in cross flow conditions (the most common is jet trajectory). These models can be classified, according to the suggestion of Demuren [4], in four classes:

- **Empirical models** rely essentially on the identification of the parameters of a predefined correlation by means of the comparison with experimental data. Prediction accuracy may depend on the closeness of the actual working condition to those used for the correlation. These represent simple methods to realize first order estimates and qualitative checks.
- **Integral models** are based on integral equation derived by considering the balance of forces and momentum changes over an elementary control volume of the jet or by integrating partial differential equations governing the jet flow. A set of ordinary differential equations is obtained which can be solved analytically or numerically. Empirical input is required to assign drag coefficient and other parameters. In many of the earlier models there was the assumption of constancy of jet momentum in the initial direction, while the jet was bent over by a prescribed drag force. None of these models could predict correctly the jet trajectory over suitable ranges of v_L and v_G values. More refined integral models consider effects of both the drag force and the entrainment of cross flow on the jet. These models have been

criticized for the need to assume the shape of the jet cross section and profile functions, some of which may not be realistic for the whole evolution of the jet in cross flow. However, it appears that in spite of the apparent oversimplification, integral models can be made to perform well in same cases with proper calibration.
- **Perturbation models** required that in the jet cross flow problem a small parameter is defined; in the flow of strong jets in a weak cross flow, $\lambda = v_G / v_L$ can be used as the small parameter. These models approximate the near field as two regions of irrotational flow, the jet flow and the external cross flow, separated by a vortex sheet. Perturbation models do not require much empirical input, but they are mostly restricted to near field or far field where small parameters required for expansions can be defined. They have limited practical utility but are useful tools for investigation of flow physics.
- **Numerical models** could represent a practical predictive tools over a wide range of jets in cross flow application and require the least assumptions and empirical input. The analysis starts from the general conservation laws stated in partial differential equation form, which are the Navier-Stokes equations for the velocity field, and corresponding energy or species equations for the temperature or concentration fields, respectively. These equations, which describe unsteady, three-dimensional flow cannot be solved directly in practical applications for turbulent flows. In incompressible fluid flow, time-averaged forms, and in compressible fluid flow, density weigthed, time-averaged forms of the equations can be solved but the process of time-averaging introduces a closure problem, so turbulence models or Reynolds stress models are required for the resolution of the system equations. The use of numerical models is, at moment, limited in the practical use by the uncertainties in the choice of turbulence and Reynolds stress models, by the difficulties in assessing the initial and boundary conditions and by the high computational loads connected to the very fine meshes required.

The aim of the work is to clarify, by means of a tight comparison of experimental evidences and results of models, the relevance of the different physical effects acting during the jet atomization and bending. This approach allow not only to set up more general and reliable integral models of the jet behavior, that can relax their typical limitation to the class of experimental conditions in which they are validated and/or tuned, but also to evidence the physical phenomena and take in account the experimentally observed phenomenology in a more rational way.
In this paper an integral and a simplified numerical model of the jet are presented and their results are compared with the experimental results obtained in a high pressure-channel at different air and jet velocity. The statistical elaboration of experimental data allows for the determination of jet breakdown characteristics, e.g. jet breakup distance, by the identification of the spatial position where a loss of coherence of the jet, in consequence of the liquid column breakup, occurs. This measurement can be used to refine the integral models giving them both greater accuracy and a way to introduce other effects (like those connected to the jet atomization due to the aerodynamic interaction) in a synthetic way.

2. Experimental set-up

The experimental set-up [5] consists of a fully optically accessible tunnel, with a square cross section of 25 X 25 mm, in which the jet is injected perpendicularly to the air cross flow. Temperature, pressure and velocity of the air-flow in the tunnel can be varied independently each other in a wide range of operating conditions. The injecting system is a plane orifice one with a nozzle hole of 0.3 or 0.5 mm diameter. The liquid is supplied to the nozzle by means of a nitrogen-pressurized vessel allowing for a precise injection pressure setting.

The optical set-up (Fig. 1) is a simple shadowgraphic scheme using a low-pressure xenon flash lamp as light source. A 1 M-pixel CCD digital camera collected shadowgraphs of the spray at each light pulse. In each test condition forty 10 bits images (1024 different tonalities of grey) of the spray were collected. A set of acquisition and elaboration procedures, written in LabVIEW programming environment and based on a standard library of image manipulation algorithms (NI-IMAQ), allowed for the characterization of statistical and morphological features of the spray. A background image, obtained by a exposing the camera in absence of the spray, was subtracted in order to reduce electronic noise. From the obtained images an average image was computed in order to study the bending and penetration of the liquid jet. Using an image binarization routine it was possible to outline the boundaries of single and averaged images in each condition in order to demarcate the spray boundary. The variation both in time and space of image boundaries were used to characterize the jet stability in statistical sense.

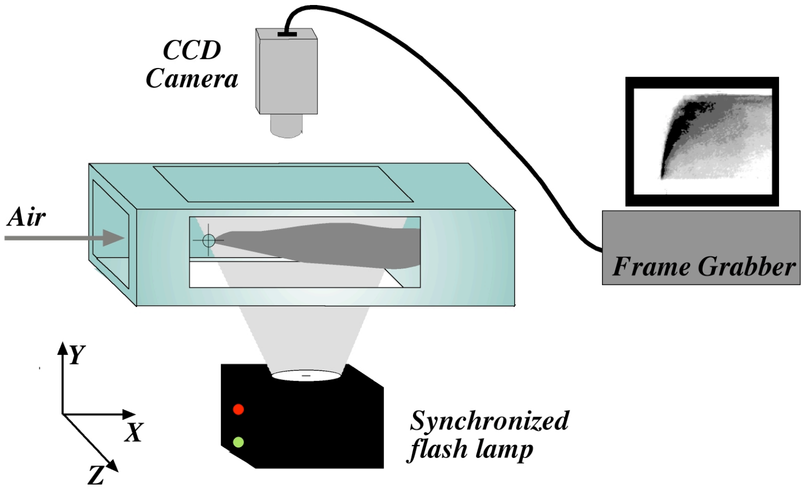

Figure 1. Diagnostics schematics.

3. Jet Breakup Estimation

In order to clarify a possible correlation between jet stability and injection conditions a square average shifting indicator, *s*, has been defined as [6]:

$$s = \frac{1}{N}\sqrt{\sum (x_n - x_m)^2} \qquad (1)$$

Where x_n is the *x* coordinate of external boundary of n^{th} image, x_m is the *x* coordinate of average image boundary and *N* is the total number of images. A typical set of profiles is reported in Fig. 2.

The graph shows, for low values of the ordinate, a slow and regular increase of the square average shifting error. This phenomenon is caused by the progress of the atomization process. In some conditions a square mean shifting error sudden increase can be observed at a distance from the nozzle greater than 15 mm. This increase could be tentatively attributed to the breakup of the liquid column. In this case the error increase can be explained by the different behavior of a cloud of droplets with respect to a liquid column. Although this is not a rigorous evaluation of the jet break-up position it appears to be at least a good first approximation estimation method of this position [6].

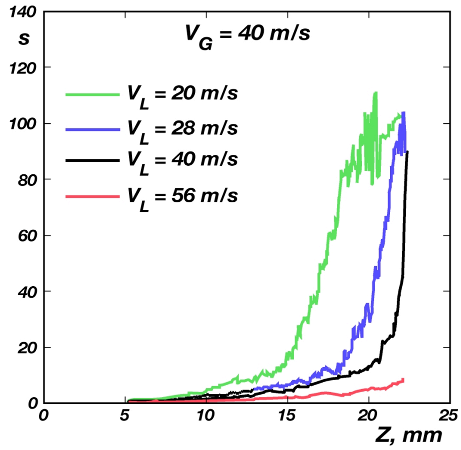

Figure 2. Square average shifting indicator versus z.

4. Numerical modeling of a jet in cross flow

4.1 Model I: Integral Model of the Jet

The first model, pertaining to the integral models class, used for the jet description resembles the approach largely followed in literature [1, 6] based on the numerical resolution of the momentum balance equations and the subsequent tuning of the model parameters by using experimental data available. In our case a momentum balance is written on a finite, thin slice of the liquid column delimited by two layers perpendicular to the initial velocity of the jet. In this model the terms taken in account are only jet inertia and drag forces. A Cartesian system is chosen with the origin in the geometrical center of the nozzle outlet, under the hypothesis of two-dimensional problem and taking *x* and *z* axes respectively oriented along the gas flow and initial velocity of the jet. Along *x* the momentum balance on the liquid volume *V* is

$$V\rho_L \frac{dv_{Lx}}{dt} = C_D \cdot \frac{1}{2}\rho_G \left|\Delta v_x\right|^2 \cdot S_x \tag{2}$$

where ρ_L and ρ_G are the densities of the two phases, assumed to be constant (the hypothesis is reasonable for the gas too, because flow is under subsonic conditions), v_{Lx} is the x component of liquid velocity, C_D is the drag coefficient, that is multiplied by the kinetic energy of gas (calculated on the gauge velocity Δv_x, responsible of the shear) and by the cross section S_x of the liquid element, i.e. the area perpendicular to the gas flow.

In such a formulation of the drag force, the friction factor C_D collects all the ignorance on shear effects, and allows to express this force in terms of quite well known quantities such as relative kinetic energy and the cross section of the object undergoing the shear action. In the present problem of a liquid jet atomized under cross flow conditions, the drag coefficient appears to be particularly stiff to evaluate or model. First of all, the efficiency of the energy transfer between phases is affected by the complexity of the interface, hard to be described in terms of elementary geometry. Second, the drag coefficient is strongly dependent from both global and local flow field, because of the multi-scale mechanism of energy transfer (turbulence).

The most rigorous approach would require the use of CFD codes as explained in the intriduction. On the opposite, the analytical approach needs strong approximations, in particular the assumption of a C_D constant for the whole liquid jet. An intermediate approach

could be represented by the choice of taking in account the dependence from the Reynolds number.
In the present model, for the evaluation of the drag coefficient in the *x* direction, the liquid jet is assimilated to an unlimited, unbending cylinder transversally invested by a gas flow. Under these hypotheses, even though Re varies significantly along the jet, due to the reduction of both relative velocity and jet diameter, nevertheless it remains in the range of high values, where C_{Dx} is practically constant with Re. Furthermore the cross section has to be referred to a cylindrical geometry. In order to take in account the lack of shape ideality, due to column deformation and surface roughness induced by the aerodynamical stripping of ligaments and drops, a correction factor ψ is introduced.
Relative velocity Δv_x is responsible of the shear in the *x* direction. Integral models [1, 6] usually assume that along *x* liquid velocity is negligible with respect to gas velocity. As a consequence, the absolute gas velocity controls drag force. In the numerical model here proposed, this strong simplification has been removed, so that the balance (2) takes the form

$$V\rho_L \frac{dv_{Lx}}{dt} = C_{Dx} \cdot \frac{1}{2}\rho_G \left|v_{Gx} - v_{Lx}\right|^2 \cdot S_{cyl,x}\psi \tag{3}$$

Assuming that the cylindrical element's height is δz:

$$\frac{S_{cyl,x}}{V} = \frac{D\delta z}{\frac{\pi D^2}{4}\delta z} = \frac{4}{\pi D} \tag{4}$$

and so the balance formulation becomes:

$$\frac{dv_{Lx}}{dt} = \frac{2}{\pi} \cdot \frac{\rho_G}{\rho_L} \cdot \left|v_{Gx} - v_{Lx}\right|^2 \cdot C_{Dx} \cdot \psi \frac{1}{D} \tag{5}$$

An analogous equation can be written for the *z* component of liquid velocity. The *z* component of the gas velocity is neglected. Briefly, this model assumes that gas flow field is not perturbed by the presence of the liquid jet. The momentum balance is

$$V\rho_L \frac{dv_{Lz}}{dt} = -C_{Dz} \cdot \frac{1}{2}\rho_G {v_{Lz}}^2 \cdot S_z \tag{6}$$

In this case the surface exposed to the drag force is only the lateral area of the liquid element, cylindrically shaped in the ideal condition. Because of the stiffness of a theoretical evaluation of the drag coefficient C_{Dz}, it appears to be useless the introduction of a correction factor in this equation too: ignorance, about shape and energy transfer effectiveness, is all put in the friction factor. Equation (6) is then re-formulated as

$$\frac{dv_{Lz}}{dt} = -2\frac{\rho_G}{\rho_L} {v_{Lz}}^2 \cdot C_{Dz} \cdot \frac{1}{D} \tag{7}$$

Integral modes, like the one proposed by Wu, are affected by the serious limit of neglecting the jet diameter reduction due to the atomization process. The most important effect of this simplification resides in the evaluation of the jet inertia, which appears to be overestimated with respect to the experimental results. In other words, in these models the bending of the jet is less strong than it should be. This could be one of the reason for the greater effectiveness of the model by Wu [1] in predicting jet trajectories in the spatial region closer to the nozzle outlet than at higher distance. As matter of fact, the approximations introduced by Wu (required to allow analytical solving of the momentum equations) give good results as long as the jet can be assimilated to an unbending cylinder perpendicular to the gas flow, as it is the case close to the nozzle outlet.

A first attempt to solve this problem is to assume a linear reduction of jet diameter along the trajectory [6]:

$$D = D_0\left(1 - z/z_{jb}\right) \tag{8}$$

where D_o is the nozzle outlet diameter and z_{jb} is the jet breakup length. This last parameter has been experimentally assessed, as seen in the previous section, when jet images show an evident column breakup. Otherwise, i.e. in case of wall impingement, z_{jb} have been extrapolated for continuity from similar conditions with lower q values.

Equations (4), (5) and (7) have been solved by implementing, in Labview programming environment, the Euler method for the integration of ordinary differential equations. Integration step has been chosen short enough to ensure numerical stability. The value of C_{Dx} was estimated by semi-empirical correlations, available in literature, for a perfectly cylindrical shape, the only two parameters still to evaluate are ψ and C_{Dz}. The first one represents the deviation of the momentum transfer efficiency from ideal values of C_{Dx} and S_x. One of the points of interest of this model is to verify whether this deviation is really small. Furthermore preliminary studies on model's results showed that the estimated drag is a little lower than measured, and so a correction factor equal to 1.20 has been assigned. For the z component of liquid velocity, integral models usually assume that kinetic energy reduction along z is negligible, and this strong approximation gave quite good results. This fact means that in z direction the momentum transfer does not perform well, and so a little value of C_{Dz} appears to be a reasonable assessment. In particular a good fitting of experimental data has been reached with a drag coefficient C_{Dz} assumed to be equal to 0.25.

4.2 Model II: Simplified Numerical Modeling of the Jet

A second model has been implemented that follows an intermediate approach between the integral and numerical models. In this case the jet trajectory is determined by using a commercial fluid-dynamic problem solver (Fluent), retaining all the complexity of the airflow and of the jet air intetractions, but schematizing the jet as a single nonspherical particle identified by a shape factor ϕ [7]:

$$\phi = \frac{s}{S} \tag{9}$$

where: s is the surface area of a sphere having the same volume of the particle and S is the actual surface area of the particle

The ϕ choice fixes, in conjunction with Reynolds number, the drag coefficient C_D that represents the model empirical input [3]. This approach comes out from theoretical considerations [8] and experimental investigations [9-11] and concerns the deformation of the cross-section of a jet injected into a cross flow (from circular to 'kidney' shape) due to the pressure gradients induced by the separation of the turbulent boundary layer. This strong distortion determines an increase of the drag coefficient with respect to its value in the solid cylinder hypothesis. The choice of the ϕ value (coupled with Reynolds number, value relative to the actual working conditions) give the opportunity of calibrate the C_D value with respect to the experimental data.

To obtain the particle trajectory in Fluent, we have used the Discrete Phase Model that, in addition to solving transport equation for the continuous phase, allows for simulating a discrete second phase in a Lagrangian reference frame. The force balance equates the particle inertia with the drag force acting on the particle and can be written in our conditions (for the x direction) as:

$$\frac{dv_P}{dt} = \frac{3}{4} C_D \rho_G \left| v_P - v_G \right|^2 \frac{1}{\rho_P D_P} \tag{10}$$

where v_P is the particle velocity, ρ_P is the particle density, v_G is the air velocity, ρ_G is the air density and D_P is the diameter of a sphere of volume equal to that of the particle.

The standard $\kappa - \varepsilon$ model have been used as turbulent model and the coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase has been taken in account in the computation.

In this paper results will be presented relative to ϕ values (i.e. C_D values) characteristics of a solid cylinder. Results relative to simulations performed using C_D values greater than the one characteristic of a solid cylinder have been performed in the same conditions and are not reported here due to lack of room. However it has been found, using higher values of the C_D, a good agreement with experimental data in particular when the q values are greater.

5. Results and Discussion

In several conditions, characterized by increasing q values, the experimental characterization of the jet trajectory has been obtained by identifying on each average image a set of (x, z) coordinates corresponding to the maximum intensity locations along the jet trajectories. These sets of points represents the preferential trajectories followed by the jet in the different experimental conditions considered. The experimental trajectories have been compared with those obtained by using the integral model (Model I) and the simplified numerical one (Model II).

In figure 3 the results of the models are presented along with the experimentally determined trajectories for 9 values of the air and liquid injection velocities. The air pressure and temperature were kept constant at 1.3 MPa and and 300 K. The liquid used was distilled water and the nozzle diameter was 0.5 mm for all the reported tests. In the figure the jet trajectories are reported (as red dots) over the average image computed in the corresponding condition in order to allow for a direct comparison of the model results with the visual observation of jet behavior.

Figures represent jet trajectory for both x and z starting from zero (nozzle outlet) up to 25 mm. The condition $z = 25\ mm$ corresponds to the opposite wall of the chamber, while at $x = 25\ mm$ the atomization process is fully developed and liquid jet does not exist longer.

When q is lower than about *100*, jet undergoes a strong bending, as it can be seen most of all in the first two figures. In these cases ($v_{air} = 40\ m/s,\ v_{jet} < 30\ m/s$) the jet trajectory appears to be characterized by an evident knee at its half height. The windward outline shows the rising of waves on jet surface, instabilities which initially promote the stripping of liquid ligaments, and then, for higher wave amplitude, take to the loss of column integrity, i.e. jet breakdown. After that, there should be only liquid fragments quickly atomized in a dense plume of droplets clearly visible in the images. These images can be identified as typical examples of column breakup atomizing jets, where the liquid column is bended significantly before its collapse with a subsequent strong variation of jet plume direction. Another possible explanation could be searched by supposing the formation of a low-pressure zone downstream the jet (due to the *bow* shape of the jet transversal section), so that the sudden increase of pressure drop generates the knee.

Both models fit well the experimental points, even though they cannot depict the typical knee. The first two conditions are the most favorable for the definition of a z_{jb} by evaluating the square average shifting indicator, and so the integral model works well, maybe because it reaches a good estimation of the jet diameter decrease. The simplified numerical model overestimates the jet bending probably due to the overestimation of the drag coefficient. For q values equal to 40, 52, 62 and 78, the trajectory simulated by the integral model is less bended than simplified numerical model, which better fits the real jet behavior probably due to the more realistic simulation of a complex interaction between the liquid jet and the airflow adopted in this case. In particular the two conditions at q=52 and q=78 do not differ significantly in the jet bending although the jet plume appear to be greater in the condition at higher q. This can be interpreted noting that while the We_G number is almost the same in the two cases on the other hand the We_L in the second case is twice the value found in the first case. Accordingly, a higher jet atomization should be expected in the second case giving rise to a more evident droplet plume dragged by the airflow in the leeward zone. Furthermore in this case the spray shadow appears to be more regular and the typical corrugation of windward interface is not observable in this case.

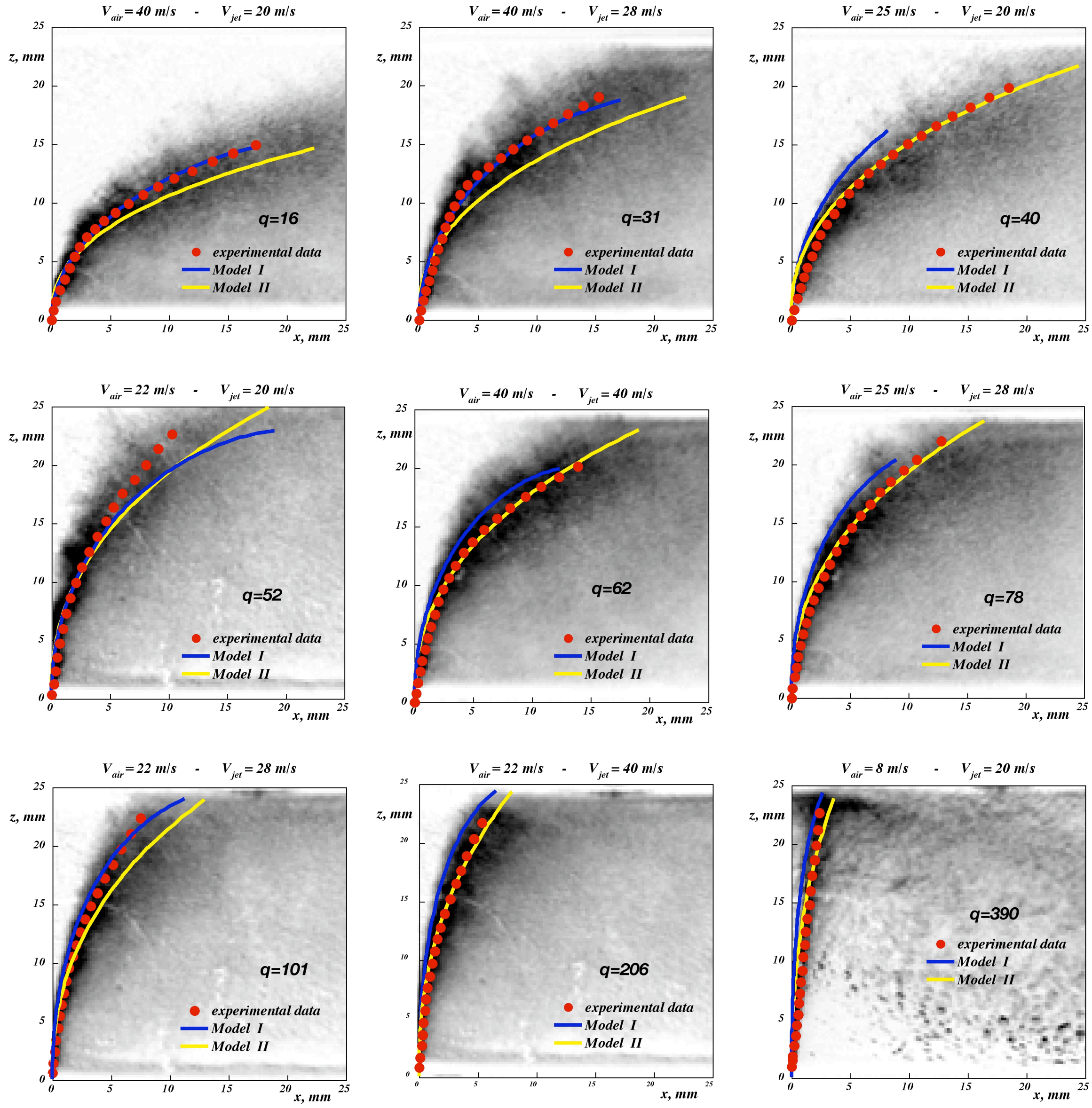

Figure 3. Comparison of the result of jet trajectory models with respect to the experimental data.

At q values above 100 the bending force induced by the aerodynamic action cannot prevent the jet impact on the opposed wall. In these conditions the kinetic energy of the jet is so strong, with respect to air, that liquid column moves straightaway towards wall with a less evident bending becoming practically negligible for values of q higher than 200. In this condition air stripping appears to be progressively less effective. It must be noted that the condition at q=206 is characterized by a We_L which is four times the We_L in the condition at q=390 suggesting a stronger atomization of the jet in the first case. Accordingly the spray plume in the leeward region is more evident in the former case while in the second case a large amount of liquid drops is produced by the impact against the wall of a conspicuous residual jet. It should be stressed that in these cases both the models work reasonably well. The trajectories forecasted by the integral model is less straight than experimental results, probably due to the fact that the reduction of jet diameter is moderate when cross flow effects are feeble making the assumption of liquid column consumption according to a linear law too strong. The second model appears to be more effective probably due to the assumption of jet behavior as a rigid cylinder.

In general results indicate that the strong hypothesis of little deviation from a rigid cylinder transversely invested by an air flow, as well as the simplification of assuming a constant drag coefficient, do not compromise the validity of the integral model. On the other hand, it has to be underscored that for all the conditions liquid column appears to be strongly bended already at the very beginning of its trajectory, maybe due to a high efficiency of the shear when gas flow collides with the emerging liquid jet. This fact means, and results of simulations seem to confirm, that at least near the nozzle outlet, the “real” drag coefficient (i.e. the product $C_{Dx}\psi$) is significantly higher than the assessed value, which instead fits well the experimental results starting at certain distance from the nozzle outlet.

6. Conclusions

The first conclusion pertains the general good description given by the proposed models of the jet trajectories. However it has to be underlined that the integral model (model I) is much more effective when the determination of the jet breakdown allows for having a reliable estimation of the liquid column rupture. This confirms the general problem of integral models in requiring an experimental validation of controlling parameters to give realistic results. On the other hand the model seems to give reasonable good results also in conditions where the estimation of jet breakup distance is less reliable or is prevented by the impact of the jet on the wall before the occurrence of jet breakup. A reason of this, somewhat unexpected, effectiveness of the model resides in the assumption of a reducing jet diameter obtained by imposing a consumption law. Correlating the consumption rate to the ongoing of atomization process could allow for refining the very simple consumption law, used here. The results suggest that the evaluation of the We_L and of the ratio of liquid to air densities can give indications of the effectiveness of atomization process and, as a consequence, of a more realistic consumption law. To this aim a more systematic analysis of the influence of these parameters is needed.

The second model, which is less dependent on the experimental determination of jet characteristics, was very effective in describing the jet trajectories, apart from some cases where the assumptions of the model were significantly different from the real processes. To this aim a more realistic determination of the drag coefficient (and as a consequence of the ϕ) could give better results. In this case too it appears important the assessment of a relationship between airflow and jet injection parameters and the drag coefficient, which does not only depend on the value of Reynolds number, but must also take in account the jet deformation and the occurrence of the aerodynamically induced atomization process.

From a practical point of view the relevance of the jet atomization by liquid stripping from jet surface (either as a direct consequence of stresses induced by the air cross flow or due to jet velocity induced atomization with a subsequent liquid drag acted by airflow) appears to be relevant for system operated at increasingly high pressures like gas turbines. For these systems the prevention of jet impact on the wall is important in order to prevent the fuel deposition on the premixing ducts but it is also important an as much as possible uniform distribution of the liquid in the whole duct section. To this aim the onset of a surface breakup mechanism appears much more efficient than a column breakup one. It appears in this case that the parameters to be controlled are, apart from the q parameter, also the parameters governing the atomization process that are the Weber numbers relative to the gas flow and to the liquid jet. In these parameters are also taken in account the dimension of the jet and the surface tension of the liquid. This suggests that suitable choices of the nozzle size (i.e. of the jet initial diameter) and of the initial jet orientation can allow for the determination of the most efficient liquid dispersion in the duct.

7. Aknowledgement

Part of this work has been done under the contract N° : G4RD-CT-2002-00644 in the project n°GRD1-2001-40198 with tytle Modelling of UnSteady Combustion in Low Emission Systems (MUSCLES)

8. References

[1] Wu P.K., Kirkendall K.A.: "Breakup Processes of Liquid Jets in Subsonic Crossflows" *Journal of Propulsion and Power*, **13**, n.1, pp. 64-73, 1997

[2] Becker J., Hassa C.: Breakup and Atomization of a Kerosene Jet in Crossflow at Elevated Pressure, *Atomization and Sprays*, **12**, no 1-3, pp49-67, 2002

[3] Bayvel L., Orzechowski Z.: *Liquid Atomization*, Taylor and Francis, Washington DC, 1993

[4] Demuren A.O.: Modeling jets in cross flow, *ICASE Report 94-71, NASA Contract No. NAS1-19480*, 1994

[5] Cavaliere A., Ragucci R., Noviello C.: "Bending and Break-up of a Liquid Jet in a High Pressure Air Flow" *The Second Mediterranean Combustion Symposium* Sharm El-Sheikh, Egypt, January 6-11, Paper #MCS02-086-III, 2002

[6] Ragucci R., Cavaliere A.: "Identification of Cross-flow Liquid-jet Structures by means of Statistical Image Evaluation", *18th European Annual Conference on liquid Atomization and Spray Systems - ILASS-Europe 2002*, Zaragoza, 9 –11 September 2002

[7] Haider A., Levenspiel O.: Drag coefficient and terminal velocity of spherical and nonspherical particles, *Powder Technology*, **58**, pp. 63-70, 1989

[8] Abramovich G.N.: The Theory of Turbulent Jets, MIT Press, Cambridge, MA, 1963

[9] L.L. Yuan, R. L. Street: Trajectory and entrainment of a round jet in crossflow, *Physics of Fluids*, **10**, N°9, pp. 2323-2335, 1998

[10] Mazallon J., Dai Z., Faeth G.M.: Primary breakup of nonturbolent round liquid jets in gas crossflow, *Atomization and Sprays*, **9**, pp. 291 - 311, 1999.

[11] Ragucci R., Cavaliere A., D'Amico R.: "Atomization of a Liquid Jet in Gas-Turbine Configuration", *Eighth International Conference on Liquid Atomization and Spray System*, Pasadena, CA, USA, July 2000.