# Study of the Influence of Initial Droplet Size Distribution on a Liquid Jet in Crossflow Model Performance

P. Di Martino[1*], G. Sorrentino[2], R. Ragucci[3]
[1] Avio Group, Pomigliano d'Arco, Naples – ITALY
[2] Dipartimento di Ingegneria Chimica - Università Federico II, Naples – ITALY
[3] Istituto di Ricerche sulla Combustione - C.N.R., Naples – ITALY

**Abstract**

New technologies in the gas-turbine engine field include LPP (Lean Premixed Prevaporization) and LDI (Lean Direct Injection), in order to reduce NOx emissions, and raise a growing interest in the injection of liquid fuel transversely to high-density airstream and in the dependence of engine performance on the efficiency of dispersion and atomization of liquid in gas phase.

Multipoint injection strategies 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 atomizers (essentially a set of plain nozzles) and of the fuel feeding system.

They can be easily optimized both on the ground of a time-modulated strategy in dependence of the working condition and with a proper choice of injection points position and orientation with respect to the airflow. In this framework airflow pattern, and as a consequence, system geometry has to be design together with injection system to satisfy the general requirements of the combustion process.

There is a rather large corpus of knowledge available in literature on jet injection in cross-flow and it appears that the status of knowledge is at a satisfying level at least for the phenomenological description of the process [1-3]. 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 reliable modeling tools represents a key factor in the efficient and economical development of new devices allowing the determination of the most promising configurations suitable for further development and prototyping. In this respect, research in the field suffers for the lack of a significant mass of experimental data available for a thorough validation of model results.

In this paper a model based on a hybrid eulerian-lagrangian description of the liquid jet breakup and of the cloud of droplet behavior is discussed and validated against experimental data obtained by means of PIV measurements collected in some relevant conditions in a squared test channel under high pressure and high temperature conditions.

The used model was firstly introduced at a previous ILASS conference [4] and it has been thereafter further developed by modifying the droplet injection model and the interaction of the liquid jet with the surrounding airflow [5].

In essence, in the model, 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 to the canonical orthogonal intersection between the liquid jet and the airflow and that the air velocity profile can be arbitrarily assigned making the model more suitable for use in practical devices modeling.

Several models are available in literature to describe the fragmentation of liquid drops and jets. A modified version of the Boundary Layer Stripping (BLS) model adapted for the cylindrical geometry was originally used to model liquid mass removal from the jet. Other possible liquid removal mechanism have been evaluated on the ground of experimental evidences and theoretical considerations.

A fundamental problem 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 aerodynamic force. The Taylor-Analogy-Breakup (TAB) model is largely used in the modeling of secondary breakup of spherical droplets because easily applicable. It has been suggested that the deformation of the liquid jet cross-section before the breakup has significant analogies with that of a spherical drop. For these reason a TAB sub-model, modified to account for the cylindrical geometry of the elementary volume of the jet, has been used to model jet cross section deformation. Determination of the aerodynamic forces acting on the jet requires the assumption of a suitable value of the drag coefficient.

The sub-model for the liquid jet trajectory has been largely proved to be effective in reproducing the jet trajectories [6] in a very large set of experimental cases built up in several experimental campaign made by the same research group in high pressure and high temperature conditions significant to the gas-turbine engines.

* Corresponding author: ragucci@unina.it

Moreover, this model, being not limited to the canonical case of nearly uniform air velocity profile and orthogonal intersection of the liquid jet and airflow, could be implemented in a CFD package to model the interaction of a realistic air-flow with the jet.

The liquid jet model was then implemented in a CFD code to account for the mutual interaction of the liquid column with t he airflow. The droplet detached from the liquid jet have been tracked by means of a Lagragian tracking package in order to derive a general description of the spray plume evolution.

CFD results have been 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. In the paper are reported the velocity maps and the velocity profiles at selected positions to compare with the model results.

While the general agreement between the numerical results and the experimental evidences apear to be quite satisfactory a sensitivity analysis has been carried out by varying the initial droplet diameter and the droplet size distribution.

A first example of the results is given in figure 1, where the velocity magnitude maps (with superimposed velocity arrow field) both from experimental measurements and numerical simulation are reported for one of the considered model variations. The general appearance of the spray plume is satisfactorily reconstructed by the model. A more close comparison have been made by comparing velocity components profiles at selected distances from the injection point. The comparison showed that downstream 10 mm from he injection pont the velocities are generally well reproduced while there a dependence oninitial velocities and size of the droplets could still be determined. This poses the problem an accurate choice of the primary break-up mechanism selection in order to give to the model the required robustness and effectiveness to be used in design of practical premixing devices.

**Acknowledgement**

This work has been partly supported by the European Community in the framework of the TLC contract, Contract N°AST4-CT-2005-012326. Authors wish to thank Antonio Picarelli, Alessandro Bellofiore and Vincenzo D'Antuono for their contribution in experimental work and Antonio Cavaliere for his invaluable advising.

**References**

[1] R. Ragucci, A. Bellofiore, A. Cavaliere, Proceedings of the Combustion Institute 31, (2) (2007) 2231-2238.
[2] R. Ragucci, A. Bellofiore, A. Cavaliere, Atomization and Sprays 17, (1) (2007) 47-70.
[3] J. Becker, C. Hassa, Atomization and Sprays 12, (1-3) (2002) 49-68.
[4] A. Bellofiore, P. Di Martino, G. Lanzuolo, R. Ragucci, 22nd Annual Conference on Liquid Atomization and Spray Systems (ILASS-Europe), (2008) paper 136.
[5] P. Di Martino, G. Sorrentino, R. Ragucci , Paper submitted to the 33rd International Symposium on Combustion, Beijing August 1-6 (2010).
[6] A. Bellofiore, A. Cavaliere, R. Ragucci, , 22nd Annual Conference on Liquid Atomization and Spray Systems (ILASS-Europe), (2008) paper 137

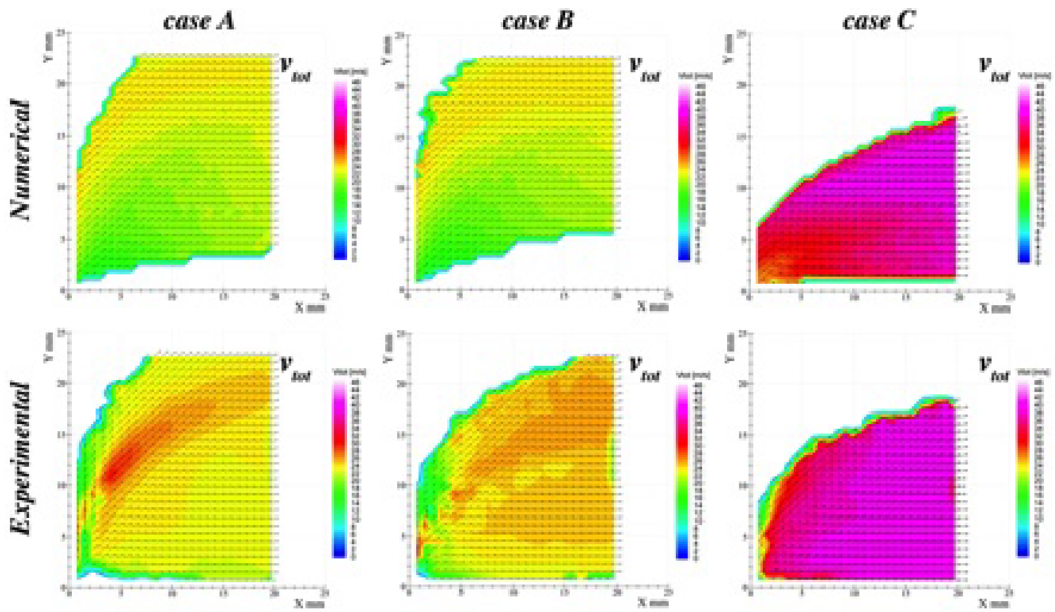


**Figure 1.** Comparison of the total velocity patterns computed using the model with the experimental patterns for the three test cases considered.