DNS-like simulations of atomization in the nozzle near field in plain-orifice atomizers

R. Payri, F.J. Salvador*, M. Crialesi
CMT-Motores Térmicos, Universitat Politècnica de València, Spain
*Corresponding author: fsalvado@mot.upv.es

Introduction

The understanding of the atomization process in energy generation is of fundamental relevance to achieve a cleaner combustion able to reduce the environmental impact. However, such an understanding is still not available due to the complex nature of this kind of multiphase flow. Multiphase DNS simulations are one of the most useful tools to obtain flow characteristics that can be effectively used to understand the flow physics and to interpret available experimental data.

As already proved in previous works [1], in this kind of simulations, the boundary inlet conditions for the spray are of great importance in order to accurately model the atomization process. In fact, it has been proved that turbulence is one of the main mechanisms that enhance and accelerates atomization process [2].

The present work aims to study the behavior of internal flow and effectively link it to the external flow. For the internal flow simulation, a Large Eddy Simulation approach has been used to model the turbulence structures inside a turbulent pipe (Re=5050). The open source code OpenFOAM has been chosen as a simulation environment, while an incompressible Wall Adaptive Local Eddy viscosity model has been selected to model the subgrid turbulence behavior. The simulation domain consists of a cylindrical pipe of L/D= 8, with a diameter of 90 μm.

Material and methods

For the external flow modelling, Direct Numerical Simulation (DNS) has been used due to its capability of providing a high amount of data on both space and time, while modelling all scales of motion in the flow. In this work the *one-fluid* method described in [3] and implemented in the PARIS-Simulator is used.

The main aim of this work is to effectively and accurately correlate the turbulence properties extracted from the analysis of the internal flow to the spray simulation though an algorithm for boundary condition generation. The algorithm used in this work is a *Digital Filter Based Method* from [4], which allows to control the size of the turbulent structures, their location and their temporal distribution, while maintaining the random behavior typical of the turbulent flows. This method calculates punctually the velocity turbulent component as:

$$u' = \sum_{n=-N}^{N} b_n r_{m+n} \qquad (1)$$

Where b_n are the filter coefficients, r_{m+n} are the zero-mean random data series components, N represents the filter support length and m indicates the grid point.

In order to determine the filter coefficients, the assumption of a fully developed homogeneous turbulent field is made in [4], consequently the autocorrelation from [5] for the u' is used as:

$$R_{u'u'}(d, 0, 0) = exp\left(\frac{\pi d^2}{4L^2}\right) \qquad (2)$$

Where L is the prescribed integral scale and d is the distance vector. Through the correction of these coefficients the temporal and spatial distribution of the turbulence can be adjusted, acting on the integral length scale, adequately interpreted for the time.

Finally, the velocity profile, as well as the radial turbulence distribution, needs to be implemented in the code. The velocity profile can be implemented once the linear wall correlation and the log-law wall respectively have been verified for the inner nozzle flow upstream of the spray simulation:

$$y^+ = y^+ \; ; \; u^+ = \frac{1}{\kappa} ln\, y^+ + B \tag{3}$$

Where κ is the von Karman constant and B is a fitting constant.

In order to determine the turbulence radial distribution, the coefficients of the diagonal form of the Reynolds stress tensor $a_{ii}(r)$ must be determined as function of the radius, so that the determination of the velocity can be achieved through the equation below:

$$u(r,t) = U \cdot f(r) + u'(t) \cdot a(r) \cdot I \tag{4}$$

Where $f(r)$ is a correlation derived from the solution of equation (3), by determining B and y^+ through the LES analysis and I is the turbulence intensity. The velocity profiles, as well as the turbulence distribution and maximum intensity can be obtained from the statistical analysis of the internal flow and validated against experimental results [6] and DNS results [7].

Results and Discussion

In Figure 1 the external spray (left) and vorticity (right) are depicted for two different cases. In the upper part, the case without considering turbulence inside the nozzle (flat profile without any velocity fluctuation) is represented. In the lower part, the case with a turbulence intensity of 5% and integral Length scale of *L*=0.1 *D* is shown.

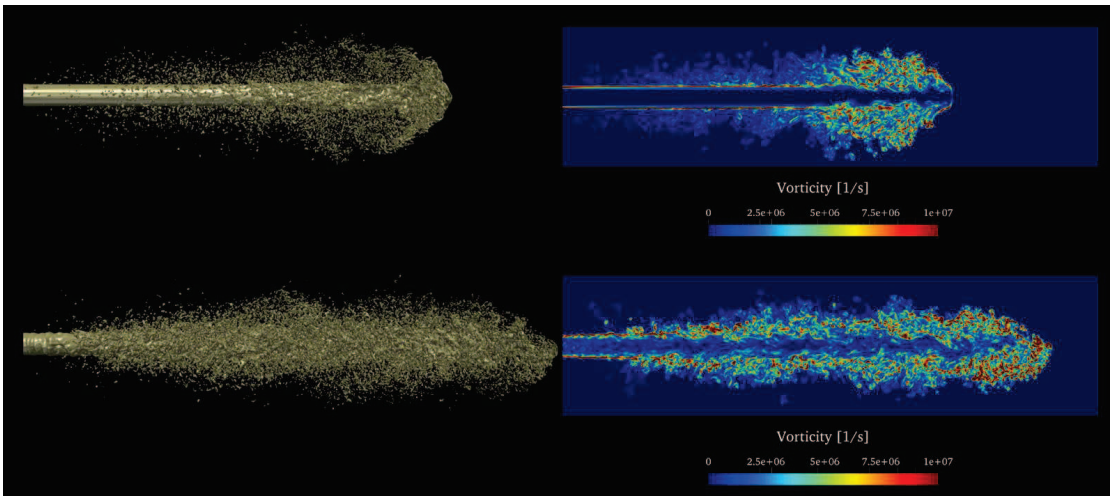

Figure 1. External appearance of the atomization process and vorticity for cases without turbulence inside the nozzle (upper part) and considering turbulence inside the nozzle (lower part).

As it can be noted, the atomization in the case where a flat velocity profile is used and turbulence is not considered (upper part of Figure 1), the atomization is very poor. In this case, it can be seen a long non-perturbed external length characterized by the presence of small quantity of droplets and atomization is produced mainly in the front of the spray, near of the spray tip characterized by the typical mushroom shape where, as can be seen in the upper part (right) of the Figure 1 vorticity is higher. On the other hand, in the case where a real velocity profile provided by LES simulation of the nozzle flow and so, considering turbulence, depicted in the lower part of that Figure, atomization starts faster and, therefore, closer to the nozzle exit. It can be noted that the number of droplets dramatically increases. In the right-hand picture, the vorticity field is depicted where it can be highlighted that maximum values are not only located in the front of the spray but even closer of the nozzle exit.

In Figure 2, the intact core length for the case considering turbulence is depicted. In the top, the time-averaged of the fuel volume fraction field is shown, where the typical cone shape representing the maximum probability of finding pure fuel is clearly seen. The same information is depicted in the bottom of the Figure but in this case for a given instant, and so, showing a more chaotic behavior.

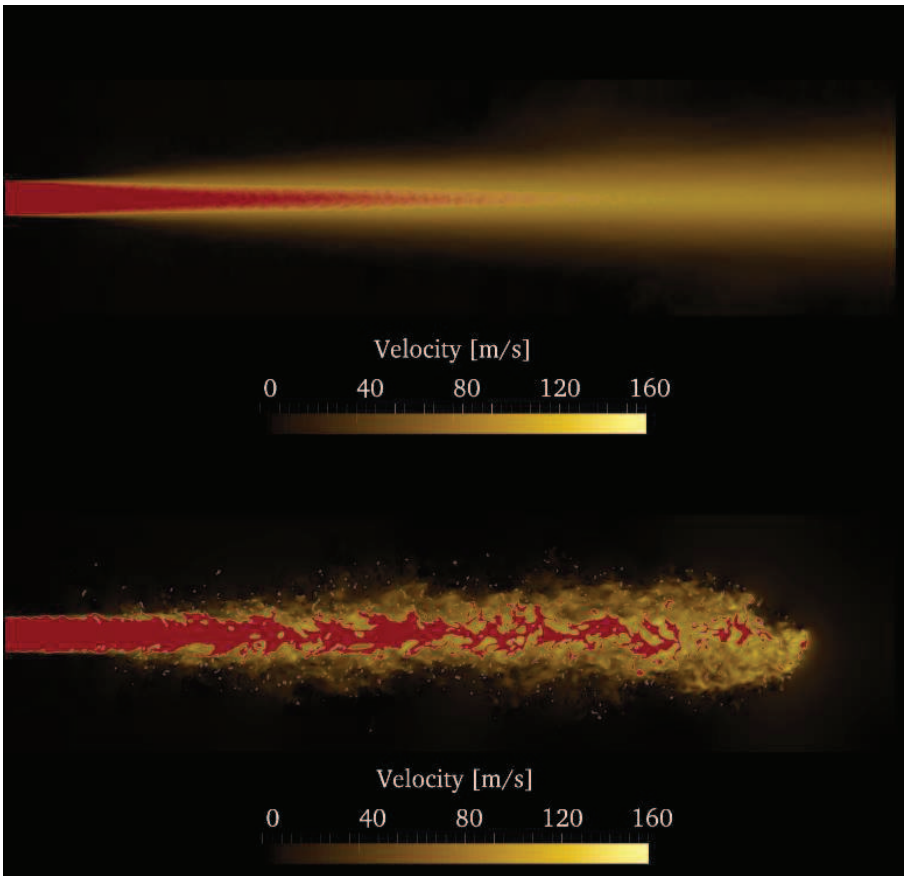

Figure 2. Time-averaged internal core length and instantaneous core length for the case considering turbulence inside the nozzle (Intensity, *I*= 5%, turbulence length scale L=0.1 *D*).

As a conclusion, it can be drawn that VOF DNS-like simulations are a powerful tool to study the mechanisms that enhance the atomization and it is of primary importance to take into account the turbulence generated in the nozzle since turbulence greatly improves atomization process, increasing the number of droplets and drastically reducing the external non-perturbed length and intact-core length. Further details on this investigation can be found in [8].

Nomenclature

B	Logarithmic law constant [-]
D	Nozzle Diameter [m]
I	Turbulence Intensity [%]
L	Turbulent lengthscale [m]
R	Autocorrelation function [-]
Re	Reynolds number [-]
U	Velocity mean component [m/s]
b	Filter coefficient [-]
d	Distance vector [m]
n	Discrete lenghtscale [m]
r	Random component [-]
u	Velocity field [m/s]

u’ Velocity fluctuating component [m/s]
y^+ Non-dimensional wall distance [-]

References

[1] Salvador, F. J., Romero, J.-V., Roselló, M.-D., and Jaramillo,D. Numerical simulation of primary atomization in diesel spray at low injection pressure, J. Comput. Appl. Math., vol. 291, pp. 94–102, 2015.
[2] Reitz, R.D., and Bracco, F.V., Mechanism of atomization of a liquid jet, Phys. Fluids, vol. 25, no. 10, pp. 1730–1742, 1982.
[3] Ling, Y., Zaleski, S. and Scardovelli, R., Multiscale simulation of atomization with small droplets represented by a Lagrangian point-particle model, Int. J. Multiph. Flow, vol. 76, pp. 122–143, 2015.
[4] Klein, M., Sadiki, A. and Janicka, J., A digital filter based generation of inflow data for spatially developing direct numerical or large eddy simulations, *J. Comput. Phys.*, vol. 186, no. 2, pp. 652–665, 2003.
[5] Batchelor, G. K. The theory of homogeneous turbulence. Cambridge university press, 1953.
[6] Eggels, J. G. M. et al., Fully developed turbulent pipe flow: a comparison between direct numerical simulation and experiment, J. Fluid Mech., vol. 268, no. 1, p. 175, 2006.
[7] El Khoury, G. K., Schlatter, P., Noorani, A., Fischer, P.F., Brethouwer, G. and Johansson, A. V. Direct numerical simulation of turbulent pipe flow at moderately high reynolds numbers, Flow, Turbul. Combust., vol. 91, no. 3, pp. 475–495, 2013.
[8] Salvador, F. J., Ruiz, S., Crialesi-Esposito, M., Blanquer, I., Analysis on the effects of turbulent inflow conditions on spray primary atomization in the near-field by DNS , ,InternationalJournalofMultiphaseFlow102,pp.49-63,2018.