



HAL
open science

A novel methodology to simulate fuel injection in multipoint systems - Application to liquid jet in crossflow

Carlos Guillamon, Romain Janodet, Léa Voivenel, Renaud Mercier, Vincent Moureau

► **To cite this version:**

Carlos Guillamon, Romain Janodet, Léa Voivenel, Renaud Mercier, Vincent Moureau. A novel methodology to simulate fuel injection in multipoint systems - Application to liquid jet in crossflow. 5e Colloque du réseau d'INitiative en Combustion Avancée (INCA), Apr 2021, Virtual, France. hal-03513123

HAL Id: hal-03513123

<https://hal.science/hal-03513123>

Submitted on 5 Jan 2022

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Noname manuscript No. (will be inserted by the editor)
--

A novel methodology to simulate fuel injection in multipoint systems

Application to liquid jet in crossflow

Carlos GUILLAMON – Romain JANODET – Léa VOIVENEL – Renaud MERCIER – Vincent MOUREAU

Received: 05/02/2021 / Accepted: date

Abstract

A novel strategy to build lagrangian injectors for spray injection in disperse phase computations is proposed. These injectors can reproduce the spray state (injection location, flow rates, droplets sizes and velocities) extracted from resolved atomization simulations performed with a sharp-interface approach (ACLS/GFM) [1]. The application case is a high-pressure, non-reactive kerosene jet in crossflow (JICF) atomizer configuration [2]. Resolved simulations of atomization for this configuration are performed and validated with the experimental correlation for the jet trajectory. The models learn the spray from these simulations and process it to create spatially distributed injectors for lagrangian droplets injection. Finally, the models are applied to the same configuration and compared to experimental data.

Keywords Multi-phase flows, Atomization, Lagrangian simulations, Jet-in-crossflow

1. Introduction

Simulations of aeronautical combustion chambers require an accurate description of the fuel liquid phase, as the injection and atomization processes have a strong influence on the flame. Nevertheless, computations comprising all the physical processes from injection to combustion cannot be achieved with one numerical formalism due to their different nature. In particular, modeling fuel injection needs special care due to the multi-scale nature of atomization. Approaches aiming at resolving primary atomization, such as diffuse or sharp interface methods, can accurately resolve the liquid-gas interface but become too expensive when many small droplets are present and cannot be used in reactive simulations. Other approaches, such as lagrangian methods, can be used for performing reactive computations at the expense of neglecting atomization. Consequently, they require a proper modeling of injection, which is often circumvented by injecting experimental distributions.

In order to improve the modeling of fuel injection in multipoint systems, we have developed a new methodology to build injectors for lagrangian simulations. These injectors learn the spray from resolved atomization simulations and use it to inject lagrangian droplets in combustion simulations. The strategy is first developed on a non-reacting liquid jet-in-crossflow. The blockage effect from the liquid jet to the gas is characterized in the resolved simulations and modeled with the actuator line method (ALM) [3]. A secondary atomization model is embedded in the lagrangian simulations. The models are then applied to the same jet in crossflow configuration and compared with existing experimental data. Relative influence of the modeling bricks (dense core potential effect, droplet injection assumptions) are also discussed.

2. Numerical setup and operating condition

The test case chosen to simulate fuel injection is a high-pressure kerosene liquid jet in crossflow experiment tested in [2]. It is chosen for its application in multipoint injectors for aeronautical combustion chambers. The numerical domain replicating the test bench is shown in Fig. 1. It consists of a plenum where pressurized air (the crossflow) is injected through a gaseous inlet. Kerosene is injected perpendicularly to the crossflow through a tapered nozzle located at the bottom of the plenum. Fuel is injected through the liquid inlet specified in Fig. 1b following a Poiseuille profile. For crossflow injection, it is mentioned in [2] that the boundary layer thickness at the lip of the injector is 5 mm. As the numerical domain is reduced with respect to the experimental bench, a velocity profile with a boundary layer following a $1/7^{\text{th}}$ power law is prescribed. The thickness of this boundary layer is calculated considering that the gas evolves from the inlet to the injector as a turbulent boundary layer along a flat plate [4]. Outside the boundary layer, the velocity follows a parabolic profile.

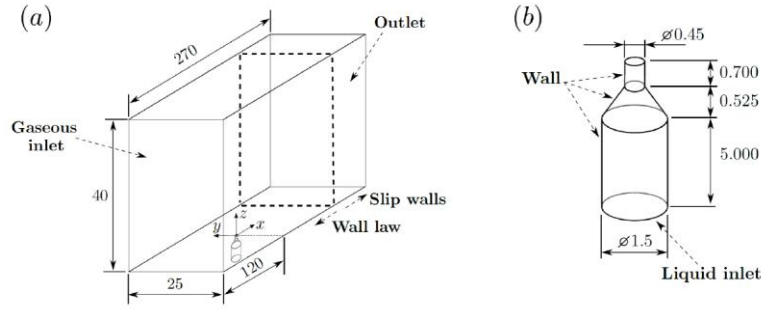


Fig. 1 Numerical domain and boundary conditions of the experimental facility tested by [2].
a) Complete domain. b) Detailed view of injection nozzle. All dimensions are in mm

The operating point of the jet in crossflow is governed by two main dimensionless numbers: the momentum ratio q and the gaseous Weber number We_g . The condition simulated is the baseline case reported in [2], defined by $q=6$ and $We_g=1470$.

3. Resolved simulations of jet in crossflow atomization

3.1. Accurate Conservative Level-Set methodology

In order to solve for primary atomization, the interface capturing method known as Accurate Conservative Level-Set (ACLS) method has been used [1]. This methodology distinguishes between phases by defining a level-set function ψ which is a hyperbolic tangent profile bounded between 0 and 1. The interface is denoted by the iso-level $\psi = 0.5$, lower values denote gas phase and larger ones indicate liquid phase. The pressure jump at the interface due to surface tension is accounted for with the Ghost-Fluid Method. Furthermore, in order to better resolve the interface dynamics and save computational resources, this method is complemented with an Adaptive Mesh Refinement (AMR) strategy to refine the mesh locally at the liquid-gas interface by specifying the element size Δx_{\min} [5]. More details regarding this methodology can be found in [6].

3.2. Results

Two mesh resolutions were simulated for the operating point previously specified: $\Delta x_{\min} = 20 \mu\text{m}$ (coarse mesh) and $10 \mu\text{m}$ (fine mesh). They can be specified with respect to the injector diameter as $d_{\text{inj}}/\Delta x_{\min} = 22.5$ and 45 respectively. Fig. 2a shows the trajectories obtained for both cases compared with an experimental correlation provided by [2]. The trajectories correspond to the windward side of the jet and have been time-averaged. Both numerical trajectories show a good agreement with the experimental one. The finest resolution trajectory is closer to experiments, while the coarsest resolution is slightly underestimated.

Fig. 2b shows an instantaneous snapshot of the developed jet for the coarse mesh. The jet shows a good behavior: it leaves the nozzle and bends towards the crossflow direction as a consequence of its interaction with the air. Instabilities leading to primary atomization are seen in the windward side of the main jet column (column breakup mechanism), also known as dense core. Consequently, the jet will break into big blobs and ligaments that will keep on atomizing further downstream. At the same time, droplets are stripped-off the dense core closer to the injection nozzle due to the strong shear force (surface breakup mechanism).

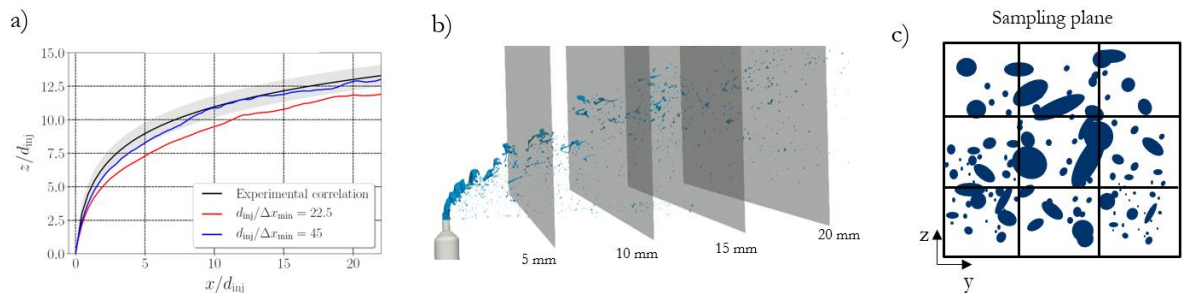


Fig. 2 a) Trajectory of the simulations performed compared with the experimental trajectory from [2]. b) Instantaneous view of the jet for resolution $d_{\text{inj}}/\Delta x_{\min} = 22.5$, showing also four spray sampling planes. c) Schematic representation of spray spatial discretization in one sampling plane.

4. Lagrangian injection models

4.1. Flowchart and submodels

The developed models are represented by the flowchart of Fig. 3. **Resolved atomization simulations** (see Section 3) are performed, spray characteristics (droplets sizes, flow rates, centroid location and velocities) are extracted and processed by the **lagrangian injectors learning** block to create the injectors and initialize **lagrangian simulations**. The injectors learning part performs the following steps:

- **Spray sampling.** Droplets are tracked by their centroid and accumulated in planes perpendicular to the crossflow direction (see Fig. 2b)
- **Spatial discretization.** The accumulated spray is discretized into elements for an in-plane characterization (Fig 2c). Each element will compose a separate spray with different statistics of distributions (diameters, velocity) and scalars (mean velocities, fluxes, mean diameters).
- **Spray convergence** is checked in order to know if enough droplets have been accumulated to build the features statistics. For that purpose, droplets size distribution is compared at several time instants. Convergence is achieved when it does not change as more droplets are accumulated.

Once spray features are converged, lagrangian injectors can be built. As only one operating point has been simulated here, the learning process consists in a simple linear interpolation of the discrete statistics obtained at the same sampling location. An example of Sauter Mean Diameter (SMD), volume flux and mean axial velocity maps obtained for a sampling plane located at $x = 10$ mm (see Fig. 2b) which can be used for injection is shown in Fig. 4. The developed models are complemented with two extra submodels (top part of Fig. 3):

- **Dense core learning for liquid jet/air interaction** to replicate the blockage effect of the dense core in lagrangian simulations, where it is not taken into account a priori. ALM [3] is used to substitute it by a body force applied to the gas that accounts for its presence. This body force, named actuator, is learnt from resolved simulations.
- **Secondary atomization models** are plugged into the lagrangian simulations to account for further breakup of spherical droplets.

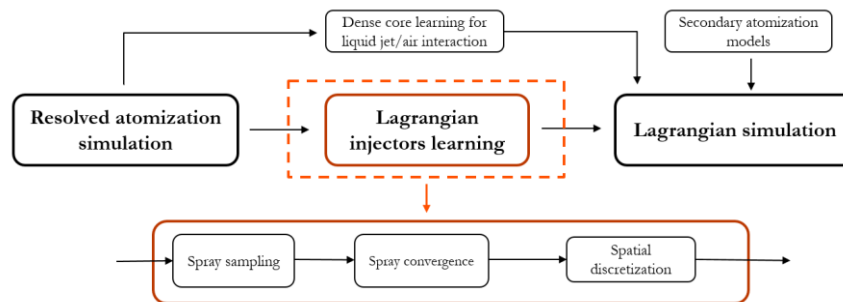


Fig. 3 Flowchart of proposed models for lagrangian injection

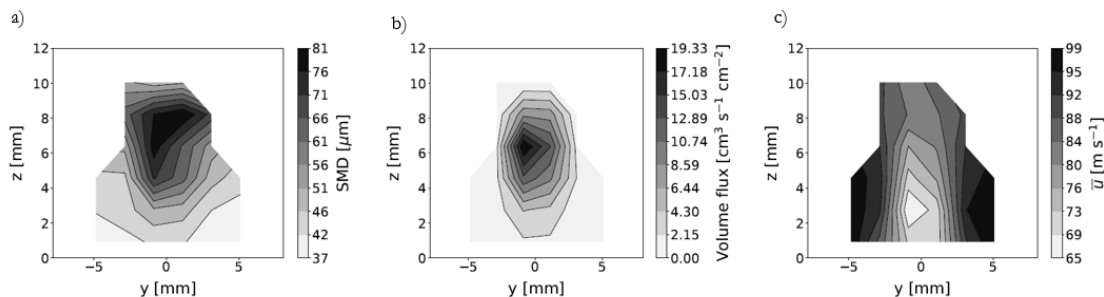


Fig. 4 Maps of a) SMD b) Liquid flow rate c) mean liquid axial velocity obtained after discretization of spray sampled at $x = 10$ mm

4.2. Application to a lagrangian simulation

A lagrangian simulation has been performed by injecting the statistics at $x = 10$ mm shown in Fig. 4. Mean and RMS velocities in the axial, lateral and vertical directions are also prescribed. Gorokhovski's stochastic model for secondary breakup has been used [7]. Validation with experimental data is done by comparing the maps fluxes in the plane $x = 80$ mm (which is not accessible with the resolved simulations). Results are shown

in Fig. 5. The flux values are in the same range as the experiment, with a slight decrease of the maximum flux value in the simulation. The circular pattern of the experimental spray is recovered, with a difference in the bottom part due to droplets closer to the wall. Symmetry with respect to the axis $y=0$ is shown as in the experiments. The spray bounds and the maximum flux location are overestimated, which is attributed to the injection velocity imposed to the droplets and to an underestimation of the force caused by the dense core blockage effect. A more refined modeling of the imposed actuator and another strategy for imposing the droplets velocities could improve the results shown.

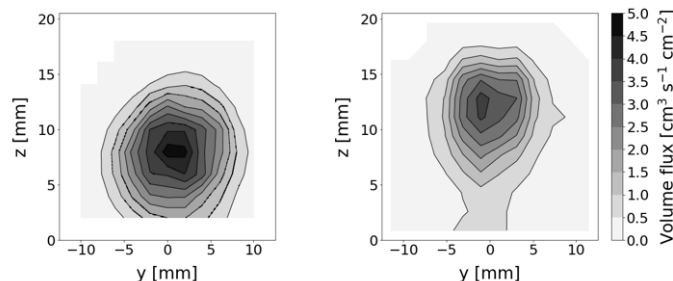


Fig. 5 Maps of volume flux from experiments (left) and from the performed simulation (right)

5. Conclusions

A new strategy to build lagrangian injectors has been proposed. A jet in crossflow test case has been simulated and processed to build the models. Injectors have been developed by studying the resulting spray from resolved atomization simulations. The models are complemented with refined physics such as secondary atomization models and liquid-air interaction. Finally, the obtained injectors have been tested in the same test case, showing a good prediction of the circular spray shape and the flux values. However, the maximum vertical flux location and the spray bounds are overestimated. Improvements will be done by a better determination of the injection velocities and by a more accurate modeling of the dense core blockage effect, which is thought to currently underestimate the force imposed by the actuator on the air flow. Further research includes a more accurate spatial discretization of the injectors by quad-trees [8] and the implementation of subgrid dispersion models [9]. In the long term, the models will be fed with more operating conditions and finally applied to an academic multi-staged fuel injector more representative of aeronautical injection systems [10].

Acknowledgments

This project has received funding from the European Union Horizon 2020 research and innovation program under the Marie Skłodowska-Curie grant agreement No. 765998 in the project ANNULIGHt. Computer resources have been provided by GENCI, France, under the allocation A0092B11072. The authors would also like to acknowledge Thierry Poinot from CERFACS, France, for fruitful discussions on the methodology.

References

1. Desjardins et al., An accurate conservative level set/ghost fluid method for simulating turbulent atomization, *J. Comp. Physics*, vol. 227, pp. 8395-8416 (2008)
2. Becker and Hassa, Breakup and atomization of a kerosene jet in crossflow at elevated pressure, *Atom. Sprays*, vol. 11, pp. 49-97 (2002)
3. Sørensen and Shen, Numerical Modeling of Wind Turbine Wakes, *J. Fluids Eng.*, vol. 124, pp. 393-399 (2002)
4. White, *Viscous Fluid Flow*. McGrawHill, Singapore (2006)
5. Leparoux et al., Primary atomization simulation applied to a jet in crossflow aeronautical injector with dynamic mesh adaptation, 14th Triennial International Conference on Liquid Atomization and Spray Systems ICLASS (2018)
6. Janodet et al., A massively parallel accurate conservative level set algorithm for simulating turbulent atomization on adaptive unstructured grids (2020) <hal-03024186>
7. Apte et al., LES of atomizing spray with stochastic modeling of secondary breakup, *Int. J. Mult. Flow*, vol. 29, pp.1503-1522 (2003)
8. DeZeeuw and Powell, An adaptively Refined Cartesian Mesh Solver for the Euler Equations, *J. Comp. Physics*, vol. 104, pp. 56-68 (1993)
9. Gosman and Ioannides, Aspects of computer simulation of liquid-fueled combustors. *J. Energy*, vol. 7, pp. 482-490 (1983)
10. Providakis, PhD Thesis, Ecole Centrale Paris, Châtenay-Malabry, France (2013)

Carlos GUILLAMON

Safran Tech, Rue des Jeunes Bois, Châteaufort, 78114 Magny-les-Hameaux, France

E-mail: carlos.garcia-guillamon@safrangroup.com