Numerical simulation of primary atomization of a sheared liquid sheet. Part 2: Comparison with experimental results

P. Villedieu, G. Blanchard, D. Zuzio

Numerical simulation of primary atomization of a sheared liquid sheet. Part 2: Comparison with experimental results

Philippe Villedieu*, Ghislain Blanchard and Davide Zuzio

ONERA (The French Aerospace Lab), France

Abstract

Numerical simulations of the primary atomisation of a sheared liquid sheet have been performed using the two-fluid model of SLOSH code described in [1] by the same authors. From a numerical point of view, several experimental configurations investigated in [2] have been considered by varying the flow parameters. Direct comparisons between numerical and experimental results were made to assess the ability of the numerical model to reproduce the main physical mechanisms of the primary atomisation process as they were experimentally observed.

Introduction

In aircraft combustion chambers, the injection of fuel is commonly achieved by air-blast atomizers which create a spray by pulverising a fuel liquid sheet thanks to strong co-flowing airflows. During this process, two stages can be distinguished: the primary and the secondary atomization. Recently, numerical strategies based on the coupling of an interfacial flow model dedicated to the primary atomization and a dispersed phase model dedicated to the secondary atomization have been investigated [7] to simulate the complete atomization process. In this work, simulations of primary atomisation have been carried out with SLOSH code (described in [1]) for realistic configurations and the results have been compared to experimental results from [2] The objective of these computations is to prove the ability of SLOSH methodology (in which the interface is rather captured than exactly tracked as in sharp interface methods) to accurately simulate primary instabilities and large scale features of the primary atomization process.

It is worth pointing out that our objective was not to make a DNS of the full atomization process (from ligament creation to droplet formation) as in [3],[5] and [7] but rather to capture the larger scale phenomena of the liquid sheet fragmentation mechanisms, having in mind that, in a second step, these resolved scales could be used to provide input data (source term location, droplet velocities and sizes) for the coupling with a spray model.

Code assessment methodology

At ONERA, experimental investigations were carried out for liquid sheet atomization. In particular, the study described in [2] has provided a significant database for the primary atomization of a planar sheared liquid sheet by a planar air-blast atomizer (Figure 1). Here, this database is used as a reference and therefore we refer to [2] for the detailed description of the experimental configuration. The liquid sheet is injected through a slit located at the trailing edge of a NACA airfoil. The numerical study has been focused on one specific geometry which is denoted C9 in [2] (see Figure 2).

Two dimensional simulations of the primary atomisation of a planar liquid sheet have been performed in order to assess the ability of the code to accurately reproduce the global scale behaviour of the liquid sheet oscillations and primary atomization (such as the global oscillation frequency and the mean break-up length which are both mostly governed by the growth of 2D longitudinal Kelvin-Helmholtz instabilities). A parametric study has been carried out in which both the influence of the mesh resolution and the effect of some relevant physical parameters (inlet air jet velocity and thickness) have been investigated.

Preliminary 3D simulations have recently been performed in order to confirm the promising 2D results and to assess that the SLOSH numerical approach is also able to reproduce the 3D mechanisms of the primary atomization process.

Test case description

2D case

The main characteristics of the geometry are shown in Figure 2. For 2D simulations, 2D meshes corresponding to the median plane of the injector were used. In Figure 3, one of the meshes is shown with a

*Corresponding author. Email: philippe.villedieu@onera.fr

colour code to identify boundary conditions. Red line denotes the gas inlet where mass flow rate is specified. Blue line indicates the liquid inlet where a Poiseuille velocity profile is imposed. Green contour shows the pressure outlet boundary condition. Remaining boundary conditions were treated as no-slip walls. For the gas inlet, the mass flow rate was determined by integrating the target gas velocity profiles measured experimentally close to the injector [2]. Correspondence between experimental gas velocities and mass flow rate at gas inlet are given in Table 1. For liquid inlet, the average flow velocity was always set to 2.2m.s^{-1} (only gas velocity varies).

Experimental gas velocity V_g (m.s^{-1}) [2]	30	50	80
Equivalent 2D mass flux rate (kg.s^{-1}.m^{-2})	13.90	19.37	38.86

Table 1. Mass flow rate at gas inlet.

To simulate accurately the primary atomization of the liquid sheet, special care was paid to the mesh quality and the mesh resolution downstream of the liquid injection. Thus, meshes were built for being similar to a Cartesian uniform grid in the region from the liquid inlet up to 30mm downstream. Two mesh resolutions have been tested in this region. The first one, which is called $mesh_{10}$, has 10 cells in the sheet thickness (i.e. Δx=30µm). The second one, which is named $mesh_{20}$, corresponds to 20 cells in the sheet thickness (i.e. Δx=15µm). The total number of cells is approximately 600000 cells for $mesh_{10}$ and 900000 cells for $mesh_{20}$.

The numerical time steps, which are calculated under CFL conditions, were between 10^{-8}s and 10^{-7}s (depending on cases).

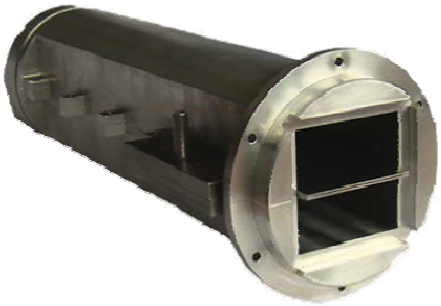


Figure 1. Air-blast atomizer used in [2].

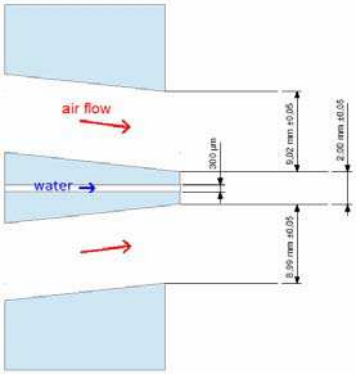


Figure 2. Injector layout for C9 configuration [2].

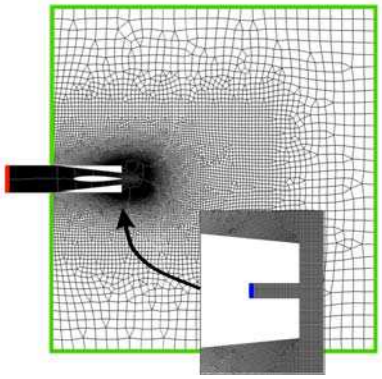


Figure 3. A 2D mesh of the median plane of the injector

Like in [2], water and air are considered for the liquid and gas phases, respectively. Fluid properties are given in Table 2.

	Density ρ (kg.m^{-3})	Dynamic viscosity µ (Pa.s)	Sound celerity (m.s^{-1})	Surface tension σ (N.m^{-1})
air	1.225	1.91×10^{-5}	341	0.07
water	1000	1.0×10^{-3}	1490	

Table 2. Fluids properties

3D case

The selected test case corresponds to a kerosene injector under a 11 bar pressure for which experimental measurements have been performed at ONERA [6]. The physical parameters are reported in Table 3.The computational domain has dimensions of 20 mm × 10 mm × 5 mm. We used a non uniform Cartesian mesh of 11 million cells, strongly refined in the liquid core region. The cell number in the liquid sheet thickness was equal to 10 which corresponds to a characteristic cell size of 30 µm. The time step was approximately equal to 10^{-7}s.

	Density ρ ($kg.m^{-3}$)	Dynamic viscosity µ (Pa.s)	Sound celerity ($m.s^{-1}$)	Surface tension σ ($N.m^{-1}$)	Inlet velocity ($m.s^{-1}$)
air	13.2	1.0×10^{-5}	108	0.00257	25
kerosene	812	1.1×10^{-3}	523		2

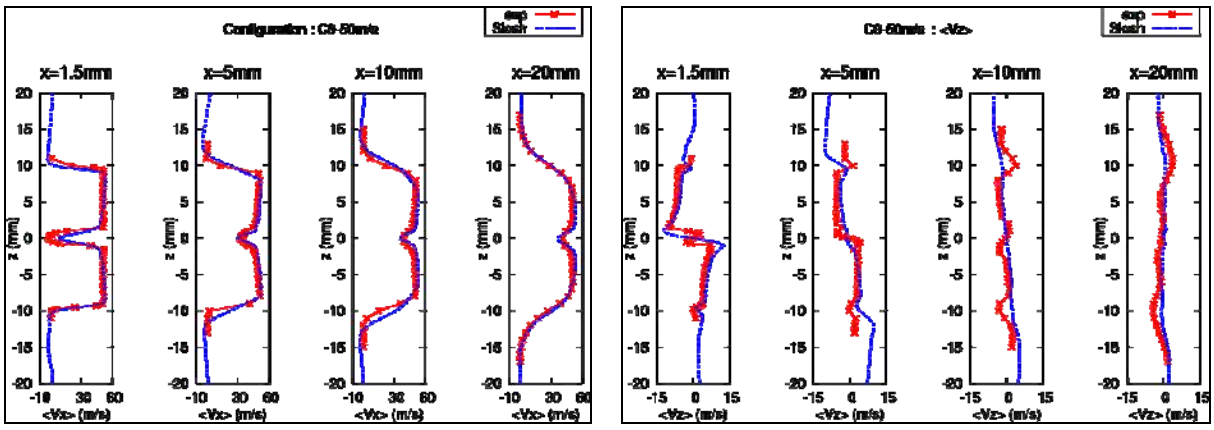
Table 3. 3D simulation parameters

Results and Discussion

One-phase flow 2D simulations

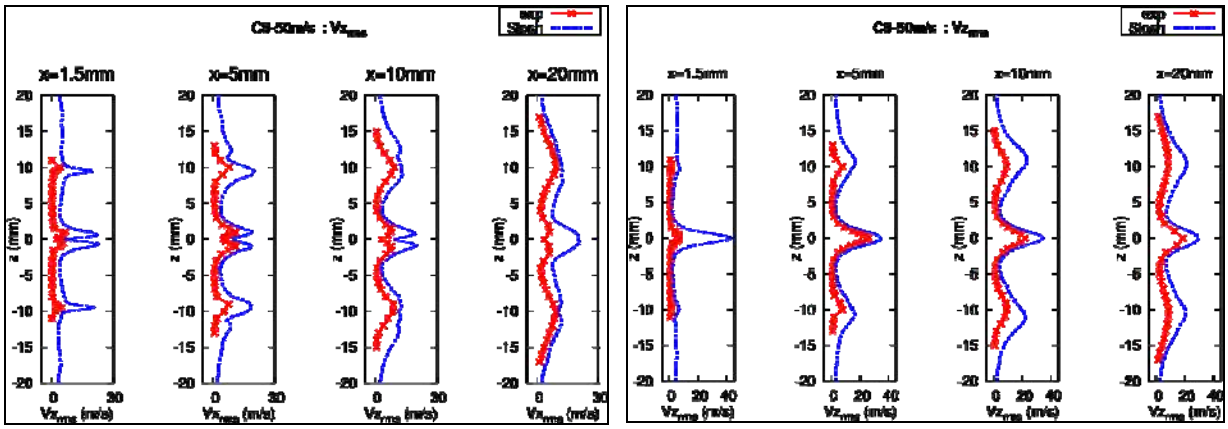
Simulations of the air flow without liquid injection were performed for ensuring that the code is able to accurately compute the air velocity field. This is an essential step before considering two-phase flows.

Figure 4-a shows that results for the vertical profiles of the mean longitudinal gas velocity downstream of the injector nozzle are very close to the experimental ones. A similar comparison is shown in Figure 4-b for the vertical profiles of the mean vertical velocity. The agreement is still correct. Figure 5 shows a comparison between the vertical profiles of the numerical RMS velocity with the corresponding experimental ones. The agreement is clearly not as good as for the mean velocity. While the RMS gas velocity peak positions are qualitatively well predicted, the numerical results tend to overestimate the fluctuation intensity. The same simulations have also been performed with CEDRE ONERA code (http://cedre.onera.fr/) and the same conclusions have been obtained. This discrepancy could be due to the fact that, unlike the mean velocity field, the fluctuation velocity field is 3D. This explanation will be explored in a future work.



(a) (b)

Figure 4. Vertical profiles of mean gas velocity at different distances from the injector nozzle. Comparison between 2D SLOSH simulations and experiments [3]**.** (a) longitudinal mean gas velocity $<V_x>$. (b) vertical mean gas velocity $<V_y>$.



(a) (b)

Figure 5. Vertical profiles of RMS gas velocity at different distances from the injector nozzle. Comparison between 2D SLOSH simulations and experiments [3]**.** (a) longitudinal RMS gas velocity $V_{x\ RMS}$. (b) vertical RMS gas velocity $V_{y\ RMS}$.

Two-phase flow 2D simulations

2D simulations have been carried out with the *mesh*$_{10}$ mesh for the three gas velocity given in Table 1. Figure 6 shows a global visualisation of the liquid sheet for V_g=50m/s. It can be observed that the sheet liquid core is well captured by the numerical methodology [1]. The interface remains sharp until the liquid sheet breakup occurs, as expected. After break-up, the mesh resolution is too coarse for the numerical solution being able to capture small liquid structures. As a consequence, the interface is smeared and the liquid phase is artificially diffused in the gas flow. It is worth pointing out that this is not a problem since the objective here is only to capture the larger scale phenomena of the liquid sheet fragmentation mechanisms, having in mind that, in a second step, these resolved scales could be used to provide input data for the coupling with a spray model.

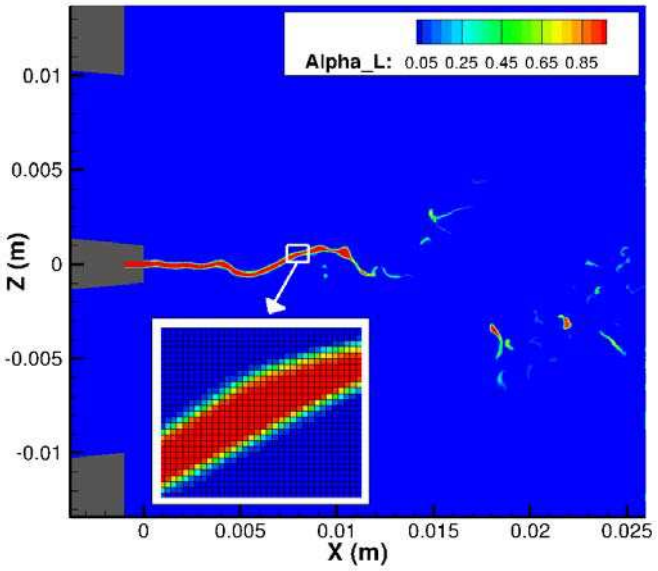


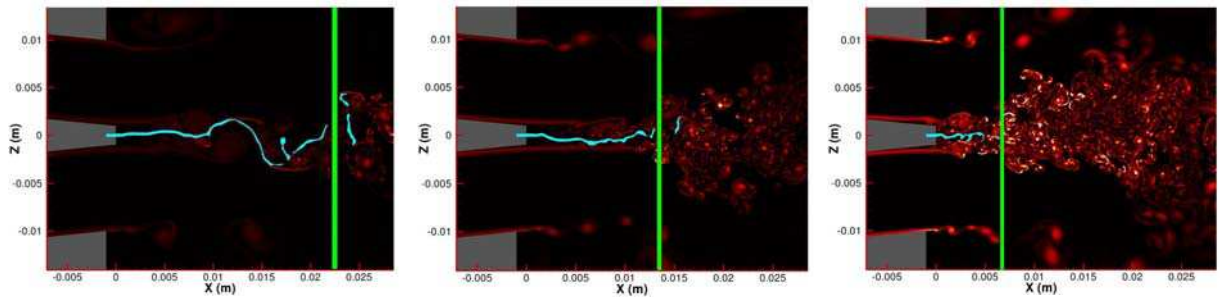
Figure 6. Global visualisation of the simulated liquid sheet.
V_g=50m/s. Mesh: *mesh*$_{10}$. Colour scale: liquid volume fraction field.

Break-up length

From the instantaneous liquid volume fraction fields obtained at time t_n=n.T (with T = 1/10000 s), the corresponding break-up lengths L_b (t_n) were determined by using exactly the same numerical algorithm as for the experimental data which is described in [2]. Figure 7 shows snapshots (at some instants after the transient phase) of the simulated liquid sheet for 3 different inlet air jet velocities. The green line indicates the experimental mean value [2] of the liquid sheet break-up distance from the injector nozzle. Figure 8-a shows a comparison between the numerical and the experimental mean values of the break-up length for the 3 different air flow velocities. It can be observed that the higher the air velocity, the shorter the break-up distance. This tendency is well reproduced in the 2D simulation even if the mean break-up length is underestimated for low gas velocities.

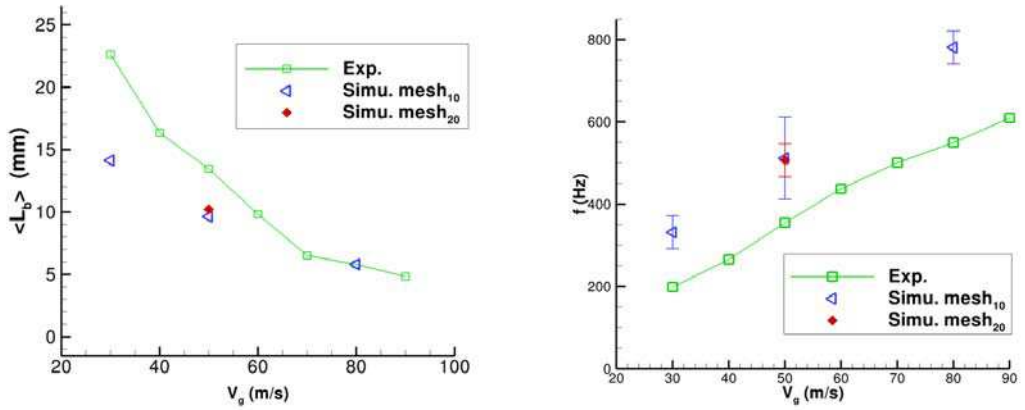
Flapping frequency

From the instantaneous liquid volume fraction fields obtained at time t_n=n.T (with T = 1/10000 s), the maximal vertical position of the liquid sheet, $y_{max,i}(t_n)$, was determined for several longitudinal position x_i. FFT of the discrete signals $y_{max,i}(t)$ were performed to get spectra of the liquid oscillatory motion at the different longitudinal positions. From these spectra, the frequency corresponding to the maximal value of the FFT was identified and compared to the experimental value of the flapping frequency obtained with the same procedure. Figure 8-b shows a comparison between the numerical results with the experimental ones. The tendency of the flapping frequency to increase with the air flow velocity is well reproduced but the quantitative agreement is not very good. This discrepancy could be due to 3D effects which are of course not reproduced in our 2D simulations. The influence of the mesh resolution was also suspected but it can be noticed in Figure 8-b that both meshes, $mesh_{10}$ and $mesh_{20}$, lead to similar results as regards the flapping frequency.



(a) Vg=30m/s (b) Vg=50m/s (c) Vg=80m/s

Figure 7. Influence of the air jet velocity on the break-up length of the liquid sheet. Cyan: snapshot of the simulated liquid sheet. Red scale: vorticity field of the gas phase. Green line: measured mean breakup length from experiments [2].



(a) (b)

Figure 8. Comparison between numerical and experimental results [1] for different air velocities. (a) mean break-up length of the liquid sheet. (b) flapping frequency of the liquid sheet.

Preliminary 3D two-phase flow simulations

Preliminary 3D simulations have been performed in order to assess that SLOSH code is also able to reproduce the 3D mechanisms of the primary atomization process. Figure 9 shows snapshots of the liquid sheet interface (0.5 - iso-surface of the liquid volume fraction). Transverse instabilities which lead to the formation of ligaments can be observed. It is worth mentioning that no artificial perturbation has been necessary for initiating them. These first results are very encouraging as regards the ability of SLOSH numerical method [1] to simulate primary atomization.

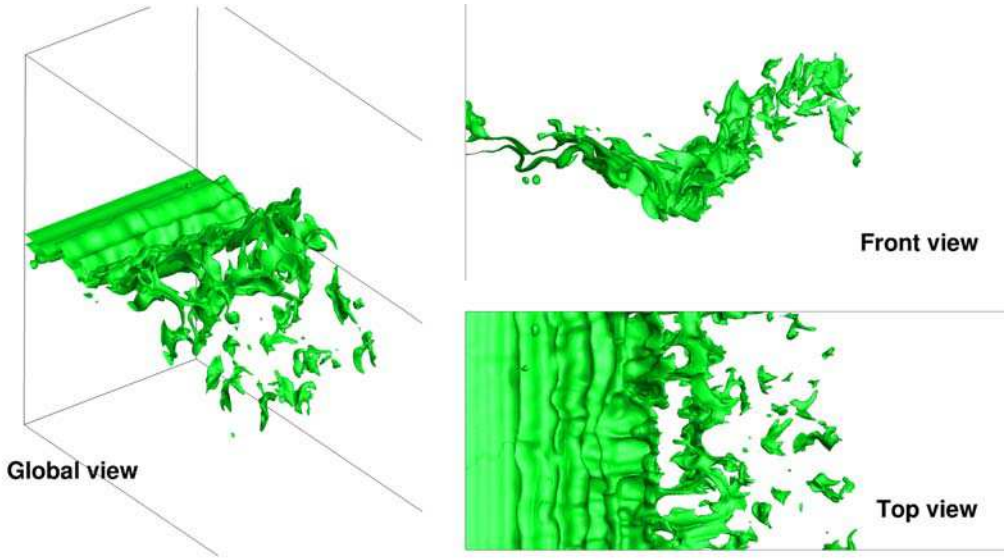


Figure 9. Simulated 3D liquid sheet.

Conclusions and way forward

In this work, numerical simulations of the primary atomization of a sheared liquid sheet have been investigated with the two fluid model of SLOSH code. For now, comparisons between numerical results and experimental data suggest that large-scale mechanisms of the primary atomization are well captured by the numerical simulations even if further quantitative validations are still necessary. These promising results are the starting point for the future development of a coupling approach between SLOSH two fluid model of and a dispersed phase model thanks to a droplet formation model, in order to simulate the whole atomization process.

Acknowledgements

This work was carried out in the framework of the European project FIRST. The authors would like to thank the EC for its financial support. The second author also gratefully acknowledges the funding support of the French Direction Générale de l'Armement.

References

[1] G. Blanchard, P. Villedieu, D. Zuzio, Numerical simulation of the primary atomization of a sheared liquid sheet. Part 1: model and numerical method description, *submitted to the 25th European Conference on Liquid Atomization and Spray Systems, Crete, September 2013.*

[2] B. Déjean, P. Berthoumieu, P. Gajan, Experimental study on the influence of the liquid and air thicknesses on a planar air-blasted liquid sheet, *submitted to the 25th European Conference on Liquid Atomization and Spray Systems, Crete, September 2013.*

[3] G. Tomar, D. Fuster, S. Zaleski, S Popinet, Multiscale simulations of primary atomization, Computers & Fluids, Volume 39, Issue 10, December 2010, Pages 1864-1874

[4] M. Herrmann, A parallel Eulerian interface tracking/Lagrangian point particle multi-scale coupling procedure, Journal of Computational Physics, Volume 229, Issue 3, 1 February 2010, Pages 745-759.

[5] T. Ménard, S. Tanguy, A. Berlemont, Coupling level set/VOF/ghost fluid methods: Validation and application to 3D simulation of the primary break-up of a liquid jet, International Journal of Multiphase Flow, Volume 33, Issue 5, May 2007, Pages 510-524
[6] V. Gutierrez Fernandez. Experimental Study of Liquid Sheet Disintegration in a High Pressure Environment. PhD thesis, Université de Toulouse, 2010.
[7] D. Zuzio et al., Numerical simulation of primary and secondary atomization, , Comptes Rendus Mécanique, Volume 341, Issues 1–2, January 2013, Pages 15-25, http://dx.doi.org/10.1016/j.crme.2012.10.003.