Numerical modeling of an aeronautical injector, from the internal flow to the dispersed spray

Contact: duret@coria.fr

1 PhD subject

1.1 Context and objectives

In order to improve the efficiency of their turbine, the Safran group develop their own fuel injector dedicated to their combustion chamber. This allows a decrease of fuel consumption during all the operative range, and also respect the norms which are more and more restrictive.

The complexity of this kind of injection system and the physical mechanism involved (film instability, atomization …) require depth studies on the injector. Experiment studies permit to establish empirical correlation between the injector used and provide, for example, the probability density function of the number of droplets obtained. However, this kind of experiment cannot cover all the operative range of the injector.

This thesis has for main objective the improvement of the design of the injector and in the end, to improve the injection system. For this purpose, different numerical approaches will be developed and compared with experimental results.

In order to perform this study, a collaboration between the CORIA laboratory (Rouen), the CMAP of the Ecole Polytechnique (Paris) and the SAFRAN group have emerged.

1.2 Scientific approach

The injector studied in this work involved complex physical phenomena (atomisation) on a large range of spatial scales. An internal swirl flow occurs in the injector, coupled with a co-flow of air at the outlet of the injector. Geometrical parameters and operative range can change dramatically the breakup process, the PDF obtained and the spray angle.

The objective is to control and monitor the atomization and the spray angle, in order to optimize it for the combustion process for all the operative range of the injector, including low fuel flows rate. The numerical approach will give a better understanding of the internal and external flow and give some correlations between the spray characteristics and the input parameters.

In order to compute this configuration, two numerical approaches will be used. The first one use the OpenFoam® code which is used at the CORIA to develop the ELSA approach [1]. Then the second one, model the atomization with a diffuse interface approach and high order moments methods to describe the spray polydispersion with the code CanoP[1], based on the p4est module (CMAP-Ecole Polytechnique) [3-8].

As a first approach, the OpenFoam® code will be used to study the internal flow (before injection) allowing the rotation of the flow. Numerical methods implemented in OpenFoam® are a good compromise because the code is compatible with complex geometry, can handle a well resolved mesh and a robust interface tracking method. Results obtained will be used as an inlet boundary condition of the atomization simulation (external flow).

Starting from this boundary condition, two computations will be performed to investigate the spray dispersion and the droplet generation: one with the ELSA formalism, and another one with the method implemented in CanoP. The ELSA method is based on an interface density equation ($\Sigma=S/V$, Eulerian approach), which give an estimation of the droplet diameter obtained in each cell and a first overview of the spray angle. The second method use a diffuse interface method and give a unified model based on topological properties of the interface, allowing a description of the dispersed phase and the dense region of the spray. This method is based on a high order moment method and numerical

[1] The CanoP code has been developed in the thesis of Florence Drui and Mohamed Essadki, in collaboration between EM2C, la Maison de la Simulation and IFPEn.

schemes combining robustness, stability and accuracy. The CanoP code permit adaptive and dynamic mesh refinement (MAR cell-based) and scale efficiently on massively parallel architecture.

Comparisons between results obtained by both approach will be done and will be use to validate the results obtained by the ELSA method for this configuration proposed by SAFRAN, and will clarify the advantages of each numerical representation in the industrial context of SAFRAN.

Then, the best operational method adapted to the use of SAFRAN will be proposed. On the scientific point of view, a better understanding of the numerical prediction and the physical phenomena involved during the full injection, from the injector internal flow to the dispersed region, is expected.

1.3 Thesis planning

The first 6 months of the PhD will be done at SAFRAN (Villaroche), so the student can familiarize with the industrial application, to analyze and describe the injector, and to perform a bibliographic study on the subject.

Then, the student will be based at CORIA (Rouen) full time during one year, with a strong collaboration with the CMAP at Paris. First, to discover and master the numerical tools used (OpenFOAM®), and then to adapt it to the thesis needs. During this period, occasional meeting will be planned: at SAFRAN to confront the numerical results to the experiments, and at CMAP-Ecole Polytechnique to learn the model and numerical methods that have been developed (CanoP code).

Finally, in the last part of the thesis, the working time of the PhD student will be shared between CORIA (90%) and SAFRAN (10%) during one year and a half. This represent approximately 2 or 3 days per month at SAFRAN. This period allows the student to finalize the different deliverables asked and to finish the PhD manuscript.

1.4 Candidate profile

The applicant should justify a research cursus: Master degree (Bac+5) or Engineering school. His CV should show an academic or professional experience in the energetic domain, fluid mechanics and/or numerical simulation, applied mathematics, and also have programming skills. Experience on the OpenFOAM® software will be greatly appreciated.

1.5 References

[1] R. Lebas, T. Menard, P. A. Beau, A. Berlemont, and F. X. Demoulin, “Numerical simulation of primary break-up and atomization: DNS and modelling study,” *Int. J. Multiph. Flow*, vol. 35, no. 3, pp. 247–260, 2009.

[2] F. X. Demoulin, J. Reveillon, B. Duret, Z. Bouali, P. Desjonqueres, and T. Menard, “Toward using Direct Numerical Simulation to improve primary break-up modeling,” *At. Sprays*, vol. 23, no. 11, pp. 957–980, 2013.

[3] F. Drui, A. Larat, S. Kokh, M. Massot, A hierarchy of simple hyperbolic two-fluid models for bubbly flows, submitted (2016) 1-23 https://hal.archives-ouvertes.fr/hal-01349441

[4] M. Essadki, S. de Chaisemartin, M. Massot, F. Laurent, A. Larat, S. Jay, Adaptive mesh refinement and high order geometrical moment method for the simulation of polydisperse evaporating sprays, Oil & Gas Science and Technology, Vol 71, No 5 (2016) 1-25

[5] M. Essadki, S. de Chaisemartin, F. Laurent, M. Massot, High order moment model for polydisperse evaporating sprays towards interfacial geometry description, submitted to SIAM Applied Mathematics (2016) 1-42 https://hal.archives-ouvertes.fr/hal-01355608

[6] F. Drui, A. Fikl, P. Kestener, S. Kokh, A. Larat, V. Le Chenadec, and M. Massot. Experimenting with the p4est library for AMR simulations of two-phase flows. ESAIM Proceedings, M. Campos Pinto and F. Charles Eds. Vol 53 (2016) 232-247

[7] F. Drui, S. Kokh, A. Larat, M. Massot, and S. Vincent. Comparisons of an incompressible and two compressible methods in the case of the problem of dam breaking. in Preparation for Computers and Fluids (2017)

[8] M. Essadki, F. Drui, S. de Chaisemartin, A. Larat, T. Ménard, and M. Massot. Statistical description of a polydisperse spray of deformed droplets through high order moment methods, submitted, available on HAL (2017).