Large-eddy simulation of multiphase compressible flows: application to low-carbon aviation

E. Goncalves Da Silva and P. Parnaudeau

1 Context and objectives of the study

With a 75-90% reduction in greenhouse gas emissions compared with conventional fuel, hydrogen, whose combustion generates only water, is the fuel of the future for aviation. In this context, numerous projects are emerging from various consortia and aircraft manufacturers, based on hydrogen-powered engine architectures. Examples include Onera's AWATAR and CICAV projects, Nasa's X-66, Airbus' ZEROe, etc. [1,2].

However, hydrogen has a number of drawbacks, not least its very low volumetric energy density, which means it has to be used in liquid phase under cryogenic conditions (LH_2 around 20 K). It requires to adapt storage and supply systems, including the fuel circuit that carries hydrogen from the tanks to the engine.

Turbojet engines are supplied with fuel by a compact pump operating at high speed. In some cases, a vaporization phenomenon can develop within the pump, i.e. the liquid vaporizes under the effect of a local decrease in pressure, forming pockets of gas attached to the blades (see Figure 1), leading to a degradation in pumping performance and strong parasitic vibrations. The design of these systems must therefore be capable of controlling the cavitation instabilities that can develop (generating high-amplitude pressure fluctuations, vibrations and radial loads on the pump bearings).

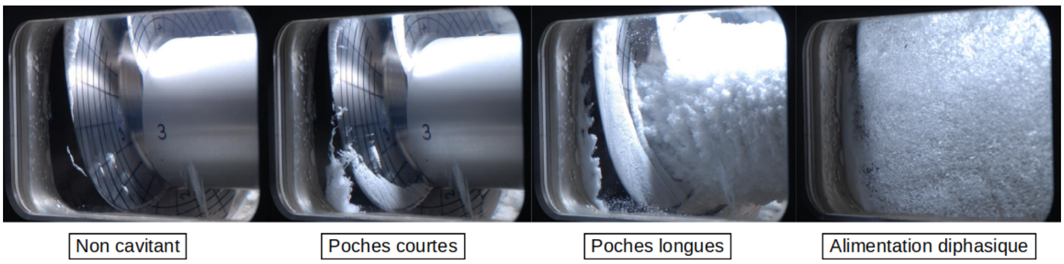

Figure 1: Experimental visualisazion of vapor pockets developing in a pump of rocket launcher - extracted from [3].

Cavitation, a well-known phenomenon in water, is a change of liquid-vapor state occurring at almost constant temperature. The transformation of liquid into vapour consumes energy that is drawn mainly from the liquid, creating local cooling. In the case of water at 20^o, this cooling is very slight (0.1 K) and has little influence on the fluid's thermodynamic quantities. On the other hand, for cryogenic fluids, and in particular for liquid hydrogen, the amplitude of this cooling is much greater (several Kelvin) and the thermodynamic properties of the fluid vary significantly as a result of local temperature variations. For example, the vapor pressure of LH_2 varies by a factor of three between 19 and 23 K. The characteristics (maximum vapour volume fraction, bubble size and shape distribution) and overall topology of gas pockets are altered compared with water.

Despite the efforts made in this field, the numerical simulation of cavitating flows in LH_2 still presents significant scientific challenges:

- In terms of modeling phase change coupled with heat transfer. To date, numerous models have been developed to account for cavitation with thermal effects (reference [4] lists the main formulations). The vast majority of models are based on a vapour volume fraction transport equation including a source term for vapour production and destruction (driven by empirical and adjustable parameters) to reproduce vaporization and condensation phenomena. Unfortunately, the dependence on these two empirical parameters is enormous, and their values are a priori unknown for a given application. More rigorous approaches therefore need to be developed.
- In terms of the complexity of turbulent flows at high Reynolds numbers (10^5 and more) with the presence of three-dimensional separation and recirculations, compressibility effects with the presence of high-amplitude pressure waves, viscous effects generating wall heating, hydrodynamic instabilities such as inter-channel coupling, etc...

In this context, the thesis aims to contribute to the modeling of compressible multiphase problems related to the development of hydrogen-powered technologies for carbon-free aviation. In particular, we aim to develop a numerical approach capable of taking into account small-scale effects interacting with multiphase structures, in order to numerically investigate realistic configurations. We propose to jointly address the following challenges:

- Build a large-scale numerical approach to study the complex interaction between turbulence and multiphase structures;
- Develop well-posed multiphase models for liquid hydrogen with respect to thermodynamic equation-of-states and complex interfacial exchanges;
- Perform high-fidelity simulations of configurations of interest for the analysis and better understanding of physical phenomena and instability mechanisms, leading to improved cryogenic pump design.

2 Program

The thesis focuses on the SCB diffuse interface compressible code, which contains a hierarchy of multiphase models: 4-equation, 5-equation and multicomponent models, integrated in a fully parallel environment capable of efficiently exploiting various architectures (CPU and GPU) [5,6].

Due to the high Reynolds numbers observed in applications, the standard approach is based on averaging operators (RANS).However, common turbulence models fail to predict the flow dynamics correctly, especially at small scales. To solve this problem, we plan to develop a large-scale formalism in the presence of a phase transition using an implicit sub-grid approach. This approach exploits the truncation error of the discretisation

schemes to model the effect of the sub-grid scales on the resolved scales. We will extend this approach, currently developed for single-phase flows [7], to compressible two-phase flows. A centred fourth-order scheme will be used in combination with a regularisation technique to model the effect of unresolved scales. Comparisons will be made with another approach based on interface stiffening methods (such as the THINC method) combined with a high-order local reconstruction (of the WENO type or a robust monotonicity preservation method).
In addition, various two-phase models will be tested (HRM relaxation model, thermodynamic tables, heat transfer modeling) and implemented in SCB.

A first validation will be carried out on canonical configurations with low Reynolds numbers for which data are available in the literature. Secondly, 2D and then 3D simulation of a liquid hydrogen flow around a hydrofoil (Hord's test case [8]) will be used to test the two-phase modeling and gain a better understanding of the physical mechanisms and complex self-sustaining dynamics. Comparisons can then be made with the Star-CCM+ software.

References
[1] Yusaf et al. "Sustainable hydrogen energy in aviation - A narrative review", Int. J. of Hydrogen Energy, Vol.52, 1026-1045, 2024.
[2] https://www.airbus.com/en/innovation/low-carbon-aviation/hydrogen/zeroe
[3] G. Boitel, "Effet thermodynamique dans un inducteur cavitant : mesures embarquées de température", thèse de l'INP Grenoble, 2007.
[4] A. Wei, L. Yu , L. Qiu, X. Zhang, "Cavitation in cryogenic fluids: A critical research review ", Physics of Fluids, Vol.34, 101303, 2022.
[5] Goncalves & Parnaudeau, "Comparison of multiphase models for computing shock-induced bubble collapse", Int. J. Num. Meth. Heat Fluid Flow, Vol.22, 2020.
[6] Dubois, Goncalves & Parnaudeau, "High performance computing of stiff bubble collapse on cpu-gpu heterogeneous platform", Comput. & Mathematics Appl., Vol.99, 2021.
[7] Vincente Cruz & Lamballais, "Physical/numerical duality of explicit/implicit subgrid-scale modelling", Journal of Turbulence, Vol.24(9), 2023.
[8] J. Hord, "Cavitation in liquid cryogens," NASA report CR-2156, 1973.