PHD THESIS

Supervisors

Annafederica Urbano (annafederica.urbano@isae-supaero.fr), Associate Professor at **ISAE-SUPAERO**
Sébastien Tanguy (tanguy@imft.fr), Associate Professor at Université Paul Sabatier (Toulouse 3) and **IMFT**

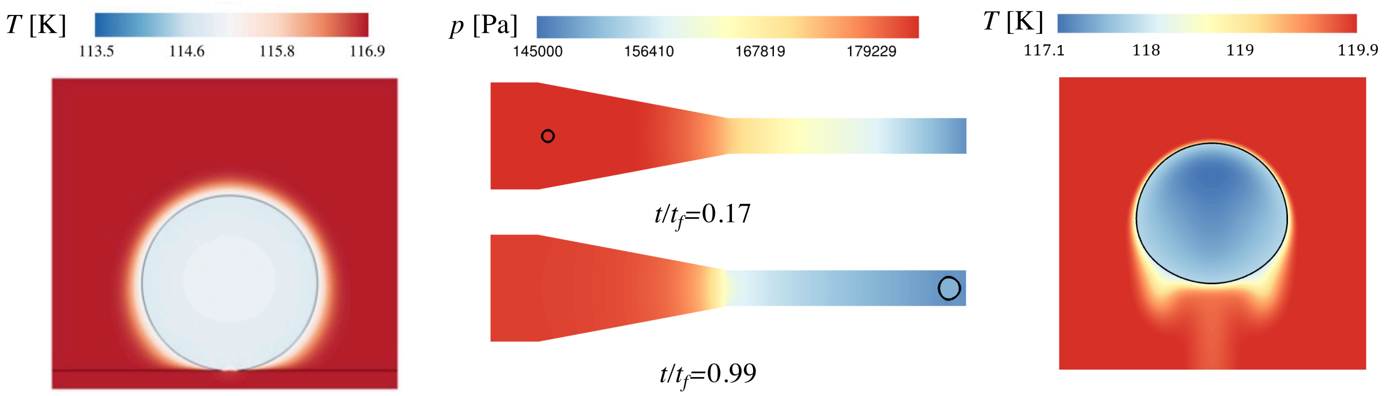
Title

Direct Numerical Simulation of single bubble cavitation at the wall.
Application to phase change in cryogenic tanks in micro-gravity and high-speed flows in variable geometry ducts.

Short title (72 characters) : *DNS of single bubble cavitation at the wall in micro-gravity*

Context

The development of upper stage featuring long costing phase and in-space cryogenic depot reveal important issues associated with cryogenic propellant management in micro-gravity. During long ballistic phase, saturation conditions are reached in the tank. Prior to engine re-ignition or transfer, the propellant is cooled down by venting vapour. Bubbles can be formed by cavitation at the wall and growth under micro-gravity conditions. The consequences are the modification of wall heat transfer and the development of vapour pockets in the liquid. For engine applications, after the tank the propellant is sent to the turbopump and formation of cavitation bubbles or pockets can occur. This can lead to a loss of efficiency and damage the blades. In fact, the bubbles condense where pressure increases and sometimes, this condensation step may be so fast that the bubbles can collapse. Shock waves or high pressure capillary micro-jets may originate from this bubble collapse and solid structures can undergo irreversible damages. Both pool cavitation in a flow at rest (e.g. in a tank) and hydrodynamic cavitation induced by the flow in a duct (e.g. in a pump), are difficult to characterize experimentally at a fundamental level because of the small scales involved that drive the phenomena. The cavitation is still not a well understood phenomenon and scientific advances are required both on the physical understanding and on the modelling capability of this phenomenon.



a) CH_4 bubble cavitation at the wall in zero gravity

b) hydrodynamic cavitation in a convergent duct, H_2O

c) corresponding temperature field around the bubble at $t/t_f = 0.99$

Figure 1: DNS of single bubble cavitation at the wall (a) or in a convergent duct (c and d).

Objectives

The question we want to address is: **What are the growth and collapse rates of bubbles nucleated at the wall, induced by a pressure variation? What are the consequence of these bubbles cavitation on heat transfer, mechanical load on the structure and acoustic trace depending on the pressure level variation and of the fluid of interest?**

In recent years, significant advances have been proposed in the field of numerical methods for two-phase flows but the numerical simulation of cavitating bubbles is still a challenge for which many issues must be overcome to capture accurately hydrodynamics effects, as the pressure drop in a high-speed liquid jet, liquid-vapour phase changes due to pressure drop, acoustic waves and shock waves generation during the bubble collapse. Moreover, it will be necessary to have a method capable of considering thermal effects associated with the wall and capillary effect that can be important at the contact line for wettable fluids. The main objective of the present project **is the development of a numerical solver for the direct numerical simulation of two-phase compressible flows in order to allow the simulation of bubble cavitation, growth and collapse at a wall, in flow or at rest, considering conjugate heat transfer will the wall and accounting for contact line effect phenomena that can be important for cryogens.**

Developments

Interface capturing methods as Level Set/Ghost Fluid methods are developed at IMFT (Institut de Mécanique des Fluides de Toulouse) in the DIVA code since several years for the numerical simulation of two-phase flows [1,2,3], and more recently in collaboration with ISAE-SUPARO [4,5,6]. The compressible solver based on a semi-implicit projection approach and makes use of real gas equations of states, has been recently validated for phase change [4,5] (see Fig. 1). In the present project the solver will be further developed in the present project according to the plan:

1. Bibliography on cavitation models, numerical methods, available experimental data for validation.
2. Developments complex geometries with conjugate heat transfer. The immersed boundary method in the DIVA compressible solver [7] will be adapted in order to allow the treatment of the contact line and conjugate heat transfer in the solid. Moreover, it will be investigated if micro-region sub-grid models are required [6]
3. Development conservative method. The solver, based on a primitive variable formulation of the conservation equations, and on level set formalism will be adapted to obtain a conservative formulation, important for large interface deformation and high velocity. This will be done working on a conservative formulation for the momentum equation and investigating LS/VOF approaches for the interface.
4. Validation and analysis. The developed solver will be validated against experimental data for two types of canonical configuration: 1) pool cavitation of a single bubble in a tank in microgravity 2) Hydrodynamic cavitation in a variable duct geometry. Parametric study will be carried out in order to extract suitable models for cavitation (growth and collapse) and its consequences (acoustics, mechanical and in terms of wall heat transfer).

Work environment

The PhD will be co-hosted by ISAE-SUPAERO and IMFT.

Requirements

Master of sciences in engineering or school of engineering. Background in energetics and fluid mechanics.

References

[1] S Tanguy, M Sagan, B Lalanne, F Couderc, C Colin, Benchmarks and numerical methods for the simulation of boiling flows. J. Comp. Phys. 264 (2014) http://www.sciencedirect.com/science/article/pii/S0021999114000412

[2] G Huber, S Tanguy, J-C Béra, B Gilles. A time splitting projection scheme for compressible two-phase flows. Application to the interaction of bubbles with ultrasound waves. J Comp Phys 302 (2015) http://www.sciencedirect.com/science/article/pii/S0021999115006129

[3] A. Urbano, S. Tanguy, C. Colin, Direct numerical simulation of nucleate boiling in zero gravity conditions. Int. J. Heat Mass Transfer 143 (2019) 118521 https://www.sciencedirect.com/science/article/pii/S0017931019315224

[4] A. Urbano, M. Bibal and S. Tanguy, "A semi-implicit compressible solver for two-phase flows of real fluids", Journal of Computational Physics, 456:111034 (2022)

[5] M. Bibal, M. Deferrez, S. Tanguy and A. Urbano, "A compressible solver for two phase- flows with liquid-vapor phase change. Applications to bubble cavitation." Journal of computational physics , (10.2139/ssrn.4509360)

[6] L. Torres, A. Urbano, S. Tanguy and C. Colin, "On the coupling between Direct Nu- merical Simulation of Nucleate boiling and a Micro-Region model at the contact line". Journal of computational physics, under review (10.2139/ssrn.4523256)

[7]M. Lepilliez, E.R. Popescu, F. Gibou, S. Tanguy. On two-phase flow solvers in irregular domains with contact line. J. Comp. Phys. 321 (2016) pp. 1217–1251 http://www.sciencedirect.com/science/article/pii/S0021999116302388