Numerical tank self-pressurization analyses in reduced gravity conditions

Francesca Rossetti[1,a *], Marco Pizzarelli[2,b], Rocco Pellegrini[2,c], Enrico Cavallini[2,d], and Matteo Bernardini[1,e]

[1]Department of Mechanical and Aerospace Engineering, “Sapienza” University of Rome, Via Eudossiana 18, Rome, 00184, Italy

[2] Italian Space Agency (ASI), Via del Politecnico s.n.c., Rome, 00133, Italy

[a]f.rossetti@uniroma1.it, [b]marco.pizzarelli@asi.it, [c]rocco.pellegrini@asi.it, [d]enrico.cavallini@asi.it, [e]matteo.bernardini@uniroma1.it

Keywords: VOF, Tank Self-Pressurization, Cryogenic, Reduced Gravity

Abstract. In this study, a suitable numerical methodology to study the self-pressurization phenomenon inside a cryogenic tank, in a reduced gravity environment is proposed. This methodology is validated with the results of a benchmark self-pressurization experiment, carried out in the liquid hydrogen tank of the second stage of the Saturn IB AS-203 vehicle. The time-varying acceleration and heat flux due to solar radiation to which the tank was exposed during the experiment, have been modeled in our analysis. Finally, the numerical results show that the proposed methodology allows to reproduce the experimental data with a reasonably good accuracy.

Introduction

Studying the thermo-fluid-dynamics behavior of cryogenic propellant in reduced gravity is crucial for the design of upper stage cryogenic storage tanks. Cryogenic propellant, having a low boiling point, is very sensitive to the heat leaks to which the tank is unavoidably subjected. These heat leaks in a closed, no-venting tank, cause propellant boil-off, self-pressurization, and thermal stratification. The gravity level, g, influences the boiling process, in particular, according to Fritz expression [1], the bubble diameter at departure varies as $g^{-1/2}$ on an upward-facing horizontal surface. Not only the boiling process, but also the dynamics of the free-surface is influenced by the gravity level. Indeed, in case of low Bond number, the free-surface tends to go up along the tank walls. This phenomenon must be countered, through settling strategies, in cases when in-orbit engine start-up and operation is necessary. Important results on propellant behavior in tanks under reduced gravity conditions have been provided by the experiments, nevertheless, the uncertainty of the experimental data and the cost and complexity associated to experiments carried out in a reduced gravity environment highlight the potentialities of CFD simulations. The first self-pressurization experiment in reduced gravity, with data sufficiently detailed for validating storage tank models, was carried out in the liquid hydrogen (H_2) tank of the second stage of the Saturn IB AS-203 vehicle [2]. In the present study, a state-of-the-art numerical methodology [3], which allows to describe the main thermo-fluid-dynamics phenomena occurring in cryogenic tanks during self-pressurization in reduced gravity, is presented. This methodology is validated with the experimental data of Ward et al. [2].

Mathematical formulation and thermophysical properties

Mathematical formulation. The Volume-of-Fluid (VOF) method [4] is used to track the two-phase fluid interface. Moreover, to compute the mass transfer due to phase change, the Lee model [5] is selected. In addition, the Continuum Surface Force (CSF) model [6] is used to address the effects of surface tension force. The flow is modeled as turbulent, being the modified Rayleigh number (based on the liquid height) of liquid H_2 of the order of 10^{14}, so highly above the critical value of 10^{11}. In particular, the SST k-ω model of Menter [7] is selected.

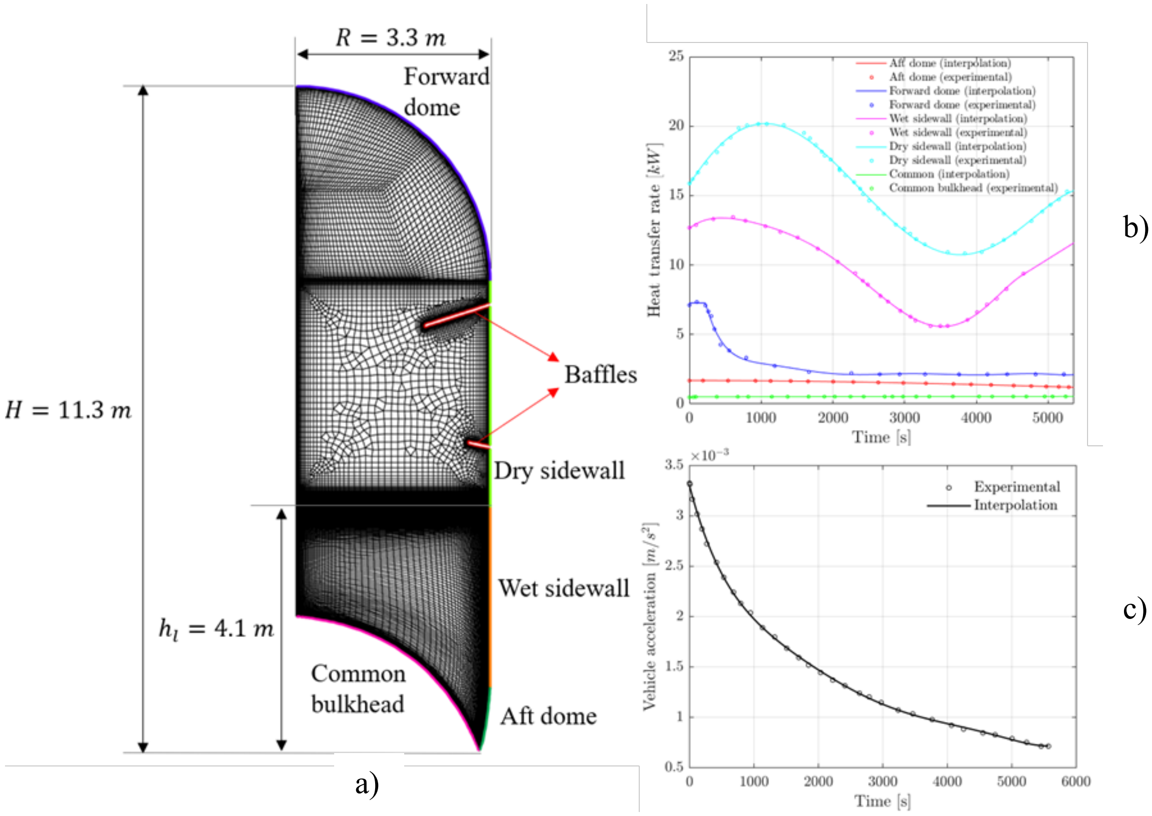
Thermophysical properties. For the liquid H_2, the Boussinesq approximation is selected for the density, and constant thermophysical and transport properties, taken from the NIST [8] database, at the average pressure and liquid temperature, are used. The gaseous H_2 is modeled as an ideal gas, with temperature dependent specific heat, thermal conductivity, and viscosity. The latter are modeled as a piecewise linear fit of NIST data, at the average pressure. The H_2 saturation curve is approximated with a piecewise linear fit of NIST data.

Computational setup and preliminary results

Test case description. The test case under consideration is a self-pressurization experiment carried out in the liquid H_2 tank of the second stage of the Saturn IB AS-203 vehicle [2]. This tank has a height, H, of 11.3 m, a radius, R, of 3.3 m. The initial liquid mass is 7103.3 kg, corresponding to an initial liquid height, h_l, of 4.1 m, and the ullage contains only evaporated H_2. The tank characteristic dimensions are schematized in Fig. 1 (a). The tank shares a common bulkhead with the liquid oxygen (O_2) tank, which is placed below it. The test tank was in a circular low Earth orbit, and, thus, absorbed a time-varying heat flux due to solar radiation. Moreover, it was subjected to a time-varying axial acceleration resulting from the balance between an axial thrust and the drag force.

Flow solver. The pressure-based solver of the commercial CFD software Ansys Fluent® [9] is used to simulate the transient self-pressurization phenomenon. The flow is modeled as 2D axisymmetric, a minimum time step of 0.01 s is used in order to keep the residuals below a proper limit. Second order upwind schemes are used for spatial discretization of the convective terms in the governing equations. Ansys Fluent®'s Compressive scheme [9] is used for the volume fraction equation. The chosen pressure-velocity coupling scheme is SIMPLE. An interpolation of the experimental time-varying heat flux is imposed, as boundary condition, on each part of the tank wall, instead the walls of the baffles are treated as adiabatic. The different parts of the tank wall are indicated in Fig. 1 (a). Fig. 1 (b) represents the evolution of both the experimental and the interpolated heat transfer rate on the different parts of the tank wall. The tank axial acceleration level is modeled as an interpolation of its experimental values. This interpolation, together with the experimental acceleration, are represented in Fig. 1 (c). The tank is initialized at a pressure of 85495 Pa, the initial liquid temperature is 19.72 K, the ullage is initialized using a linear stratification between the liquid temperature and a temperature of 22.5 K [2].

Grid independence study. A grid independence study has been carried out, comparing the results obtained with two grids, a coarse one, having 23401 cells, and a medium one, having 34017 cells. For both grids the height of the first cell at the wall and at the interface is of 4 mm. Fig. 2 *(a)* shows the experimental and the numerical pressure evolutions obtained with the two grid levels. The pressure rise rate obtained during the first 500 s is in good agreement with the experimental one. Unfortunately, experimental data are lacking in the period between 500 s and 4350 s, so it is not possible to verify the accuracy of the numerical prediction in that time



a)

b)

c)

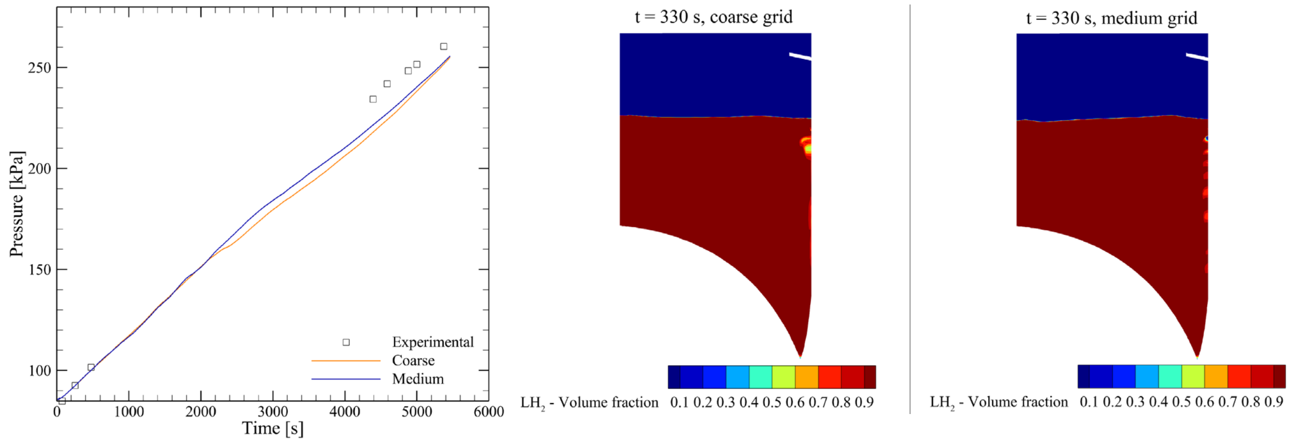
Figure 1: *(a) Selected computational grid with the indication of the tank characteristic dimensions and of the names of the different parts of the tank wall, (b) Experimental [2] and interpolated heat transfer rates, (c) Experimental [2] and interpolated vehicle acceleration.*

interval, but, certainly, some discrepancies with the experimental data arise, as it is confirmed by the numerical underestimation of the pressurization rate, in the last part of the experiment. The difference between the self-pressurization rates obtained with the two grids is negligible, but the finest grid is considered more appropriate because it allows a better representation of the vapor bubbles generating at the wall, as it will be clear in the following paragraph.

Preliminary results. Fig. 2 *(b)* shows the contours of the liquid H_2 volume fraction at 330 s, obtained with the coarse grid (left) and with the medium grid (right). Bubble nucleation is visible at the wall for both grid levels, and the resolution of the finest grid is able to better represent the shape of the bubbles. Nucleate boiling develops due to the high heat flux imposed at the wall. The liquid-ullage interface remains settled during the whole simulation, in agreement with the experiment [2]. Indeed, even if the tank was in orbital flight, the low acceleration imposed to it was sufficient to keep the propellant stabilized. This behavior is explained by the estimation of the Bond number based on the liquid level, which is of the order of 10^3, evidencing the prevalence of the gravity force over the surface tension force.

Conclusions

In this work, a numerical methodology to study the self-pressurization of cryogenic propellant in tanks characterized by reduced gravity conditions is proposed, and validated with experimental data from [2]. The agreement between the numerical and the experimental data is reasonably good, but further analyses are necessary in order to refine the methodology, and assess its applicability to test cases with different propellants, dimensions, and operating conditions.



(a) *(b)*

Figure 2: (a) Experimental [2] and numerical pressure evolutions obtained with the two grid levels used for grid independence study. (b) Contours of the liquid H_2 volume fraction at 330 s, obtained with the coarse grid (left) and with the medium grid (right).

Acknowledgments

This research was jointly funded by Sapienza University and the Italian Space Agency - Agenzia Spaziale Italiana (ASI) as part of the research project: "Technical Assistance for launchers and propulsion" N.2019-4-HH.0 carried out under a framework research agreement CUP:F86C17000080005.

References

[1] W. Friz, Maximum volume of vapor bubbles, J. Physic. Zeitschz. 36 (1935) 354-379.

[2] W. D. Ward, et al., Evaluation of AS-203 low-gravity orbital experiment, NASA CR 94045 (1967) 13.

[3] F. Rossetti, et al., Setup of a numerical methodology for the study of self-pressurization of cryogenic tanks, submitted to Journal Cryogenics (2023).

[4] C. Hirt, B. Nichols, Volume of fluid (VOF) method for the dynamics of free boundaries, J. Comput. Phys. 39 (1981) 201–225. https://doi.org/10.1016/0021-9991(81)90145-5

[5] W. H. Lee, A pressure iteration scheme for two-phase flow modeling, Computational Methods for Two-Phase Flow and Particle Transport (2013) 61–82. https://doi.org/10.1142/9789814460286_0004

[6] J. Brackbill, et al. A continuum method for modeling surface tension, J. Comput. Phys. 100 1992) 335–354. https://doi.org/10.1016/0021-9991(92)90240-Y

[7] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, AIAA J. 32 (1994) 1598–1605. https://doi.org/10.2514/3.12149

[8] M. M. Lemmon, E.W., D. Friend, Thermophysical properties of fluid systems, NIST Chemistry WebBook, NIST Standard Reference Database Number 69, Eds. Linstrom, P.J., and Mallard, W.G., National Institute of Standards and Technology, Gaithersburg MD, 20899. URL: https://webbook.nist.gov

[9] Ansys Fluent documentation, release 2022 R1, 2022.