Preliminary study on the simulation of violent sloshing flows with thermal conduction using the δ-LES-SPH model

Andrea Colagrossi, Salvatore Marrone, Matteo Antuono
CNR-INM, Institute of Marine Engineering
Via di Vallerano 139, 00128, Rome, Italy.
andrea.colagrossi@cnr.it, salvatore.marrone@cnr.it

Javier Calderon-Sanchez, Leo M. González-Gutierrez
ETSI Navales, CEHINAV,
Universidad Politécnica de Madrid (UPM), Madrid, Spain.
leo.gonzalez@upm.es, javier.calderon@upm.es

I. Introduction

One of the most significant challenges facing the aerospace industry is achieving effective decarbonization through the use of clean fuels. In this demanding endeavour, the potential use of hydrogen as a fuel is one of the key possibilities currently being explored by the industry. A critical component of this complex strategy is the design of the fuel tank, where hydrogen would exist in a two-phase liquid-vapour state.

In addition to the complex thermodynamics occurring within the tank — such as heat leakage through the tank walls and heat and mass exchange between the two phases — there are also the accelerations experienced by the tank as part of the aircraft. These accelerations lead to sloshing phenomena in the fluid mixture, which in turn affect the heat and mass exchange between the phases and with the tank's surroundings. Precise control of the thermodynamic variables in both phases is crucial, as this directly impacts the methodology for pumping and injecting the fuel into the engine. This work presents an initial approach to the sloshing problem with heat exchange between the phases and with the external environment using the SPH method.

In particular, the experiments and numerical analysis presented in [1] are considered. A tank, shaped to represent a realistic hydrogen storage tank, is partially filled and subjected to a forced harmonic rolling motion. In the experiment, water is used as both the liquid and gaseous medium. The liquid is heated up to boiling conditions and all air inside the tank is removed to let water vapour fill the empty space in the tank. Before starting the sloshing stage the liquid temperature is $120°C$ and the vapour is at $132°C$. When the tank is set into motion, depending on the sloshing regime induced, a pressure drop is observed in the vapour as a consequence of the heat exchange between the different phases. The pressure in the vapour is measured by means of probes located on the top walls and the heat exchange is significantly favoured by breaking events. Thus, depending on the violence of the sloshing flow, this pressure drop can be abrupt and, in the actual fuel tanks, may represent a critical issue.

The present work is a preliminary study of this complex problem, investigating the potential of reproducing the same thermodynamic behaviour using a two-dimensional single-phase SPH model with thermal conduction. The objective is to simulate only the liquid phase, which drives the sloshing motion and induces the pressure drop in the vapour. Heat flux at the free surface is evaluated under the assumption of a constant and homogeneous temperature in the vapour, which is not explicitly modelled.

The ultimate goal is to verify whether the heat flux, calculated in this manner, can serve as a surrogate for the pressure behaviour in the vapour. Although this model is relatively simplistic, it could be useful for generating sloshing response operators, which require numerous long-duration simulations across a large parameter matrix. Performing such simulations with a full 3D two-phase model would likely be computationally prohibitive, particularly given the high Reynolds numbers and the need for accurate free surface descriptions. Thus, addressing this problem within a simpler 2D single-phase framework is a far more practical approach.

II. Description of the δ-LES-SPH model

The Navier-Stokes equations for weakly-compressible liquid are discretized within a Quasi-Lagrangian SPH framework as follows:

$$
\begin{cases}
\dfrac{d\rho_i}{dt} = -\rho_i \langle \mathrm{div}(\boldsymbol{u}_i + \boldsymbol{\delta u}_i) \rangle + \langle \mathrm{div}(\rho_i \boldsymbol{\delta u}_i) \rangle + \mathcal{D}_i^{\rho} \\
\rho_i \dfrac{d\boldsymbol{u}_i}{dt} = \langle \nabla p \rangle_i + \boldsymbol{F}_i^B + \boldsymbol{F}_i^V + \boldsymbol{F}_i^S + \langle \mathrm{div}(\rho_i \boldsymbol{u}_i \otimes \boldsymbol{\delta u}_i) \rangle \\
\dfrac{d\theta_i}{dt} = \dfrac{1}{\rho_i c_p} \langle \nabla \cdot (\kappa + \kappa_i^T) \langle \nabla \theta_i \rangle \rangle - \dfrac{1}{\rho_i c_p} \boldsymbol{u}_i \cdot \boldsymbol{F}_i^V + \langle \mathrm{div}(\theta_i \boldsymbol{\delta u}_i) \rangle \\
\dfrac{d\boldsymbol{r}_i}{dt} = \boldsymbol{u}_i + \boldsymbol{\delta u}_i, \; V_i(t) = m_i / \rho_i(t), \; p_i = c_0^2 (\rho_i - \rho_0)
\end{cases}
\tag{1}
$$

where θ is the temperature, $\boldsymbol{F}_i^B$ are the body forces, $\boldsymbol{F}_i^V$ the viscous forces and $\boldsymbol{F}_i^S$ the surface tension. The formulation of the pressure gradient $\langle \nabla p \rangle_i$ is the same as in [2] in which a non-conservative renormalized formula with proper corrections for the free surface is proposed. The diffusive term $\mathcal{D}_i^{\rho}$ and $\boldsymbol{F}_i^V$ are derived from a Large Eddy Simulation (LES) SPH model as described in [3] and recently applied in the context of sloshing flows in [4]. As for the surface tension $\boldsymbol{F}_i^S$, the model described in [5] is adopted.

In the energy equation the terms depending on fluid compressibility have been neglected. The parameter c_p is the specific heat capacity and symbols κ and κ_i^T represent, respectively, the molecular and turbulent thermal conductivity. The latter is classically expressed as $\kappa_i^T = \rho_i c_p \nu_i^T / \mathrm{Pr}^T$ where ν^T is the turbulent kinematic viscosity (evaluated as in [4]) and Pr^T is the turbulent Prandtl number. In the present study, following [1], Pr^T is constant and set as $\mathrm{Pr}^T = 2$. The body forces $\boldsymbol{F}_i^B$ are expressed as:

$$\boldsymbol{F}_i^B = \rho_i \hat{\boldsymbol{g}} (1 - \beta \Delta\theta_i) + \rho_i \boldsymbol{f}_i^{NI},$$

in which the Boussinesq approximation has been adopted to take into account buoyancy effects due to temperature variations [6]; β is the thermal expansion coefficient and $\Delta\theta_i = \theta_i - \theta_0$, θ_0 being the temperature at the free surface. The Boussinesq approximation has been already employed and validated in several works in the SPH literature such as, *e.g.*, [7], [8]. Since in the present work a non-inertial frame of reference is employed (simulations are performed in the reference integral to the tank), the gravity vector $\hat{\boldsymbol{g}}$ rotates and the non-inertial accelerations $\boldsymbol{f}_i^{NI}$ are also taken into account (see, *e.g.*, [9]).

The vector $\delta\boldsymbol{u}$ is the shifting velocity related to the Particle Shifting Technique (PST) adopted to regularize the spatial distribution of the particles during their motion. Since the particles are moving with modified velocity $(\boldsymbol{u} + \delta\boldsymbol{u})$ and the above equations are accordingly written in a quasi-lagrangian framework,, the continuity, momentum and energy equations contain terms with spatial derivatives of $\delta\boldsymbol{u}$. The specific law chosen for $\delta\boldsymbol{u}$ is the same reported in [2]. The smoothed divergence operators of (1) are given in [10]. The spatial gradients are approximated through convolution summations with a kernel function W_{ij}. A C2-Wendland kernel is adopted in the present work with support radius set to $2h = 4\Delta x$. Solid boundaries are modelled by means of the clone particle technique as proposed in [11].

III. Problem description

A sketch of the considered tank geometry as described by [1] is provided in figure 1. The tank has two semi-elliptic walls which are composed of piecewise circular arcs as depicted in the sketch. The tank height in the experiment is $D = 0.35m$. The experimental setup described in [1] features a movable platform with a single degree of freedom. During all tests, the tank rotates around its bottom centerplane, with a harmonic motion and an angular amplitude of 3°. In the experiment, water is used as both

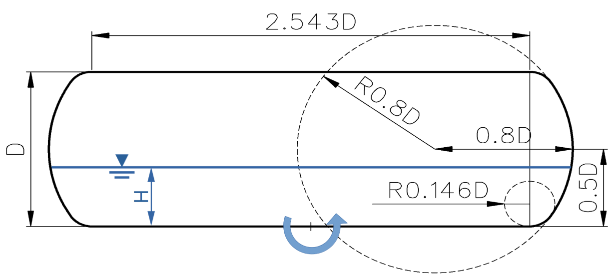

Fig. 1. Tank geometry

the liquid and gaseous medium. The liquid phase has a filling height of $H/D = 0.5$. As mentioned in the introduction, prior to the sloshing stage, there is a temperature difference between the liquid and the vapour. This temperature gap is responsible for the pressure drop observed when violent sloshing develops.

A first study is conducted in order to evaluate whether the adopted SPH model, which includes LES turbulence modelling and buoyancy due to heat expansion, is adequate for this problem. To this purpose a first series of simulation is performed on the tank at rest. In this case the liquid is initially set to $\theta = \theta_0$ and heated through the walls. Dirichlet boundary conditions are employed at the tank walls and at the free-surface and $\theta_{\mathrm{wall}} > \theta_0$, θ_0 being the temperature at the free surface. For this case of natural convection the Reynolds number $\mathrm{Re} = UD/\nu$ is directly determined by the Grashof number, $\mathrm{Gr} = g\beta(\theta_{\mathrm{wall}} - \theta_0)D^3/\nu^2$, as $\mathrm{Re} = \sqrt{\mathrm{Gr}}$.

In figure 2 the obtained results for two different Reynolds numbers, $\mathrm{Re} = 60,000$ and $\mathrm{Re} = 240,000$ are reported. All the physical parameters adopted are those of water and, for the sake of brevity, are not reported here. It is evident that in the case at higher Re the density currents are stronger up to the extent of perturbing the free-surface. In the bottom plot of the same figure the time history of the liquid average temperature is reported: the case at $\mathrm{Re} = 240,000$ exhibit an initial stage characterized by a quite steep increase of the average temperature compared to $\mathrm{Re} = 60,000$. This is due to the predominant role of convective currents but also to the fact that higher values of the turbulent thermal conductivity (linked to turbulent viscosity) are developed.

After that, test cases including sloshing motion have been considered. In this case the liquid temperature is initially set to $\theta = \theta_{\mathrm{wall}} < \theta_0$ where θ_0 is the temperature at the free surface assumed equal to the vapour one. Unlike the previous case, the liquid here is heated from the free surface. (*i.e.* the vapour). In figure 3 two different sloshing regimes are compared. These are characterized by different oscillation periods: $T = T_1$ and $T = 2.39T_1$, where T_1 is the first resonance period of a rectangular tank of the same length. The Richardson number is $\mathrm{Ri} = \mathrm{Gr}/\mathrm{Re}^2 = 0.01$ where, in this case, $\mathrm{Re} = \sqrt{gD}D/\nu = 728,000$.

At lower frequency, the fluid exhibits only small motion, with the free surface remaining essentially flat. Consequently, heat diffusion at the surface occurs primarily through conduction, leading to a stratified temperature distribution. Conversely, at higher frequencies, more violent sloshing develops, resulting in efficient mixing of the liquid. As a result, the time histories of the average temperature in the two cases differ significantly. This aligns with the findings of [1], where similar sloshing regimes were investigated. For these two sloshing cases, convergent results have been obtained (though not shown here due to space constraints).

These preliminary results show that the adopted model is able to describe the influence of the sloshing regime on the heat exchange at the free surface. Comparisons with experimental results and 3D multi-phase simulations are still needed and several regimes have to be studied for a comprehensive description of the phenomenon.

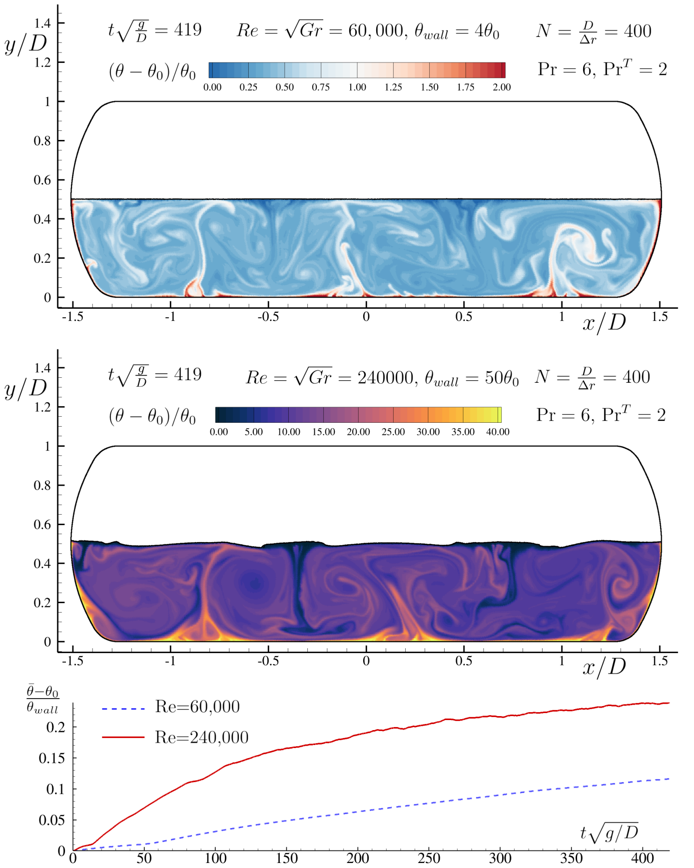

Fig. 2. Preliminary test with the fixed tank having a temperature θ_{wall} larger than the free-surface one θ_0. Two different Grashof/Reynolds numbers, Gr $=$ Re^2, are consider. Top: temperature field at time $t\sqrt{g/D} = 419$, Re=60,000. Middle: temperature field at time $t\sqrt{g/D} = 419$, Re=240,000. Bottom: time histories of the average temperature.

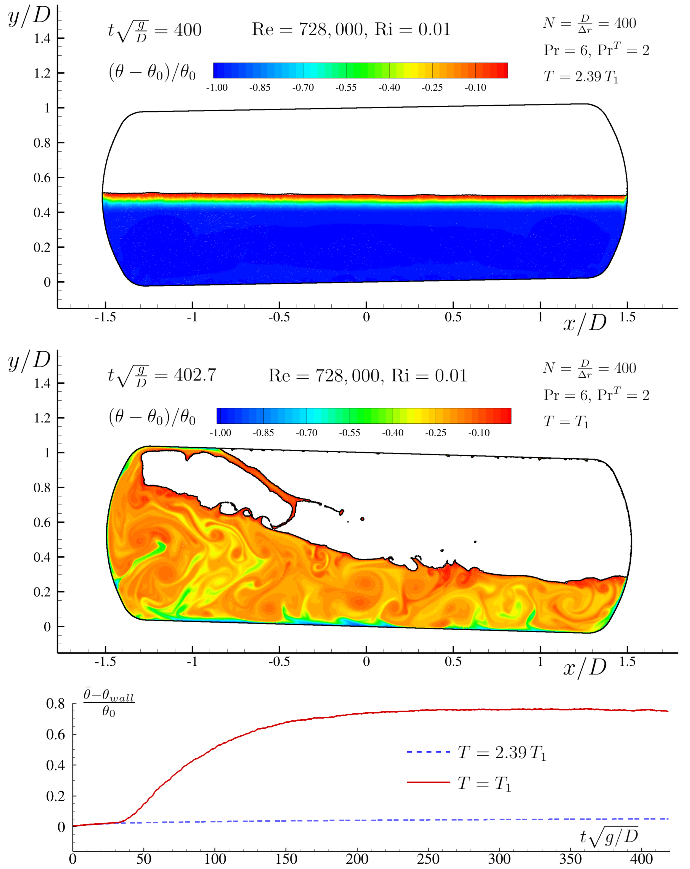

Fig. 3. Sloshing flows with temperature flux at Re=728,000 and Ri=0.01. Top: temperature field at time $t\sqrt{g/D}$=402.7, $T = T_1$ Middle: temperature field at time $t\sqrt{g/D}$=402.7, $T = 2.39\,T_1$. Bottom: time histories of the average temperature for the two sloshing conditions.

Acknowledgment

The research leading to these results was partially funded by the HASTA project (Grant No. 101138003) as part of the European Union Horizon research programme. Views and opinions expressed are however those of the authors only and do not necessarily reflect those of the European Union. Neither the European Union nor the granting authority can be held responsible for them.

This work was also partially supported by the project "Next Generation SPH schemes for complex multiphase flows" NEOGEO (CUP B83C23003850006) in collaboration with Ecole Centrale de Nantes in the framework of their Chair programme funded by Siemens Digital Industries Software.

References

[1] E. L. Grotle and V. Æsøy, "Numerical simulations of sloshing and the thermodynamic response due to mixing," *Energies*, vol. 10, no. 9, p. 1338, 2017.

[2] J. Michel, A. Colagrossi, M. Antuono, and S. Marrone, "A regularized high-order diffusive smoothed particle hydrodynamics scheme without tensile instability," *Physics of Fluids*, vol. 35, no. 10, 2023.

[3] M. Antuono, S. Marrone, A. Di Mascio, and A. Colagrossi, "Smoothed particle hydrodynamics method from a large eddy simulation perspective. generalization to a quasi-lagrangian model," *Physics of Fluids*, vol. 33, no. 1, p. 015102, 2021.

[4] C. Pilloton, J. Michel, A. Colagrossi, and S. Marrone, "A numerical investigation on three-dimensional swirling instability in viscous sloshing flows," *Applied Ocean Research*, vol. 138, p. 103621, 2023.

[5] A. Vergnaud, G. Oger, D. Le Touzé, M. DeLeffe, and L. Chiron, "C-csf: Accurate, robust and efficient surface tension and contact angle models for single-phase flows using SPH," *Computer Methods in Applied Mechanics and Engineering*, vol. 389, p. 114292, 2022.

[6] L. Landau and E. Lifshitz, *Course of Theoretical Physics, Vol. 6: Fluid Dynamics*. Butterworth-Heinemann, 1987.

[7] M. Hopp-Hirschler, M. S. Shadloo, and U. Nieken, "A smoothed particle hydrodynamics approach for thermo-capillary flows," *Computers & Fluids*, vol. 176, pp. 1–19, 2018.

[8] Z. Zhang, K. Walayat, C. Huang, J. Chang, and M. Liu, "A finite particle method with particle shifting technique for modeling particulate flows with thermal convection," *International journal of heat and mass transfer*, vol. 128, pp. 1245–1262, 2019.

[9] A. Colagrossi, G. Colicchio, C. Lugni, and M. Brocchini, "A study of violent sloshing wave impacts using an improved sph method," *Journal of hydraulic research*, vol. 48, no. sup1, pp. 94–104, 2010.

[10] M. Antuono, P. Sun, S. Marrone, and A. Colagrossi, "The δ-ALE-SPH model: An arbitrary lagrangian-eulerian framework for the δ-SPH model with particle shifting technique," *Computers & Fluids*, vol. 216, p. 104806, 2021.

[11] M. Antuono, C. Pilloton, A. Colagrossi, and D. Durante, "Clone particles: A simplified technique to enforce solid boundary conditions in sph," *Computer Methods in Applied Mechanics and Engineering*, vol. 409, p. 115973, 2023.