Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 81 (2015) 856 – 865

Energy Procedia

[S1]69th Conference of the Italian Thermal Engineering Association, ATI 2014

A methodology for the reduction of numerical diffusion in sloshing analyses through automated mesh adaptation

Stefano Fontanesi[a]*, Giuseppe Cicalese[a], Gianluca De Pasquale[a]

[a]*Department of Engineering "Enzo Ferrari", University of Modena and Reggio Emilia, Via P. Vivarelli 10, 41125 Modena, Italy*

Abstract

The paper proposes a methodology to improve the accuracy of Volume of Fluid (VoF) multiphase problems involving liquid/gas sloshing in fuel/lubricant tanks without penalizing the computational cost of the simulations. In order to correctly track the complex trajectory of the liquid/gas interface and the presence of liquid droplets in the gas phase, the VoF method requires a fine mesh at each interface location to reduce modeling errors.

The investigated case is a lubricant tank of a sport car subject to typical race track maneuvers. Due to the geometrical extent and the complexity of the computational domain and to the relevant accelerations, resulting in dispersed liquid structures within the gas phase, the use of a generalized fine mesh would result in computational costs far beyond the industrial practice.

A methodology is then proposed to reduce the overall number of computational cells through a combination of local interface tracking and mesh refinement, which is combined with an active control of the time step to comply with Courant-Friedrichs-Lewy number limits.

The methodology is at first validated against experimental measurements for a simplified test case, and then applied to the actual oil tank sloshing case, showing a relevant reduction of the numerical diffusion and a consequent higher accuracy.

Peer-review under responsibility of the Scientific Committee of ATI 2014

Keywords: VOF; sloshing; multi-phase; CFL; mesh

1. Introduction

Fluid motion inside tanks has been studied since a long time for the analysis of a multitude of problems involving not only fluid dynamics but also the interaction of the fluid with the containing structure. From a general perspective, this interaction can be analyzed from two different points of view: the first one consists of determining

* Corresponding author. Tel.: +39-059-2056114; fax: +39-059-2056126.
E-mail address: stefano.fontanesi@unimore.it

Peer-review under responsibility of the Scientific Committee of ATI 2014
doi:10.1016/j.egypro.2015.12.096

the forces exchanged between the fluid and the tank due to the motion of the fluid mass, while the second one is particularly focused on the forecast of the temporal evolution of the free surface shape and position as a consequence of an over-imposed motion.

The first kind of analysis, better known as FSI (Fluid Structure Interaction), is widely adopted in the transportation field: both for marine and ground applications, the inertial forces due to the motion of the fluid mass induced by pitch and roll oscillations have to be taken into account in terms of mechanical stresses on the walls and effects on the dynamic behavior of the vehicle. In aerospace applications, the instability induced by the fluid is even more complex to be analyzed because the fuel sloshing can affect the performance of the aircraft especially during landing operations. As an example, during the first lunar landing with Apollo 11, a dramatic propellant slosh problem occurred at the end of a yaw maneuver.

The investigation on the free surface position, on the other hand, plays an important role in all the applications where the liquid mass is not relevant enough to influence the handling of the vehicle, but problems could arise if the free surface moves to "critical" locations: for example, criticalities may arise if such fluid motion discovers the suction side of a feed pump, with a consequent sudden pressure drop in the circuit. If this behavior appears in the lubricating circuit of an internal combustion engine, for instance, mechanical failures can occur even if the lack of lubrication lasts just a few tenths of milliseconds.

The experimental investigation of sloshing problems is extremely difficult, since it should enable the visualization of the free-surface motion in transparent models, the acquisition of a series of problem-relevant experimental data through pressure/strain transducers and the application of motion and accelerations as close as possible to those experienced by the object under actual operations.

As a consequence, a massive use of CFD (Computational Fluid Dynamics) tools is emerging for the analysis of sloshing, since they can be faster (and cheaper) than the experimental practice and more effective in the understanding and rationalization of both local and global phenomena; CFD, in fact, allows to mimic both the fluid-structure interaction and the free surface motion, in order to properly calculate the mechanical loads and to foresee possible undesired deformations of the liquid mass. Despite this very promising premise, however, the CFD simulation of multi-phase/multi-fluid sloshing problems usually requires very long calculation times and massive computational resources, because a detailed description of the fluid/gas interface in usually geometrically complex domains is needed [1-3].

This activity aims at optimizing the simulation procedure in order to well predict the motion of the free surface by reducing the overall calculation time.

2. The Activity

The final goal of the activity is the accurate prediction, through CFD simulations, of the behavior of a lubricant tank for high performance engine applications under actual vehicle accelerations on a racetrack. In particular, because of the high accelerations experienced by the tank, two main issues need to be addressed: first, if there are vehicle maneuvers where the pump feeding section might be uncovered; second, if backflow into the air/oil separator could occur. Within the first issue, moreover, it is necessary to take into account the air entrapment during the oil motion: a sudden drop in the oil pump delivery pressure, in fact, can be addressed to a sudden drop of oil density, even if moderate, and to the arise of cavitation phenomena induced by the trapped air itself.

3. The CFD approach to liquid sloshing

In sloshing problems a free surface separating two (or more than two) fluids exists. The phases are mixed at a macroscopic scale and they are characterized by different convection velocities. The different fluids are immiscible and each of them represents a large structure inside the computational domain.

To well track the interface motion, a VoF (Volume of Fluid) computational sub-model is adopted in the CFD calculations [4, 5]. This model assumes a common pressure, velocity and temperature field for all the phases placed within the same cell. The governing Navies-Stokes equations are solved in the same way as for a single phase flow, but density ρ, molecular viscosity μ and specific heat c_p are calculated on the basis of the volume fractions α_i of the components:

$$\alpha_i = \frac{V_i}{V}$$

$$\rho = \sum_i \rho_i \, \alpha_i$$

$$\mu = \sum_i \mu_i \, \alpha_i$$

$$c_p = \sum_i c_{p,i} \, \alpha_i \tag{1}$$

The interface between the phases is calculated by solving a transport equation of the volume fractions α_i:

$$\frac{d}{dt}\int_V \ \propto_i dV + \int_S \ \propto_i (v - v_g) \cdot da = \int_V \left(s_{\propto_i} - \frac{\propto_i}{\rho_i}\frac{D\rho_i}{Dt}\right) dV \tag{2}$$

where $s_{\propto_i}$ is the source or sink term of the i[th] phase and $^{D\rho_i}/_{Dt}$ is the Lagrangian Derivative of the phase densities ρ_i.

The shape of the interface between the different phases can be described with a level of accuracy that depends on the grid resolution: interface details larger than cell minimum dimension can be represented, as explained in Fig. 1.

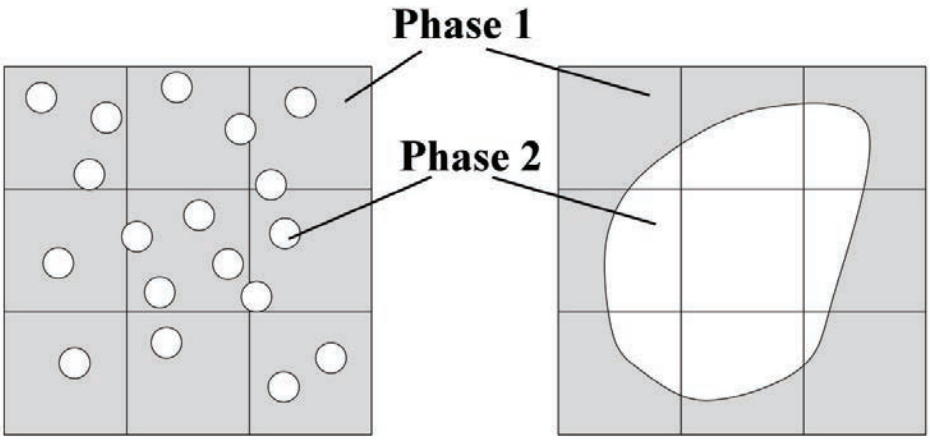

Fig. 1. Grid suitability for the VoF interface representation.

To accurately represent the interface, it has to be fully included in one cell, i.e. between two cells with volume fractions equal to 1 and 0 respectively just one cell has to be present: this is necessary to limit the numerical diffusion during the simulation.

A further requirement for an accurate tracking of the interface shape derives from the use of an appropriate time step during the simulation: sloshing problems can be analyzed only by transient approaches, and the Courant-Friedrichs-Lewy Number (CFL) should generally be lower than 1. As well-known, the CFL non-dimensional number is defined for a simplified one dimensional problem as follows:

$$CFL = \frac{u \cdot \Delta t}{\Delta x} < 1 \tag{3}$$

where u is the flow velocity, Δt is the computational time step and Δx is the cell size. To further strengthen the above

condition, in order to limit the blending of the solution when large time variations of the free surface shape are present, the CFL computed at the interface should be less than 0.5.

4. The methodology

As it will be explained in the following paragraphs, the usual CFD procedure to correctly perform sloshing simulations requires a very fine computational grid, able to accurately describe the free surface in terms of both shape and position. This approach, however, implies very long calculation times and high computational costs: because of the fluid motion, the cell dimension should be constantly fine all over the computational domain, i.e. all over the oil tank, even if the interface occupies just a small portion of the domain for most of the time.

As an alternative, Adaptive Mesh Refinement (AMR) techniques might be used, in which the computational mesh is locally refined according to user-specified threshold criteria: for the specific case, the presence of the liquid/gas interface might be used. Still, mesh adaptions should be specified at regular time-step intervals and in a "a posteriori" manner, which might imply loss of accuracy in-within two subsequent mesh update events and/or high computational times spent to generate the computational grid.

The developed methodology (which could be referred to as "Predictive mesh Refinement", PMR) aims at overcoming the above limitations. As stated earlier, a high density mesh is needed only where the interface is located, whilst a coarser mesh can be used far away from it. As a consequence, it is necessary to totally include the interface in a very fine mesh during the whole simulation; such refined mesh has to be generated so that it "forecasts" the free surface motion and position.

A procedure is therefore developed in order to dramatically reduce computational costs and times. The interface between the liquid and the gas is represented by the iso-surface of volume fraction equal to 0.5: once the interface is identified, it is translated along the three directions in space by a fixed length, to define a refinement threshold where the free surface will be reasonably still included for the subsequent time-steps, see Fig. 2. At every time step, the maximum velocity component of the previously defined iso-surface is considered, in order to estimate the distance the interface should have covered: if such distance is less than the thickness of the threshold, it means the interface is still included, whilst if the distance is close to that thickness, re-mesh is performed and a new refinement threshold is defined on the basis of the last position of the iso-surface.

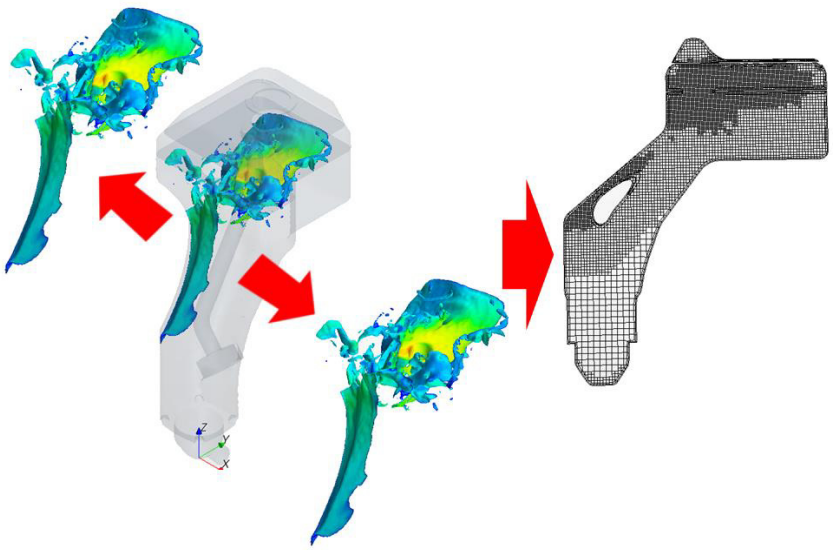

Fig. 2. Refinement threshold identification.

The procedure performs also a check on the simulation time step, so that the average CFL number is less than 1 all over the domain and less than 0.5 on the free-surface cell threshold, as required by the computational sub model used for the simulation. With reference to two subsequent time steps, defined as 0 and 1, it is possible to define the CFL number for both of them:

$$CFL_0 = \frac{u_0 \Delta t_0}{\Delta x_0} \qquad CFL_1 = \frac{u_1 \Delta t_1}{\Delta x_1} \tag{4}$$

If no remesh is performed from time-step 0 to time-step 1, the following identity can be written:

$$\Delta x_0 = \frac{u_0 \Delta t_0}{CFL_0} = \Delta x_1 = \frac{u_1 \Delta t_1}{CFL_1} \tag{5}$$

It is now possible to calculate time-step 1 as:

$$\Delta t_1 = \frac{u_0}{u_1}\frac{CFL_1}{CFL_0}\Delta t_0 \approx \frac{CFL_1}{CFL_0}\Delta t_0 \tag{6}$$

where the velocity *u* is assumed to be almost constant going from time-step 0 to time-step 1. The calculation of time-step 1 is performed twice, both on the whole computational domain, where CFL_0 is computed as a volume average while CFL_1=1 is imposed, and on the interface, where CFL_0 is computed as a surface average and CFL_1=0.5, and the minimum one is chosen to ensure the compliance both the numerical constraints over all the computational domain.

A schematic of the procedure is reported in Fig. 3.

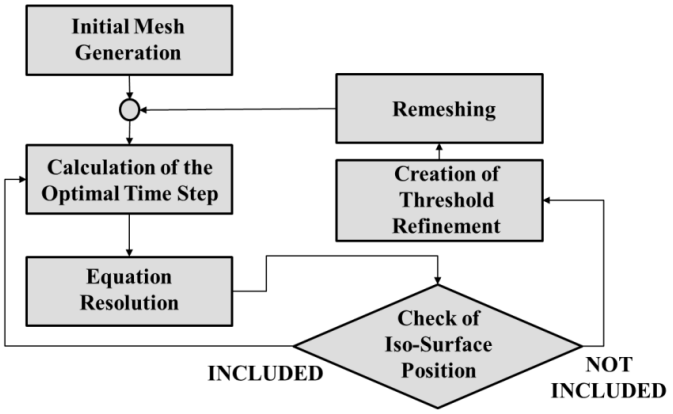

Fig. 3. PMR workflow

From a theoretical point of view, this procedure shows some advantages:

- the use of a variable time-step allows a reduction of the numerical diffusion during the simulation and a better tracking of the interface shape and position; moreover, time-steps can assume large values during relatively

"calm" portions of the simulation, while the choice of a constant value should be as conservative as to guarantee the CFL constraint based on the most critical situations;

- the PMR leads to a relevant decrease in the total number of cells, while simultaneously local grid refinements allows the representation of dispersed bubbles of the liquid phase inside the gas phase and vice versa.

When addressing the CFD problem from a more general perspective, it is necessary to consider the additional time spent to re-mesh the domain, which has to be taken into account to estimate the overall simulation time.

To evaluate the performance of the developed methodology, it is preliminarily applied to a simplified test case available in literature, where experimental measurements are performed [6]: the case consists of a tank filled of tap water representing a portion of a marine tanker, as represented in Fig. 4, whose sway motion along the direction, indicated by the arrow, is over-imposed to the displacement law.

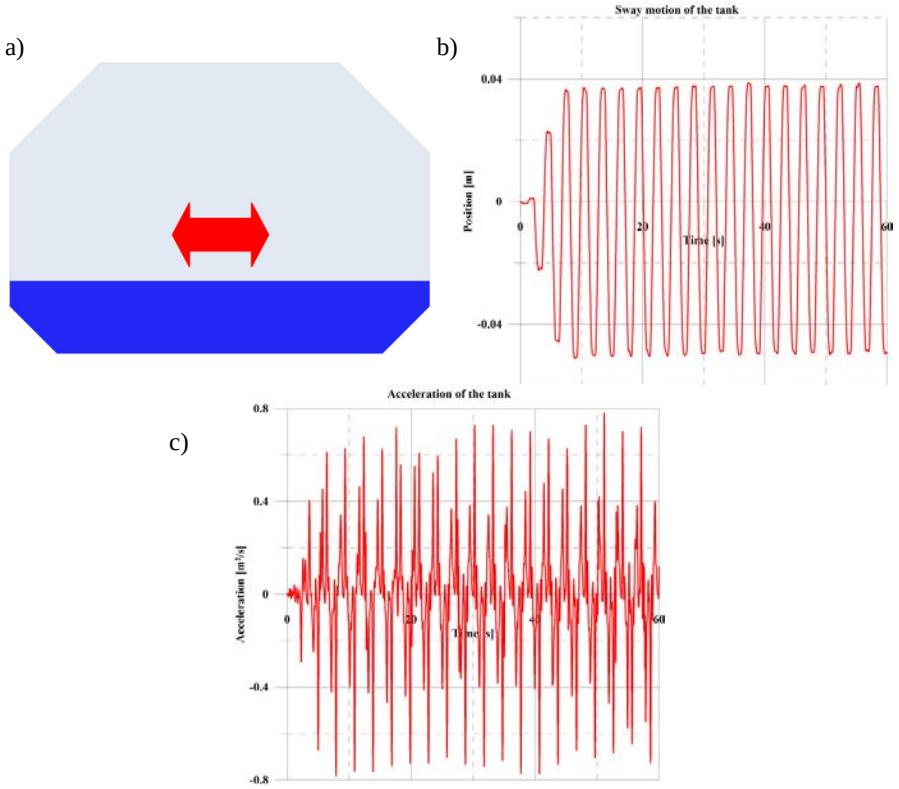

Fig. 4. (a) Preliminary test case; (b) sway motion vs. time; (c) tank acceleration vs. time.

5. Results

5.1. The simplified test case

As aforementioned, the proposed PMR uses a variable time-step and an automatic re-meshing procedure to better represent the motion of the liquid/gas interface. To evaluate the effectiveness of this approach, a comparison with a standard simulation, based on a constant time-step and a uniform grid, is performed. Results are also compared to experimental data: pressure is measured at some locations on the tank wall and the height of the free-surface is then estimated; in particular, data are available for the sensor depicted in Fig. 5.

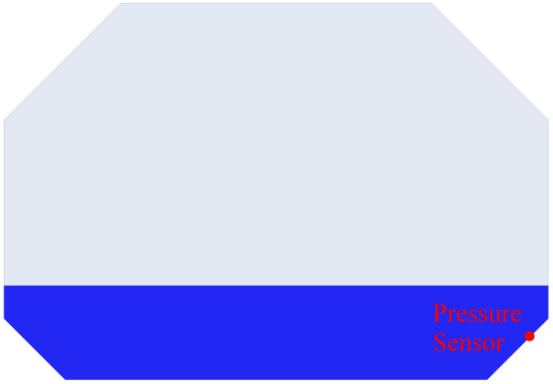

Fig. 5. Pressure sensor location.

In order to speed up the calculations and the mesh generation, a hexahedral grid is used for both the standard and the PMR case. A comparison between the constant mesh and the refined one is reported in Fig. 6: the cell dimension for the refinement threshold around the interface is purposely set equal to the cell size of the uniform grid; it is evident how the overall cell number can be strongly limited thanks to the proposed approach, without losing any details in the interface representation.

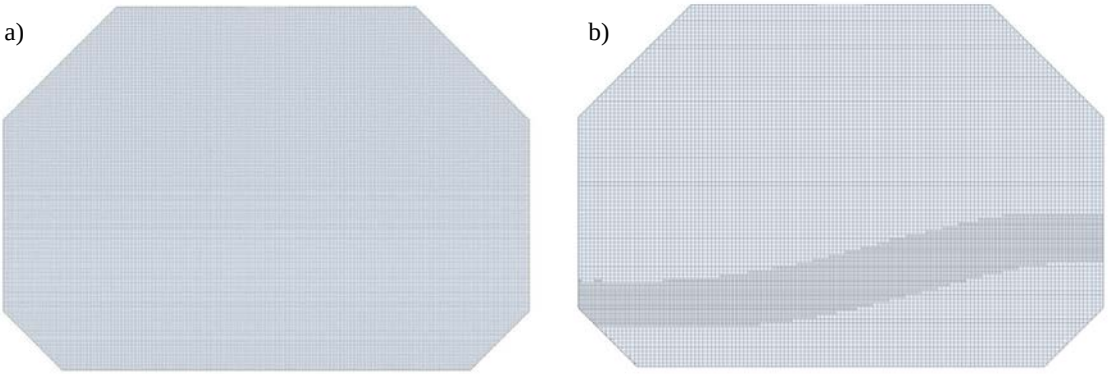

Fig. 6. (a)Standard (uniform) grid; (b) PMR grid.

Within the VOF framework, the CFD setup for all the simulations is based on a k-ε Two-Layer turbulence model. Because of buoyancy effects, gravity is taken into account and its magnitude and direction are modified by over-imposing the accelerations on the tank reported in Fig. 4(c).

Although the proposed methodology introduces a frequent re-meshing procedure, the total computational time is almost half that of the constant grid one, as it can be seen from the data reported in Table 1.

Table 1. Comparison between the fixed mesh and the dynamically refined one

Mesh	Standard	PMR
Minimum Cell Dimension	15 mm	15 mm
Number of re-meshings	0	373
CPU time	35.8 h	13.5 h
Time for remeshing	0 h	3.8 h
Total computational time	35.8 h	17.3 h

A good agreement emerges from the comparison of the experimental pressure trace with the calculated ones, see Fig. 7: no significant deviations can be seen between the standard CFD approach and the enhanced one proposed by the authors, as it can be seen in the following graph.

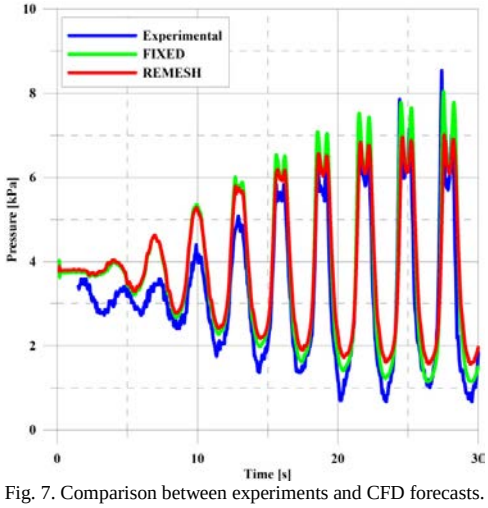

Fig. 7. Comparison between experiments and CFD forecasts.

The PMR methodology can therefore be considered effective to significantly reduce the computational time and costs of such liquid sloshing simulations as well as to track the motion and the shape of the liquid-gas interface; it is then applied to the actual geometry of a tank for the lubricating circuit of an internal combustion engine for high performance car applications.

5.2. Analysis of the lubricating oil tank

As stated earlier, given the promising results (in terms of trade-off between accuracy and computational costs) of the proposed PMR methodology, tank sloshing of lubricating oil is analyzed under the actual accelerations/decelerations of a sport car during typical racetrack maneuvers.

In particular, two different portions of the circuit are investigated, and are visible in Fig. 8: the former corresponds to the maximum left-to-right change in lateral acceleration (marked as “4” and “5”), while the latter corresponds to the maximum acceleration magnitude (marked as “6”).

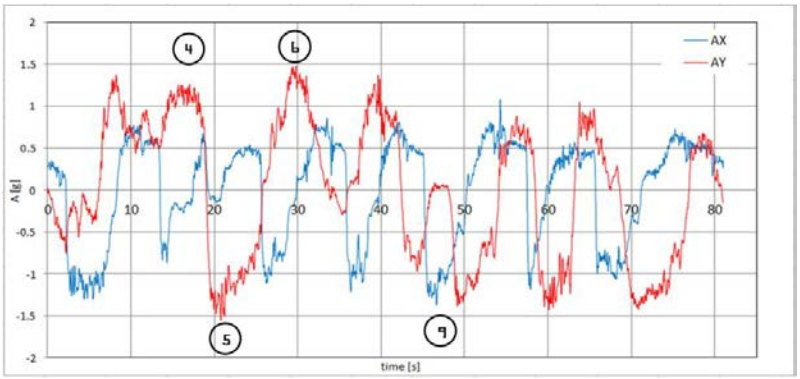

Fig. 8. Racetrack accelerations vs. time.

The computational domain covers the whole oil tank from the air/oil separator to the lubricating pump feeding section, as visible in Fig. 9. The resulting computational domain is made up of nearly 800.000 cells for the standard (fixed) mesh, while for the proposed PMR approach the cell number ranges from the initial 386.000 cells to a maximum of 770.000 cells during the simulation.

For the sake of brevity, only the maneuver “4/5” is here discussed. A first comparison between the two methodologies is performed at a macroscopic level, i.e. the liquid/gas interface behavior is analyzed to see if differences can be noticed. Two screenshots are reported, one for each method, in Fig. 9, at a time interval corresponding to maximum tank acceleration and corresponding iso-surface deformation: as visible, no noticeable differences can be seen, and in both cases the pump feeding section seems to be completely surrounded by the liquid phase.

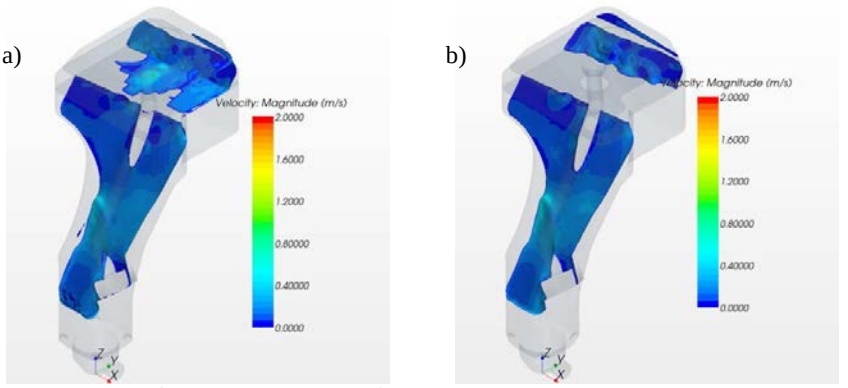

Fig. 9. Iso-surface comparison at max. deformation. (a) standard approach; (b) PMR approach.

A second visual comparison is then performed on the numerical behavior of the two simulations: the VoF scalar field evolution is reported in Figg. 10 and 11, once again at relevant tank acceleration/iso-surface deformation time intervals and for the two CFD approaches. Given the assumption of immiscible fluids, a clear separation between the two fluids should be seen in the domain, i.e. cells should be either blue- or red-colored, the remaining color range highlighting regions of high numerical diffusivity and related solution inaccuracy. As visible, the combination of the PMR and the *CFL* sensitive time-step approach allows the separation between the two fluids to be much clearer than for the standard approach.

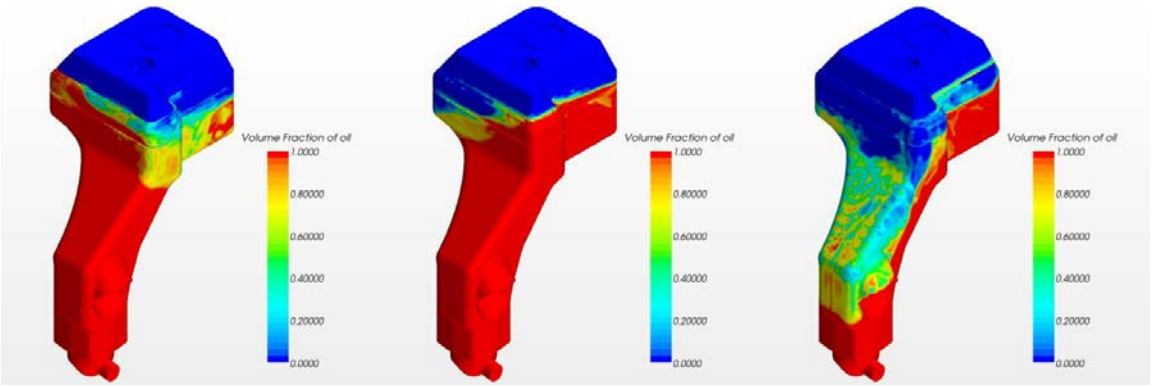

Fig. 10. VoF temporal evolution: standard approach.

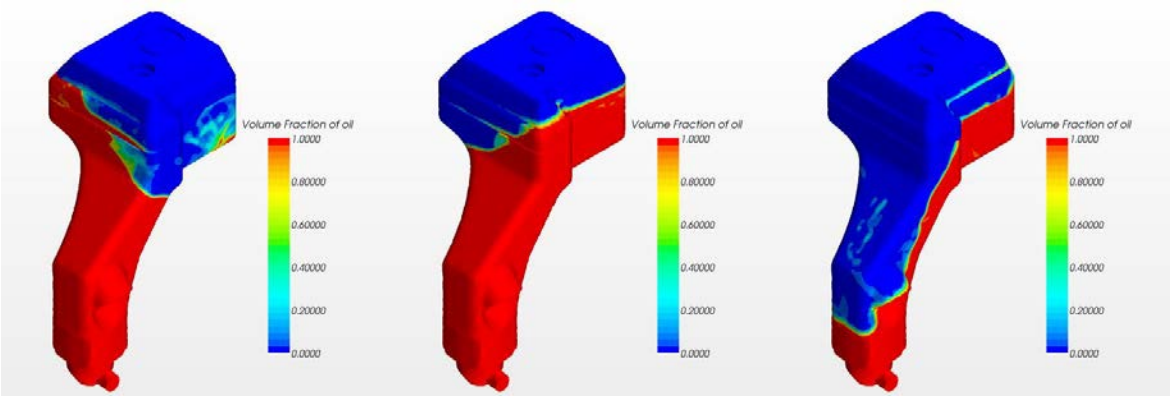

Fig. 11. VoF temporal evolution: PMR approach.

Results are even more promising when the comparison is extended to the analysis of the overall computational cost of the simulations, which is reported in Table 2: the use of the PMR approach more than halves the computational time required for the simulation of the full "4/5" maneuver.

Table 2. Comparison between the standard and the PMR approach

Mesh	Standard	PMR
Minimum Cell Dimension	2.8 mm	2.8 mm
Number of remeshes	0	119
CPU time	193.9 h	80.0 h
Time for remeshing	0 h	3.9 h
Total computational time	193.9 h	83.9 h

6. Conclusions

The paper reports a numerical activity aiming at improving the computational efficiency of 3D-CFD sloshing simulations. A Predictive Mesh Refinement method is proposed which efficiently combines time-step sensitivity to local flow conditions and liquid/gas interface motion.

The methodology effectiveness is at first evaluated for a simple geometry, for which experiments are available, and it proves to allow the computational cost to be strongly reduced without loss of accuracy. Secondly, a lubricating oil tank of a sport car is analyzed under actual racetrack acceleration. Once again, the PMR approach is able to both reduce numerical diffusion issues and computational demand.

References

[1] M. Moradi Larmaei, Tew-Fik Mahdi. Simulation of shallow water waves using VOF method. Journal of Hydro-environment Research; Volume 3; Issue 4; March 2010; p. 208-214, ISSN 1570-6443, http://dx.doi.org/10.1016/j.jher.2009.10.010.

[2] Afkhami S, Zaleski S, Bussmann M. A mesh-dependent model for applying dynamic contact angles to VOF simulations. Journal of Computational Physics; Volume 228; Issue 15; 20 August 2009; p. 5370-5389, ISSN 0021-9991; http://dx.doi.org/10.1016/j.jcp.2009.04.027.

[3] Fontanesi S, Giacopini M, Cicalese G, Sissa S, Fantoni S. Numerical investigation of the cavitation damage in the wet cylinder liner of a high performance motorbike engine. Engineering Failure Analysis; Volume 44; September 2014; p. 408-423; ISSN 1350-6307; http://dx.doi.org/10.1016/j.engfailanal.2014.05.025.

[4] Brackbill JU, Kothe DB, Zemach C. A Continuum method for modeling surface tension. J. Comp. Physics; 100; p. 335-354 (1992).

[5] Muzaferija S, Peric M. Computation of free surface flows using interface-tracking and interface-capturing methods. Nonlinear Water Wave Interaction; Chap. 2; Computational Mechanics Publications in O. Mahrenholtz and M. Markiewicz (eds.); WIT Press; Southampton; 1999

[6] Iwanowski B, Lefranc M, Wemmenhove R. Numerical Simulation of Sloshing in a Tank, CFD Calculations Against Model Tests. Paper OMAE2009-79051; p. 243-252; doi:10.1115/OMAE2009-79051