

# NUMERICAL SIMULATION OF SLOSHING DYNAMICS IN A TANK

Sorin DIACONU

University “Dunarea de Jos” of Galati, Mechanical Engineering,  
Applied Mechanics Department, ROMANIA  
sorin.diaconu@ugal.ro

## ABSTRACT

*The present work describes a study which describes the effect of sloshing liquid in the LNG tanks. More simulations have been made by a CFD code which is based on Navier-Stokes equations and uses two phase flow effects. These phenomena are important for sloshing in the tanks and the complex mixture of the liquid and gas phase around the free surface involves a challenge to numerical simulation. Moreover, the two phase flow is strongly affected by both the filling amount of the tank and the irregular motion of the tank due the offshore conditions.*

*Numerical results are presented from large scale of an LNG carrier filled 10% with liquid and compressible gas, which shows both fluid height and fluid pressure exerted on tank walls. This paper presents only part of the analyzed results, respectively, four pressure points and three water height probes, a regular sway motion being considered.*

**Keywords:** sloshing, numerical simulation, two-phase flow, hydrodynamic wave loading

## 1. INTRODUCTION

The evolution and growth of the LNG (Liquefied Natural Gas) transport in partially filled tanks lead to increased demand to have accurate methods to predict the fluid behavior in these sloshing tanks and the effect of the fluid on the tanker motion.

During different types of weather conditions, on the LNG tanks walls appear various pressure loads. The motion of the inside fluid is normally restricted by the loading conditions of the vessel, ie either the tank is nearly empty or full.

The motion of LNG inside the tanks is important to be investigated, especially during more violent weather conditions, when appear the interaction between ship movement and the motion of LNG inside tanks.

The fluid distribution in the tanks strongly depends on both the tank filling ratio and the weather conditions the tanker is subjected to. So the inside fluid from these tanks are generally a complex mixture of different fluids, with a strong mixing and the fluid distribution strongly depends on both the amount of filling and the influence of the weather conditions. In this condition the LNG fluid is more likely to be induced into resonance due

to wave action and roll motions. This sloshing behavior of the fluid will lead to high impact pressures.

Along the time there were numerous accidents, the first documented LNG rollover incident has been the LNG ship Esso Brega, where the tank developed a sudden increase in pressure and the tank roof slightly damaged.

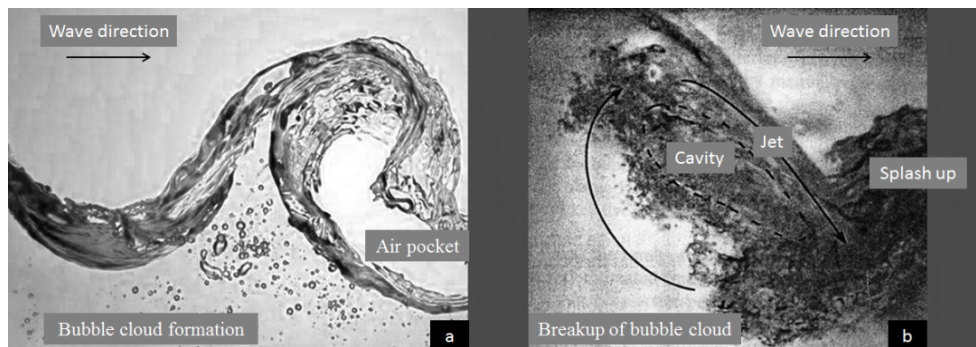
Therefore, to these types of tanks must be given a special attention at their construction and design, because they store large amounts of liquid over long distances. At the same time special attention must to be paid on various tests to estimate the tank liquid levels, temperatures, pressures, etc.

The sloshing effects which are present especially in LNG tanks can be quite detrimental. Abramson et al [1] presented a study where all kinds of damages due to sloshing are described.

It is recommendable to be used a two-phase flow model when flow conditions are getting more violent, spatial and temporal scales of entrapped/entrained air in the flow being a serious problem in simulations. Scardovelli and Zaleski [5] give a suitable overview of existing two-phase flow models and in the last years have been recorded a significant progress in the simulation of two-phase flows [1], [4], [8].

The application of a two-phase flow model for wave impact is especially useful for the simulation of breaking waves, but also for the simulation of other free surface. The effect of the two-phase flow phenomena on the water level around the loading on offshore structures is worthwhile to investigate and the effect of air pockets on the overall hydrodynamics can be evaluated for the wave impact on a vertical wall.

In Fig. 1 a number of two-phase flow phenomena are sketched that occur around a breaking wave.



**Fig. 1.** Schematization of two-phase flow phenomena for a breaking wave: a) an air pocket is enclosed b) the air pressure decreases, resulting in breakup of the pocket and bubble entrainment

The pressure level variation during a violent wave impact is determined by both the impact pressure and the smaller reflecting pressure peak. The reflecting pressure peak is lower, but longer-lasting than the impact pressure peak.

The present paper shows the main aspects of the governing equations of the ComFlow model and in the last part the results of two-phase numerical simulations of sloshing in a tank are presented.

## 2. GOVERNING EQUATIONS

The fluid flow is governed by the Navier-Stokes equations presented in integral form for an arbitrary control volume  $\Omega$  with boundary  $\Gamma$ , the equations being natural because are discretised with the finite volume method.

First, conservation of mass results from the equation

$$\oint_{\Gamma} \mathbf{u} \cdot \mathbf{n} \cdot d\Gamma = 0 \tag{1}$$

which is commonly known as the continuity equation for incompressible flow.

The symbols are the velocity in three dimensions  $\mathbf{u} = (u, v, w)^T$  and outward-pointing normal vector  $\mathbf{n}$  of  $\Gamma$ .

Applying the conservation of momentum results in the system of equations

$$\frac{d}{dt} \int_{\Omega} \mathbf{u} \cdot d\Omega + \oint_{\Gamma} (\mathbf{F}_{inv} - \mathbf{F}_{vis}) \cdot \mathbf{n} \cdot d\Gamma = \int_{\Omega} \mathbf{g} \cdot d\Omega \tag{2}$$

The matrices  $\mathbf{F}_{inv}$  and  $\mathbf{F}_{vis}$  are the inviscid and viscous flux through the boundary of the control volume. The viscous flux has a diffusive term. The inviscid flux has a convective and a pressure term.

$$\mathbf{F}_{inv} \equiv \mathbf{u} \otimes \mathbf{u}^T + \frac{1}{\rho} p \mathbf{I} \tag{3}$$

$$\mathbf{F}_{vis} \equiv \frac{\mu}{\rho} \nabla \mathbf{u} \tag{4}$$

The vector  $\mathbf{g}$  denotes the acceleration due to body forces, which are only gravitational forces in the problems considered, pressure  $p$ , the density  $\rho$ , dynamical viscosity  $\mu$ , the  $3 \times 3$  identity matrix  $\mathbf{I}$  and the vector tensor product  $\otimes$ .

### 3. NUMERICAL MODEL

The discretisation of the mathematical model is presented in two dimensional terms, to avoid complex and unintelligible notations and takes into account moving bodies and cut cells. The extension to three dimensions is straightforward. After the definition of the grid and geometry, the discretisation is given in space, time and with special attention to the free surface. In the closing section some attention is given to the cut cell method in three dimensions and to the discretisation of the density. Given the staggered arrangement of grid variables, pressure and density are both located in the cell centers, like in Fig. 2.

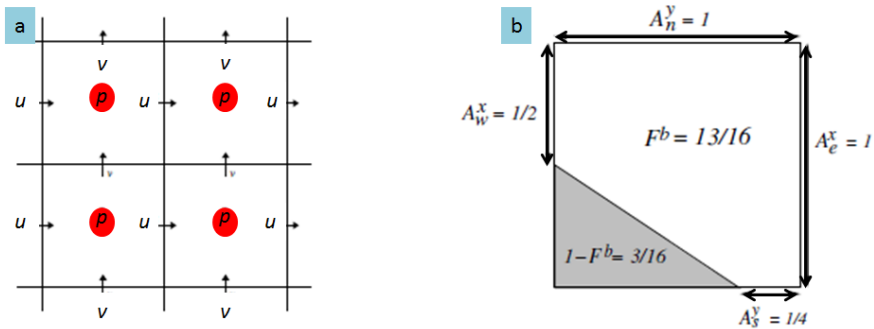


Fig. 2. Staggered arrangement of the velocity components and pressure (a) and an illustration of volume and edge apertures (b)

The numerical model has been implemented in a 3D VOF Navier-Stokes solver called COMFLOW and local height function improves the treatment of the free surface. The VOF function  $F_s$  determines whether or not the flow field in a grid cell is calculated, in contrast

with the two-phase approach, where the liquid-gas interface is no longer considered a free surface.

Before describing the discretisation of the equations, it is worthwhile to describe the cell labeling first. The variable  $F_b$  describes the fraction of a grid cell open for fluid, while the variable  $F_s$  describes the fraction of a grid cell filled with the liquid phase. So, the Navier-Stokes equations are solved in grid cells and to every grid cell is given a label (geometry and fluid cell labels) to distinguish between boundary, air and fluid.

Figure 3 represents an example of geometry cell labeling and free-surface cell labeling for wave impact on a rectangular body.

#### 4. SLOSHING SIMULATION OF A LNG TANK MODEL

The sloshing tank model was based on an LNG tank, with the width of the tank of 3.790 m and a height of 2.590 m.

A full grid convergence study has been performed for the lowest filling rate of 10% and the oscillation rate is 3.48. The main dimensions of the tank are presented in Fig. 4.

In the simulations have been analyzed 12 water height lines and fluid pressure at 4 points on the right side of the tank. Hereafter, the computational results will be reported for fluid height probe H1, H7 and H12. These control locations and the fluid level are also clearly marked in Fig. 5.a and the representation of the free surface in a low filling ratio sloshing computation, Fig. 5.b. For sloshing calculation has been designed a number of 8100 cells, all grids are uniform, with the same grid spacing in both computational directions.

The origin of the coordinate system is chosen at the center of the tank bottom, the motion of the tank is considered as a moving coordinate frame in the simulations, tank is subject to a regular sway motion which is registered in time, the oscillation period being of 10 s. Although the amplitude of the sway motion is less than 10 cm, a strong sloshing motion of the fluid is induced by the tank oscillation. In particular at the side walls, the water height strongly increases during the impact of the sloshing liquid, up to about one meter.

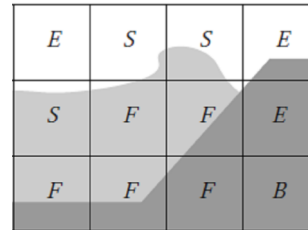


Fig. 3. Cell labelling: B(oundary), E(mpty), S(urface) and F(luid) cells

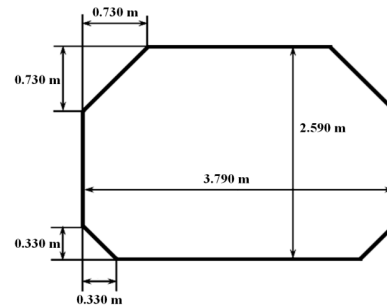


Fig. 4. Main dimensions of the LNG tank

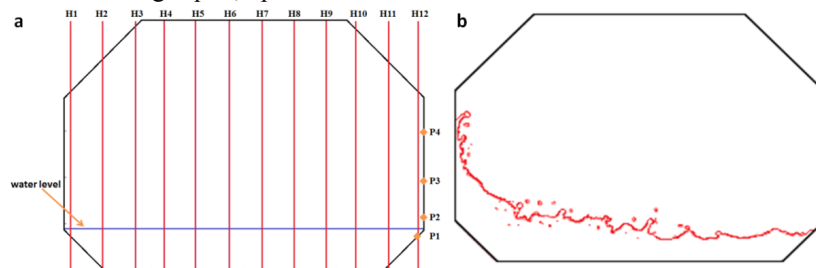
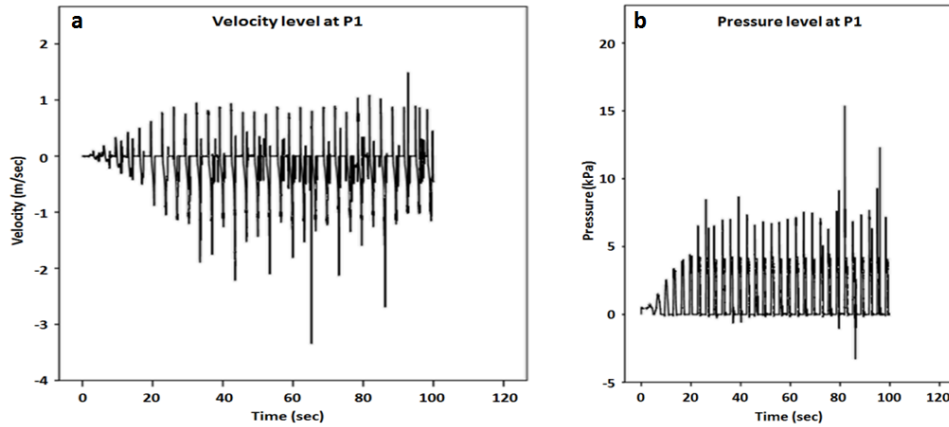


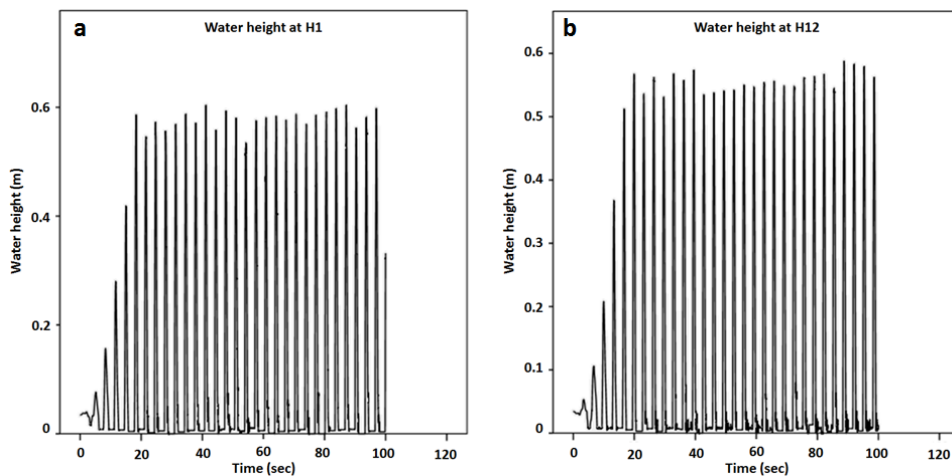
Fig. 5. Position of 4 pressure points and 12 water high analyzed (a) and the effect of the gravity-consistent density averaging in a sloshing simulation (b)



**Fig. 6.** Velocity evolution at P1 (a) and pressure level development for 10 percent filling-ratio at P1, at the side wall (b)

Fig. 6 shows the velocity (a) and pressure (b) development registered at P1, the lowest position analyzed of the right side wall.

Fig. 7 shows the development of the water height at H1 and H12 position, which is located at the left side and right side of the tank, respectively.



**Fig. 7.** Water height development for 10 percent regular motion test at probe H1 (a) and H12 (b), near the side tank wall

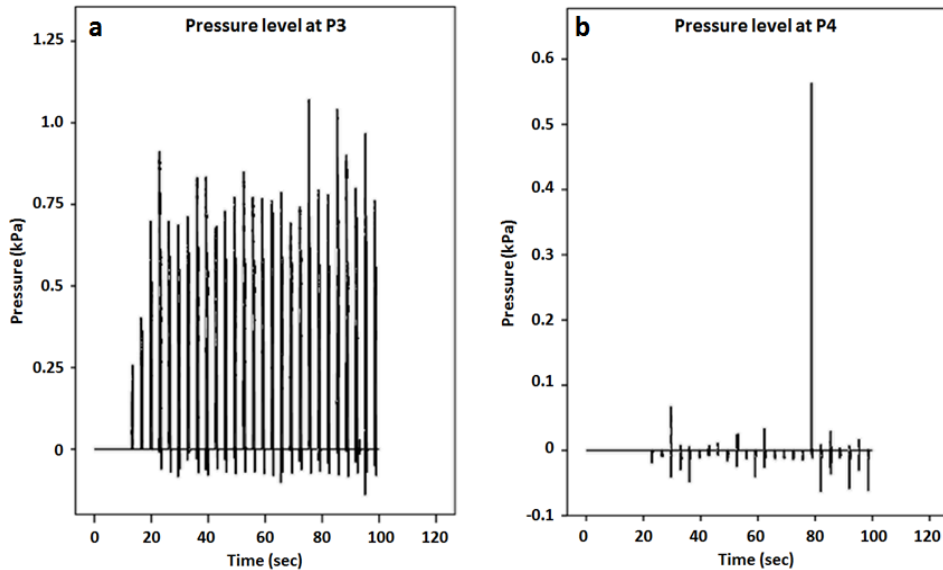
The development of the pressure level is important to predict the forcing on the tank walls due to the sloshing liquid. Fig. 8 shows the pressure development for position P3 (a) and P4 (b).

It can be noticed that in these figures the pressure level is very low and especially at P4, where only a thin liquid layer moves upwards along the side wall during the impact of the sloshing liquid, and the upward movement of this jet occurs very rapidly.

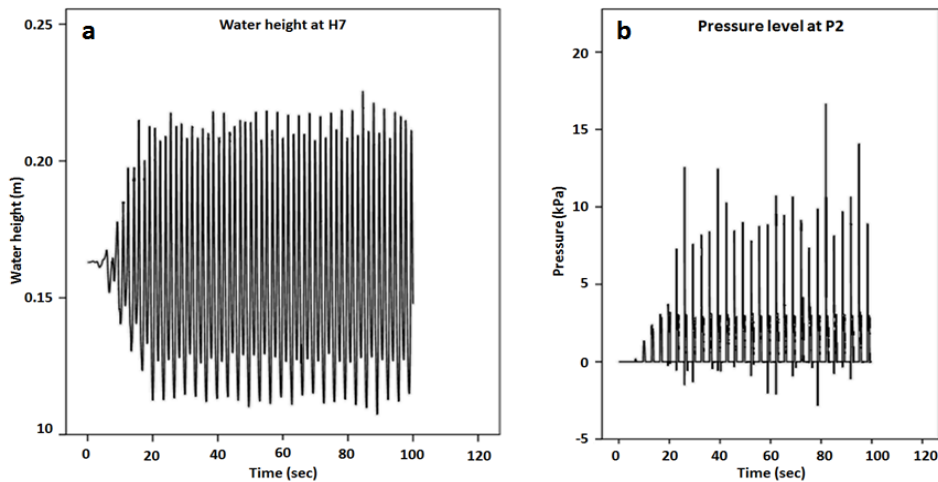
Regarding the water height development around the middle of the tank in Fig. 9 (a), we can see that the differences are more constant and the fluid height level is very low in comparison with the levels from the left and right side of the tank walls. Also the higher

development of pressure was registered at the points which are closer to the fluid, P1 (Fig. 7b) and P2 (Fig. 9b).

For the two-phase simulations, another issue to keep in mind is the first-order upwind spatial discretisation, which adds artificial viscosity to the fluid. Using a less dissipative second-order upwind discretisation may reduce the numerical damping of the sloshing liquid.



**Fig. 8.** Pressure development for 10% regular sway test at pressure point P3 (a) and P4 (b)



**Fig. 9.** Water height development for 10 percent regular motion test at probe H7, near the center of the tank (a) and pressure development at P2 (b)

## 5. CONCLUSIONS

The hydrodynamics of different offshore applications like sloshing dynamics in an LNG tank can be simulated numerically using ComFLOW model, which is based on an

iVOF Navier-Stokes solver. This paper shows the results of the numerical simulations of the compressible two phase model.

Special attention has been given to the calculation of the free surface density and at the same time to avoid spurious velocities and the pressure calculation to be precised, more caution must be given to the calculation of density at the cell extremities.

The dynamics of the air phase are compressible, the air being subjected to adiabatic compression and expansion. In this paper has been presented some results of a possible event that can occur by combining the calm weather conditions with low tank filling ratios. The results can be considered that give a good prediction of water heights and pressure, but for an exact assessment the numerical results must be compared with experimental data.

## 6. ACKNOWLEDGEMENT

The work of the first author has been made in the scope of the project SOP HRD - EFICIENT 61445/2009 (Management System for the Fellowships Granted to the PhD Students).

## 7. REFERENCE

- [1] **Abramson H.N., Bass R.L., Faltinsen O., Olsen H.A.**, 1974, Liquid slosh in LNG carriers The 10th Symposium on Naval Hydrodynamics, pp. 371-388.
- [2] **Scardovelli R., Zaleski, S.**, 1999, Direct Numerical Simulation of Free-Surface and Interfacial Flow, *Annual Review of Fluid Mechanics*, 31, pp. 567-603.
- [3] **Wemmenhove R., Luppès, R., Veldman A.E.P., Bunnik T.**, 2007, Numerical Simulation of Sloshing in LNG Tanks with a Compressible Two-Phase Model, *26th Offshore Mechanics and Arctic Engineering Conference*, paper 29294, San Diego, USA.
- [4] **Kleefsman K.M.T., Fekken, G., Veldman A.E.P., Buchner B., Iwanowski B.**, 2005, A Volume-Of-Fluid Based Simulation Method For Wave Impact problems, *Journal of Computational Physics*, 206, pp. 363-393.
- [5] \*\*\* Ship Structure Committee, 1980, Evaluation of Liquid Dynamic Loads in Slack LNG Cargo Tanks (SSC- 297).
- [6] **Wemmenhove R., Loots, G.E., Luppès R., Veldman A.E.P.**, 2005, *Modeling two-phase flow with offshore applications*, 24th Offshore Mechanics and Arctic Engineering Conference, paper 67460, Halkidiki, Greece.
- [7] **Hirt C.W., Nichols B.D.**, 1981. Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries, *Journal of Computational Physics*, 39, pp. 201–225.
- [8] **Veldman A.E.P., Gerrits J., Luppès R. Helder, J.A.**, 2007, Vreeburg, J.P.B. The numerical simulation of liquid sloshing onboard spacecraft, *Journal of Computational Physics*, 224, pp. 82-99.
- [9] **Peric M., Zorn T., El Moctar O., Schellin T.E, Kim Y-S.**, 2007, Simulation of Sloshing in LNG-Tanks, *Proceedings of 26-th International OMAE Conference*, San Diego, CA. Paper OMAE2007-29555.