



25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-1885

CFD ANALYSIS FOR THE SLOSHING PHENOMENA OF SPRAY MIXTURE TANK

Gustavo Barbosa Micheli
Marcelo Luiz de Freitas Fogal
Vicente Luiz Scalon
Alcides Padilha

Depto. de Engenharia Mecânica - Universidade Estadual Paulista Julio de Mesquita Filho - FE/UNESP, Bauru, SP, Brasil
gustavobm.micheli@gmail.com
marcelo.fogal@hotmail.com
scalon@feb.unesp.br
alcidespadilha123@gmail.com

Abstract. *The sloshing phenomena in a spray mixture tank is evaluated in a numerical analysis with results of some computer simulations of fluid dynamics (CFD). The computation of the effects of sloshing due to the braking deceleration of an agricultural vehicle is carried out in three levels of tank filling (25%, 50% and 75%) and without mechanical agitation, in order to define the most critical conditions for the structural dimensioning of the tank. It is evaluated pressure time histories against tank wall using a multiphase transient model (water and air-ideal gas) with a free surface flow in a homogenous model. Thus, we could verify the wave's performance of sloshing, and the condition of higher tank filling volume (75%) was the one that generated the highest-pressure values on the tank wall. The tank level of 50%, achieved higher-pressure oscillation frequency.*

Keywords: *sloshing, CFD, free surface, spray tank*

1. INTRODUCTION

It is understood as sloshing to the movements of the fluid in a container where there are free surface effects. A partially filled tank may experience sloshing under various circumstances, including it may be a resonance phenomenon, where the free surface can deform a lot, this because the liquid increases in amplitude along the sidewalls, even reaching the upper surface of the compartment. For agricultural machines, this effect is present both in the tanks of spray mixture for agricultural implements coupled to tractors and to self-propelled machines, which also have fuel tanks. The effects of sloshing are the increase of the dynamic loads in the walls of the tanks, damages by impact, besides exciting other neighboring structures influencing even in the comfort of the operator (Xue et al., 2017).

There are several design devices and solutions that minimize the sloshing effects that can be adopted for the tanks. One of the solutions is the use of bulkheads that divide the tank into smaller tanks by shifting the natural frequency of sloshing of the frequency range of critical excitations. However, the amount of material required for such structures is high, increasing their weight and cost.

In order to reduce costs with prototypes of the tanks, it is possible to use the numerical simulation of sloshing and the phenomena involved in the agitation, being obtained with the aid of a computational fluid dynamics (CFD) code, which in a three-dimensional analysis uses the finite volume technique as a method of discretization of the fluid domains. A major advantage of the CFD technique is its flexibility to change process parameters, flow regimes, location of wave break geometries, location of agitators, and reservoir geometry without the need to perform a full-scale prototype.

The behavior of confined water and airflows in the tank is extremely complex especially when the tank is partially filled (Hasheminejad and Aghabeig, 2011). The effect of anti-slosh baffles on free liquid oscillations depends on the baffle installation position that has distinct effects on the sloshing frequencies and the associated hydrodynamic pressure fields. For this, the phenomenon is investigated following the methodology based on the conservation of the volumetric fraction to represent the free surface.

Micheli et al. (2015) conclude that in the case of spray mixture tanks, the efficiency of the homogenization of the mixture inside the tank has a great impact on the quality of the application and the treatment of a crop, and can be hampered by the introduction of anti-sloshing devices.

The present work intends to study the numerical simulation of CFD applied to the problem of sloshing in a spray mixture tank of an agricultural sprayer.

2. NUMERICAL SETUPS

2.1 Sloshing problem description

In this work, the effect of sloshing on a 4500 liters tank of an agricultural sprayer for non-perennial crops (such as soybean, corn and wheat) shown in Fig. 1 is being evaluated. In addition, it is also possible to note the position of the pressure capture points on the tank wall for sloshing effects comparison in Fig. 2.



Figure 1. Mesh for 4500 liters sprayer tank domain.

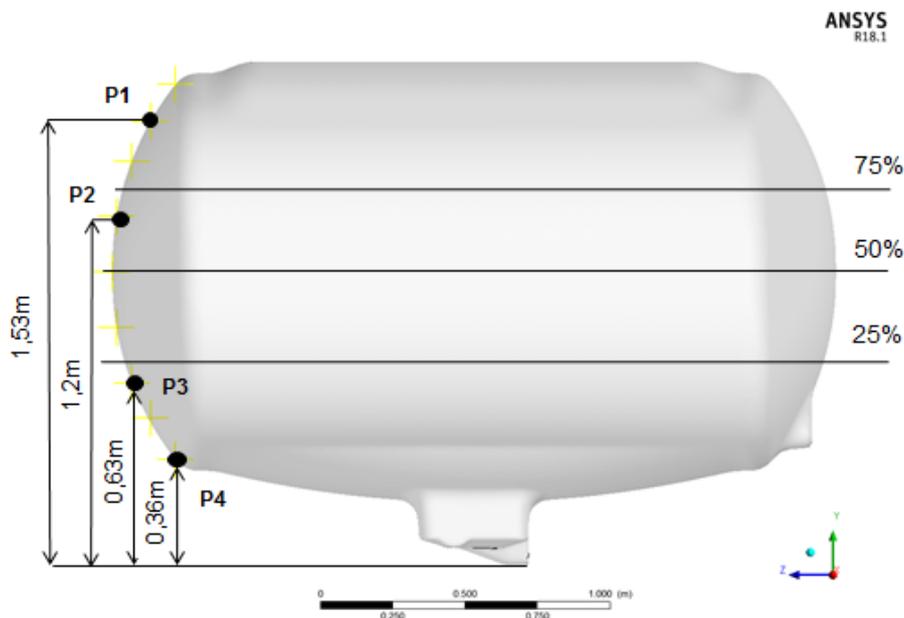


Figure 2. Positions of levels of tank filling and pressure capture points.

The evaluation of the effects of sloshing due to the braking deceleration of the vehicle (0.3g to 0.1g deceleration in z-direction) is carried out in three levels of tank filling (25%, 50% and 75%) and without mechanical agitation, in order to define the most critical conditions for the structural dimensioning of the same.

It is not clear what is more striking for the dimensioning of the tank walls between the tank's higher level of filling (larger spray mixture volume) or the volume of air that allows greater displacement of the spray mixture volume.

2.2 Numerical methods

Numerical simulations were performed using the commercial CFD code, ANSYS CFX 18.1, and it was used non-structured tetrahedral meshes generated by software ICEM CFD 18.1. All of them were carried out in 3D, performed in parallel on an Intel (R) Core(TM) i7-6700HQ CPU @2.6 GHz processors with 16 GB RAM.

In pre-processing, the definition of the parameters of the models is briefly described as follows: homogenous multiphase model (free surface model), fluids defined as water (for spray mixture) and air (ideal gas), with reference temperature equal to 25 ° C; simulation in transient regime and turbulent; isothermal flow; surface tension coefficient equal to 0.072 N/m; no-slip walls. The turbulent mean field was obtained with the Reynolds Averaged Navier Stokes (RANS) and the turbulence model used to close the set of equations was the k-ε model of two equations. The numerical convergence adopted occurred when the sum of normalized residuals was lower than 1E-04 for all variables for each time step. The total simulation time was 6 seconds for all cases.

3. MATHEMATICAL MODEL

3.1 Transport Equations

The mathematical model used in this computational simulation is based on the mass conservation equation and the Navier-Stokes transport equations, which describe the phenomena of movement, energy and mass transport, described by Fogal et al. (2018).

The analytical solution of these partial differential equations is performed through the finite volume technique, where the differential equations are discretized and the resulting algebraic equations are solved iteratively for each unit of control volume. As a result, an approximation of the value of each variable to specific points across the domain can be obtained.

3.2 Turbulence modeling

In the present work, the standard k-ε model is employed for turbulence closure in the RANS approach. Within the framework of this model, the transport equations for k and its dissipation rate ε are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + P_{kb} \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho U_j \varepsilon) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 1} P_{\varepsilon b}) \quad (2)$$

where, ρ is the fluid specific mass; t is time; U_j is the mean flow velocity; μ_t is the eddy viscosity; P is the static pressure; k is the turbulent kinetic energy; ε is the dissipation rate of turbulent kinetic energy; P_{kb} e P_{εb} represent the body forces influence, P_k is the production of turbulent kinetic energy due to viscous forces. C_{ε1}, C_{ε2}, σ_k and σ_ε are empirical constants with values 1.44; 1.92; 1.0 and 1.3, respectively. This RANS k-ε model with above coefficients was used in the liquid sloshing studies in e.g. Rhee (2005), Godderidge et al. (2009) and Thiagarajan et al. (2011).

3.3 Free surface modeling

Free surface flow refers to a multiphase flow situation where the phases are separated by a distinct resolvable interface, as a homogeneous model. The volume fractions are 0 or 1 except near interface. The phases not mixed at microscopic scale and share the velocity field (U_i = U_k), essentially, a single-phase momentum equation with mixture density and viscosity (CFX, 2017).

4. RESULTS AND DISCUSSION

As results of the numerical simulation, it is possible to identify the formation of the free surface for the volume of water inside the tank by isosurfaces (volume fraction value = 0.5). Figures 3, 4 and 5 show the profile of the sloshing waves formed during the time of the deceleration imposed on the models, simulating the braking operation of an agricultural sprayer. It is possible to notice on the left-hand side the moment where there is the displacement of water on the tank front wall, and on the right-hand side on the opposite wall, for each tank level studied.

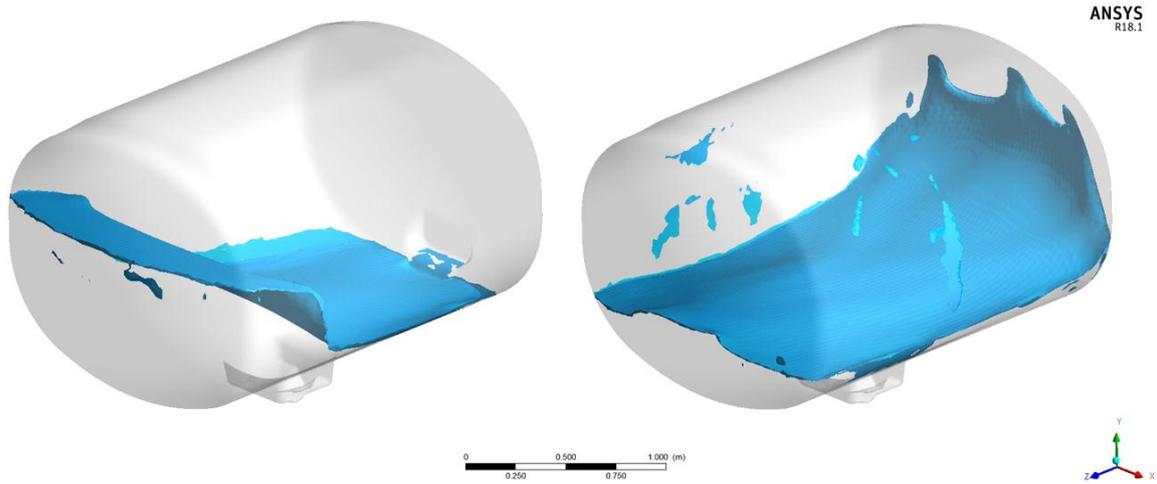


Figure 3. 3D visualization of sloshing in the sprayer tank at 25% of its capacity.

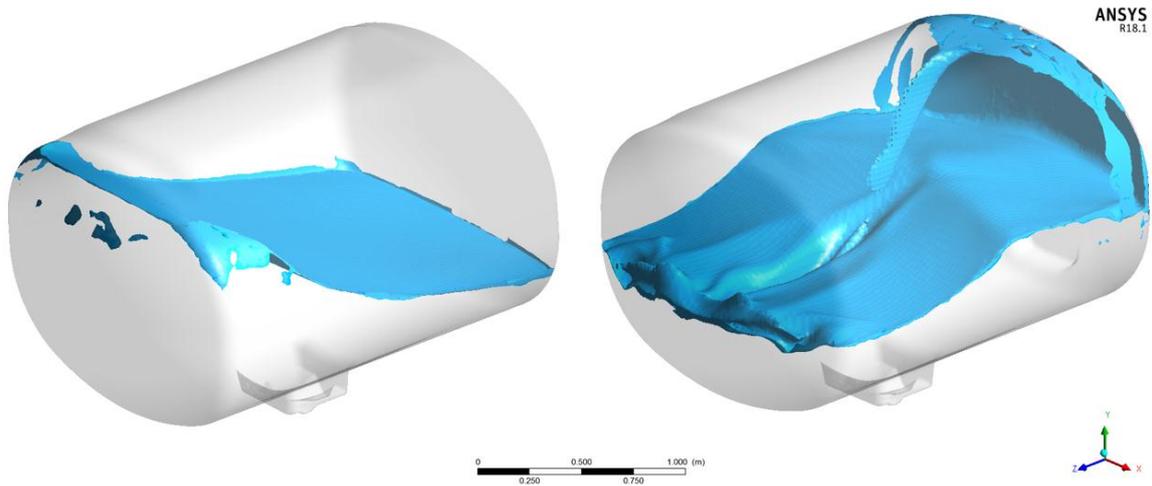


Figure 4. 3D visualization of sloshing in the sprayer tank at 50% of its capacity.

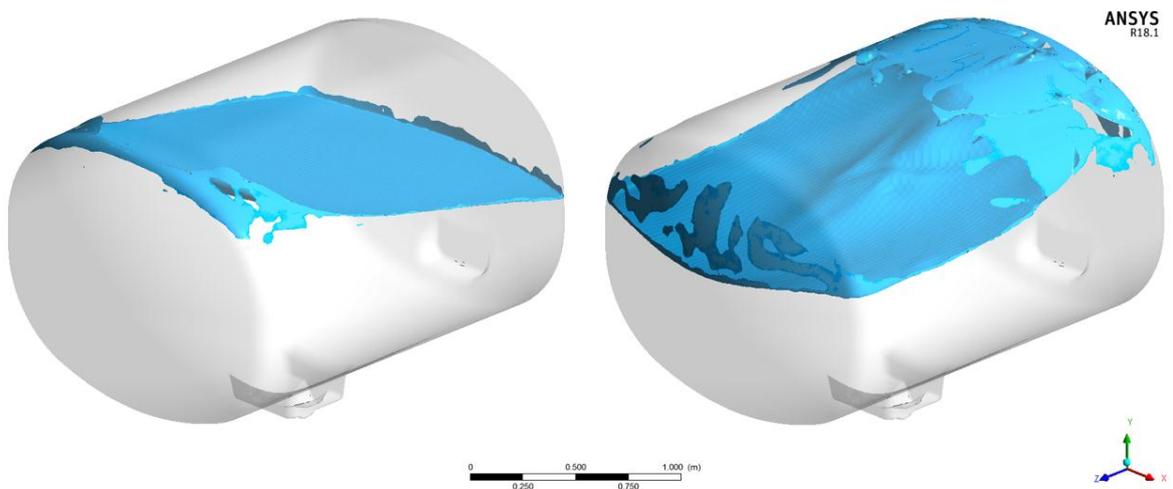


Figure 5. 3D visualization of sloshing in the sprayer tank at 75% of its capacity.

Figure 6 shows the simulation results of pressure capture for some points on the tank front wall according the positions indicated in Fig. 2.

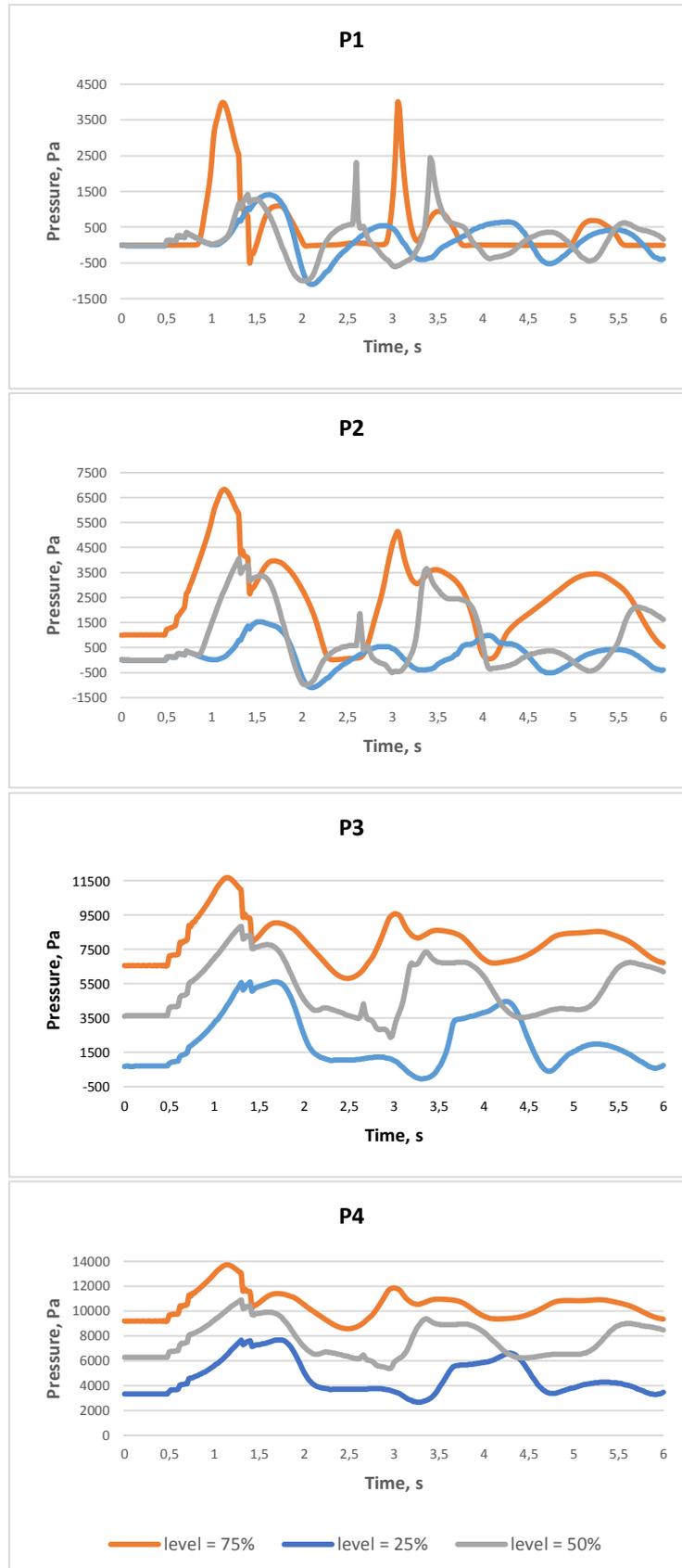


Figure 6. Pressure time histories in tank front wall.

The pressure curves in the front tank wall showed in Fig. 6 have similar shape for lower points (P3 and P4) and some differences for higher points (P1 and P2). It can be explained because P3 and P4 are submerged for all levels of tank simulated and the difference of pressure values are due to the difference of water columns under these capture points. On the other hand, for P1 and P2, there are differences too in the period of the pressure functions.

Figure 7 shows the profile of the sloshing waves for level 75% of tank capacity, formed in some periods of time that occurs peaks of pressure capture according indicated in Fig. 6.

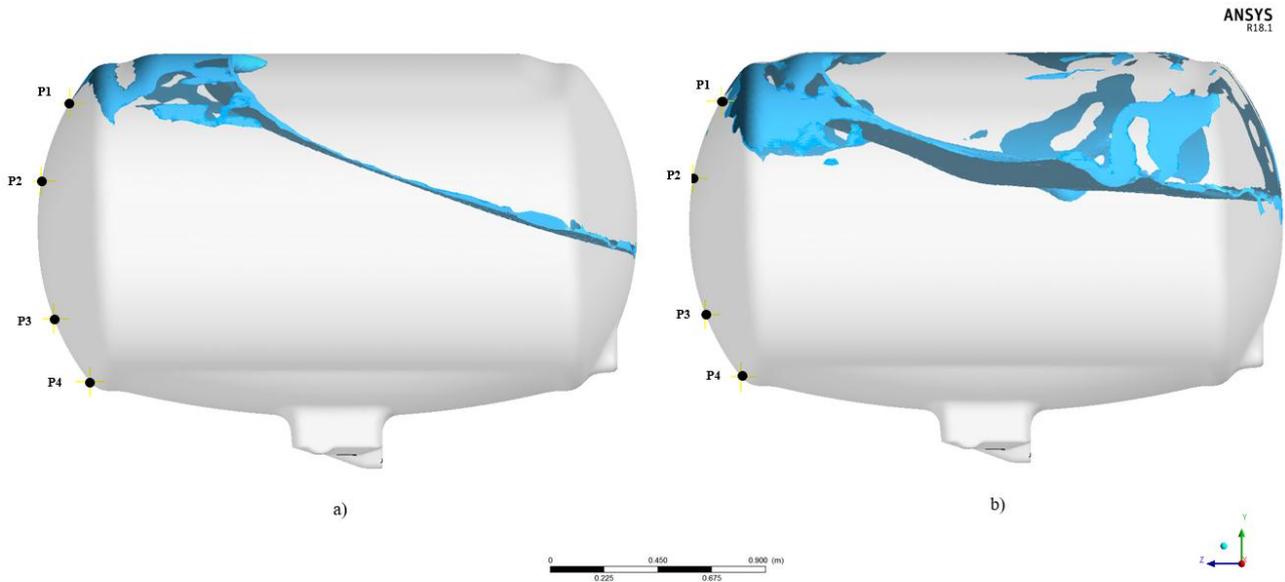


Figure 7. 2D visualization of sloshing in the sprayer tank: (a) time = 1.2s and (b) time = 3.1s.

Figure 8 show a pressure profile of the tank wall for level 75% of tank capacity in the same simulation time indicated in Fig. 7a.

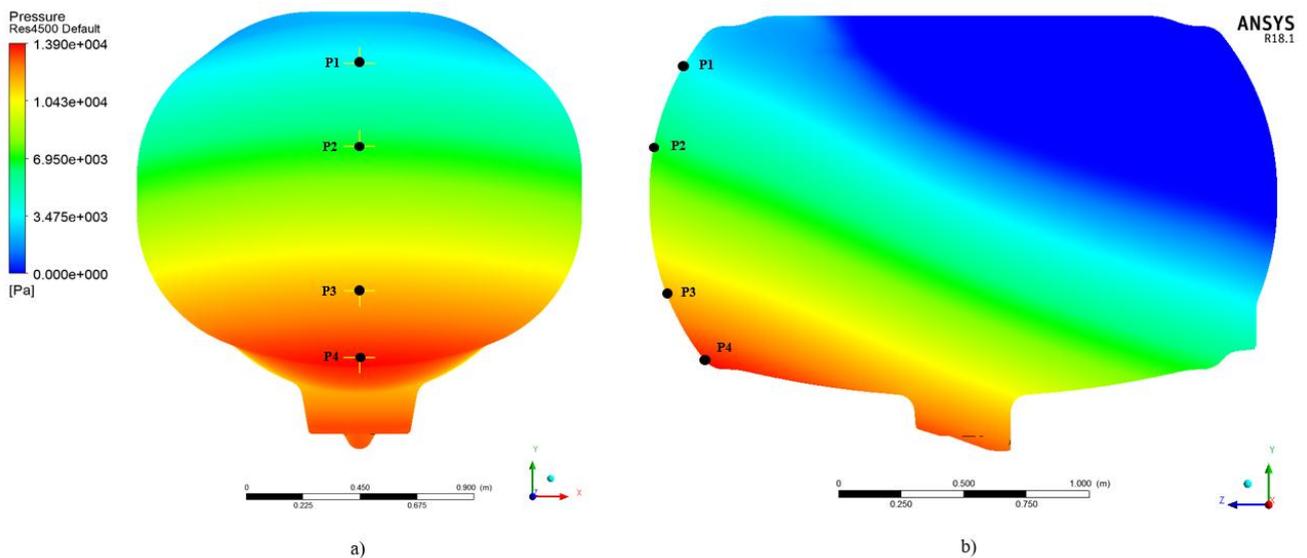


Figure 8. Pressure profile in the tank wall with simulation time = 1.2s: (a) front view and (b) side view.

Figure 9 show a pressure profile of the tank wall for level 75% of tank capacity in the same simulation time indicated in Fig. 7b.

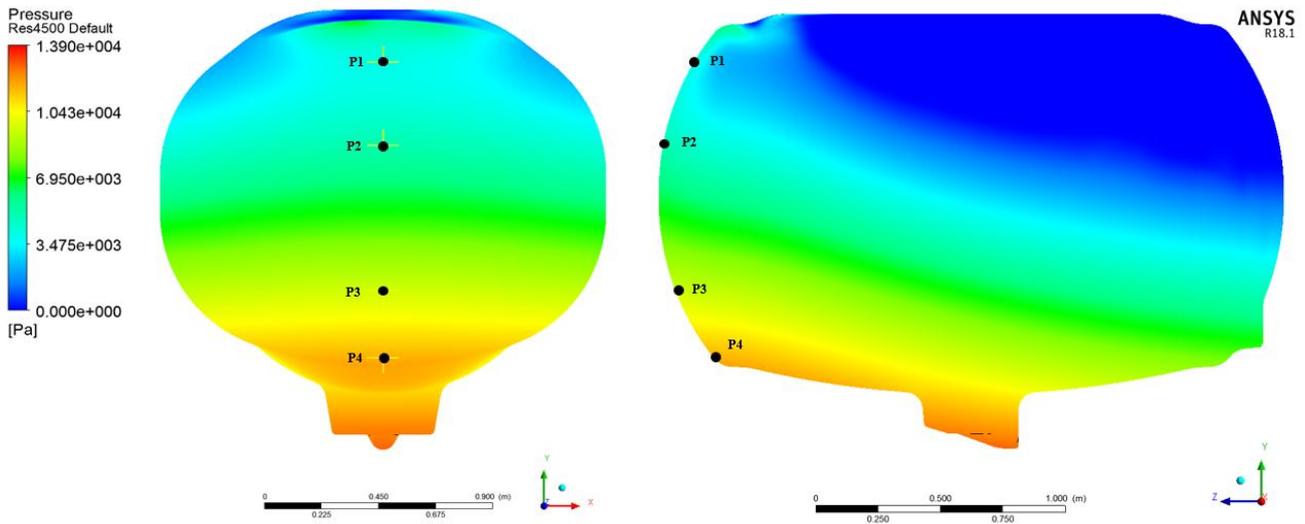


Figure 9. Pressure profile in the tank wall with simulation time = 3.1s: (a) front view and (b) side view.

Comparing the results showed in Fig. 8 and Fig. 9, it is possible to notice that the pressure capture points indicated in Fig. 2 are enough to represent the loads in the front tank wall. This is due to the design of tank and the one-dimensional deceleration (z-axis) applied in this simulation.

5. CONCLUSIONS

Numerical modeling proved useful in the design and analysis of this type of problem. It is possible to identify that both the pressure capture points in the submerged regions (mainly points P3 and P4), and in the regions where the sloshing wave reached the higher points, P1 and P2, the condition of higher tank filling volume (75%) was the one that generated the highest pressure values on the tank wall. Consequently, it represents a greater impact on the tank dimensioning. It can also be noticed that even with the minimum level of filling verified (25%), there was a recording of pressures on the wall at the point P1 and that with the level of 50%, the pressure values were closer to those of the level of 75% for all points, with higher oscillation frequency. The numerical approach allowed consider the non-linear effects of sloshing formation and being able to extract information about the stresses acting on the tank walls. This may lead to further studies to minimize these effects and reduce the costs of the tanks respecting the agitation inside them.

6. REFERENCES

- ANSYS. CFX18.1 – User guide, 2017.
- Fogal, M.L.F., Micheli, G.B., Padilha, A., Scalon, V.L., 2018. “Numerical and experimental analysis of radial fans applied in agricultural spreaders using cfd”. *Annals of ENCIT 2018*, Águas de Lindóia, SP.
- Godderidge, B., Turnock, S., Earl, C. “The effect of fluid compressibility on the simulation of sloshing impacts”. *Ocean Eng.* 36 (8) (2009) 578–587.
- Hasheminejad, M., Aghabeig, M., (2011). “Transient sloshing in half-full horizontal elliptical tanks under lateral excitation”. *Journal of Sound and Vibration.* 330:3507-3525.
- Micheli, G.B., Padilha, A., Scalon, V.L., 2015. “Análise numérico-experimental da agitação de calda em reservatórios de pulverizadores agrícolas”. *Journal of the Brazilian Association of Agricultural Engineering*, Vol. 35, n.6, pp. 1065–1078.
- Rhee, S.H. “Unstructured grid based Reynolds-averaged Navier-Stokes method for liquid tank sloshing”. *Journal Fluids Eng.* 127 (3) (2005) 572–582.
- Thiagarajan, K.P., Rakshit, D., Repalle, N. “The air-water sloshing problem: fundamental analysis and parametric studies on excitation and fill levels”, *Ocean Eng.* 38 (2) (2011) 498–508.
- Xue, M.-A., Zheng, J.H., Lin, P., Yuan, X., 2017. “Experimental study on vertical baffles of different configurations in suppressing sloshing pressure”. *China Ocean Eng.* 136, 178–189.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.