NONLINEAR WAVE SLOSHING IN WALLED CONTAINERS

Roger Matsumoto Moreira, rmoreira@id.uff.br
Vinícius Antônio Messias Martins, vinicius_martins@id.uff.br
Juan Andrés Santisteban Hidalgo, jandres@id.uff.br
School of Engineering / Fluminense Federal University
Rua Passos da Pátria, 156, bl.D, sl.563A, Niterói, R.J., Brazil. CEP: 24210-240.

Julio Tomás Aquije Chacaltana, julio.chacaltana@ufes.br
Danilo de Almeida Barbosa, danilofisico@yahoo.com.br
Environmental Engineering Department / Espírito Santo Federal University
Av. Fernando Ferrari, 514, Goiabeiras, Vitória, E.S., Brazil. CEP: 29075-910.

Michael Dickson Patterson, m.d.patterson@bristol.ac.uk
Faculty of Engineering / University of Bristol
University Walk, Bristol, BS81TR, U.K.

Abstract. *This paper aims to model numerically the generation and propagation of free surface waves in walled containers. Two different techniques are used: the finite volume method (FVM) with a homogeneous multiphase condition and the Smoothed-Particle Hydrodynamics (SPH). A good agreement is found between numerical and experimental results, which include comparisons of the free-surface evolution and the pressure distribution along the wall's container. Significant changes of the pressure profiles are found depending on the filling fluid used.*

Keywords*: Free-surface flow; wave impact; computational fluid dynamics.*

1. INTRODUCTION

When a partially filled reservoir is submitted to external exciting forces, the phenomenon known as wave sloshing may occur, causing large wave impacts at its walls. The loads produced by the wave motion have the potential to cause structural damage and/or affect the vessels stability. Sloshing waves are associated with a wide range of engineering problems, including the motion of liquid fuel in aircraft, the transportation of fluids in containers and the oil movement inside FPSO cargo tanks.

Wave sloshing is a classical free-surface flow problem. In the past, scientists such as Poisson, Rayleigh and Kirchhoff investigated this phenomenon. More recently, books have been published on this subject (Faltinsen and Timokha 2009, Ibrahim 2005). Recent references include Jiang *et al.* (2014), who analysed experimentally the hydrodynamics and structural characteristics of violent sloshing in elastic tanks; Moreira *et al.* (2012) and Chen *et al.* (2008), who assessed numerically the influence of the container's filling level, displacement and excitation frequency on the wave's dynamics; Godderidge *et al.* (2009), who studied resonant sloshing flows in a rectangular tank with a commercial Computational Fluid Dynamics (CFD) code; Nicolato and Moreira (2009), who implemented numerically geometry optimizations of the reservoir to reduce wave impact; Bredmose *et al.* (2003), who analysed experimentally and numerically the generation of nonlinear waves in a rectangular reservoir.

The Smoothed-Particle Hydrodynamics (SPH) was originally developed by Lucy (1977) and Gingold and Monaghan (1977) for modelling astrophysical phenomena. This Lagrangian particle method can easily discretize complex geometries and refine domains of interest. This method also captures the interfaces and their topological changes. This Lagrangian meshless method has been used to solve the fluid movement and the transport of physical properties from one region to another by following a finite number of fluid particles. Kim (2007) used SPH and finite-difference methods to simulate violent sloshing flows.

In this work, the Eulerian and Lagrangian reference frames are employed to study the wave motion inside a partially-filled reservoir that sways in an unsteady movement. The commercial CFD code ANSYS CFX and the SPH method are used to simulate experiment H10 proposed by Bredmose *et al.* (2003), with free-surface waves and pressure peak profiles being compared. Experiment H10 consists of a glass container with internal dimensions of 1480 x 400 x 750 mm^3 (length x width x height), which is displaced horizontally (*x*-axis) by a shaking table facility. Details of the imposed movement can be found in Fig. 1. The still-water depth (*h*) is set to 155 mm. For more experimental details, see Bredmose *et al.* (2003).
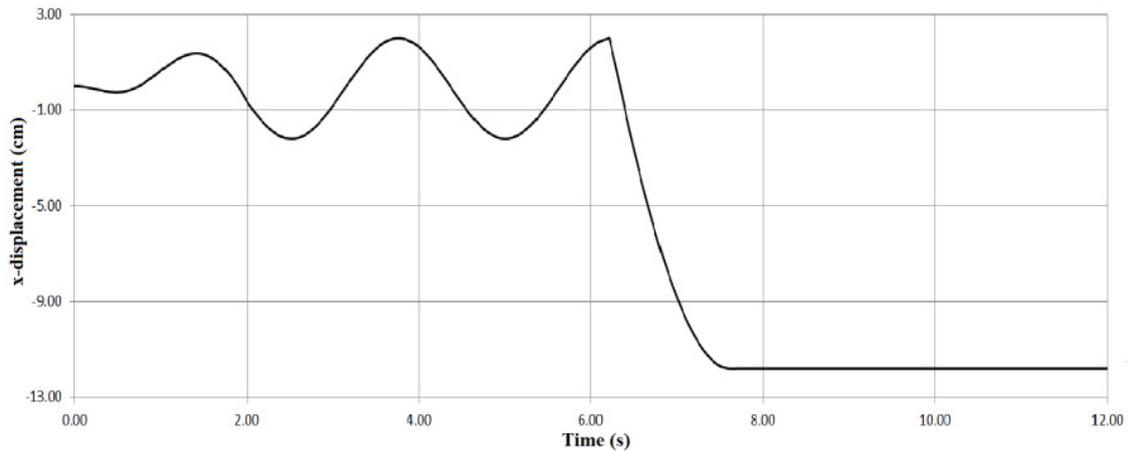
Figure 1. Horizontal displacement imposed to the tank in experiment H10 of Bredmose *et al.* (2003).

2. EULERIAN MODEL

Cartesian coordinates are defined with the $x-z$ plan at the bottom of the tank, such that the liquid occupies the region $y \leq h$ when at rest. The unsteady free-surface flow is modeled in three-dimensions with mass and momentum being conserved in the fluid domain. Continuity and Navier-Stokes equations are given by,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}) = 0, \tag{1}$$

$$\rho \frac{D\boldsymbol{u}}{Dt} = \nabla \cdot \boldsymbol{T} + \rho \boldsymbol{f}, \tag{2}$$

where ρ and $\boldsymbol{u} = (u,v,w)$ are the fluid's density and velocity; $\boldsymbol{f} = (0,-g,0)$ is the gravitational force; $\boldsymbol{T}$ is the stress tensor, which includes the effects of pressure, expansion and viscous forces. For a Newtonian fluid with viscosity μ and submitted to a dynamic pressure p, this tensor is written as,

$$\boldsymbol{T} = -p\delta_{ij} + \left(k - \frac{2}{3}\mu\right)\nabla \cdot \boldsymbol{u}\delta_{ij} + \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right). \tag{3}$$

A volume fraction ϕ is defined for the two-phase flow model,

$$\rho = \sum_{l=1}^{2} \phi_l \rho_l, \qquad \mu = \sum_{l=1}^{2} \phi_l \mu_l, \tag{4}$$

where $l=1,2$ represents liquid or gas. Turbulence is introduced via a k-ε model, where k and ε are respectively the turbulent kinetic energy and its dissipation.

Neumann and Dirichlet boundary conditions are imposed at the walls. The bottom and vertical walls of the container are considered rigid, impermeable and with a no-slip condition such that,

$$\boldsymbol{u}_{walls} = 0. \tag{5}$$

To complete our model, we assume that initially all the fluid domain is at rest when suddenly a harmonic horizontal displacement with a final "push" (see Fig. 1) is imposed to the reservoir, following experiment H10 of Bredmose *et al.* (2003). Two different liquids are used in the numerical simulations: firstly, water ($\rho = 997$ kg/m^3; $\mu = 8.9$ x 10^{-4} kg/ms), and then, glycerol ($\rho = 1,262$ kg/m^3; $\mu = 1.495$ kg/ms).

The boundary value problem is solved by the commercial CFD package ANSYS CFX release 11.0 (ANSYS 2006), which is based on a FVM (Versteeg and Malalasekera 1995, Maliska 2004). Figure 2 shows a cross-section of the container with the computational grid used, which comprises 19,800 hexahedral elements with 40,172 nodes. The mesh was refined near the free surface and walls in order to better predict the nonlinear profile of the fluid motion. A refined time grid is employed with time steps of 10ms for 12s of total time, with a Courant number smaller than 1. All the computations were carried out on a 64 bit, 2.40 GHz Intel Quad Core processor with 16 Gb of RAM.
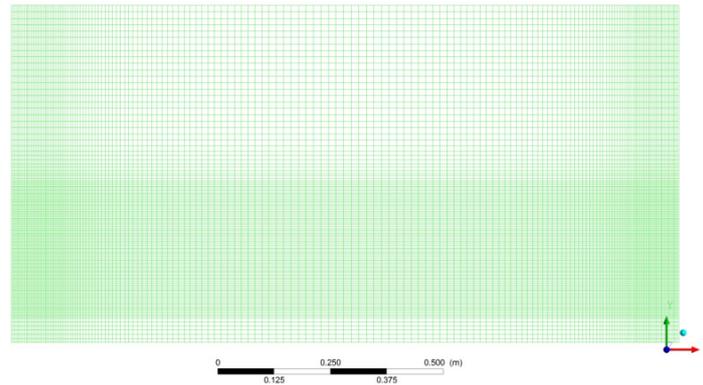
Figure 2. Computational grid of the Eulerian model.

3. LAGRANGIAN MODEL

To model the wave-sloshing phenomenon with the SPH method, the Navier-Stokes equations are written in a Lagrangian reference frame as,

$$\frac{d\rho}{dt} = -\rho \nabla . \vec{u} , \tag{6}$$

$$\frac{d\vec{u}}{dt} = \vec{g} - \frac{\nabla p}{\rho} + \nu \nabla^2 \vec{u} . \tag{7}$$

To take into account small variations of the specific mass with pressure changes of the liquid, the adiabatic Tait equation for the dynamic pressure is used (Batchelor, 2000),

$$p = \frac{\rho_0 c_0^2}{\gamma} \left(\left(\frac{\rho}{\rho_0} \right)^\gamma - 1 \right). \tag{8}$$

ρ_0 and c_0 are the density and the sound's velocity of the fluid at rest, whereas the pressure p_0 of the fluid at rest must be zero; $\gamma = 7$. For more details, see Fraga Filho & Chacaltana (2014).

The fluid domain is discretized by 150 x 15 marked points equally spaced along the *x-y* axis. To satisfy the boundary conditions, 1,207 virtual particles are distributed around the container's limits. The interaction between fluid and boundary particles is treated by a repulsive force like the Coulomb law. A constant time step of 0.05 ms is used in the simulations. The movement imposed to the container is shown in Fig. 1.

4. RESULTS

Figure 4 compares the free-surface results obtained from ANSYS CFX (left column) and SPH method (right column) with experiment H10 (center), extracted from Bredmose *et al.* (2003). Despite all the differences between the mathematical modelling (see sections 2 and 3), CFX and SPH numerical results reproduce quite well experiment H10. As the wave approaches the left wall of the container, SPH underpredicts the wave run-up compared to ANSYS CFX. The water film near the left wall falls due to gravity, which causes a significant depression of the free surface, which is captured by ANSYS CFX (see *t* = 8.04 s in Fig. 4).

Figure 5 shows the pressure history near the bed of the left wall obtained from ANSYS CFX for water and glycerol. Water results show a good agreement with Bredmose *et al.* (2003), with the "church roof" profile (double peak of pressure) being observed for 7 < *t* < 8 s and 9 < *t* < 10 s. For the same horizontal displacement and liquid-depth of the reservoir, no "church roof" profile is observed in the glycerol case, with a smoother pattern for the pressure history. For this substance, the maximum pressure on the wall is slightly higher due to its higher density. However viscous forces are greater compared to water, which attenuates the wave run-up on the wall.
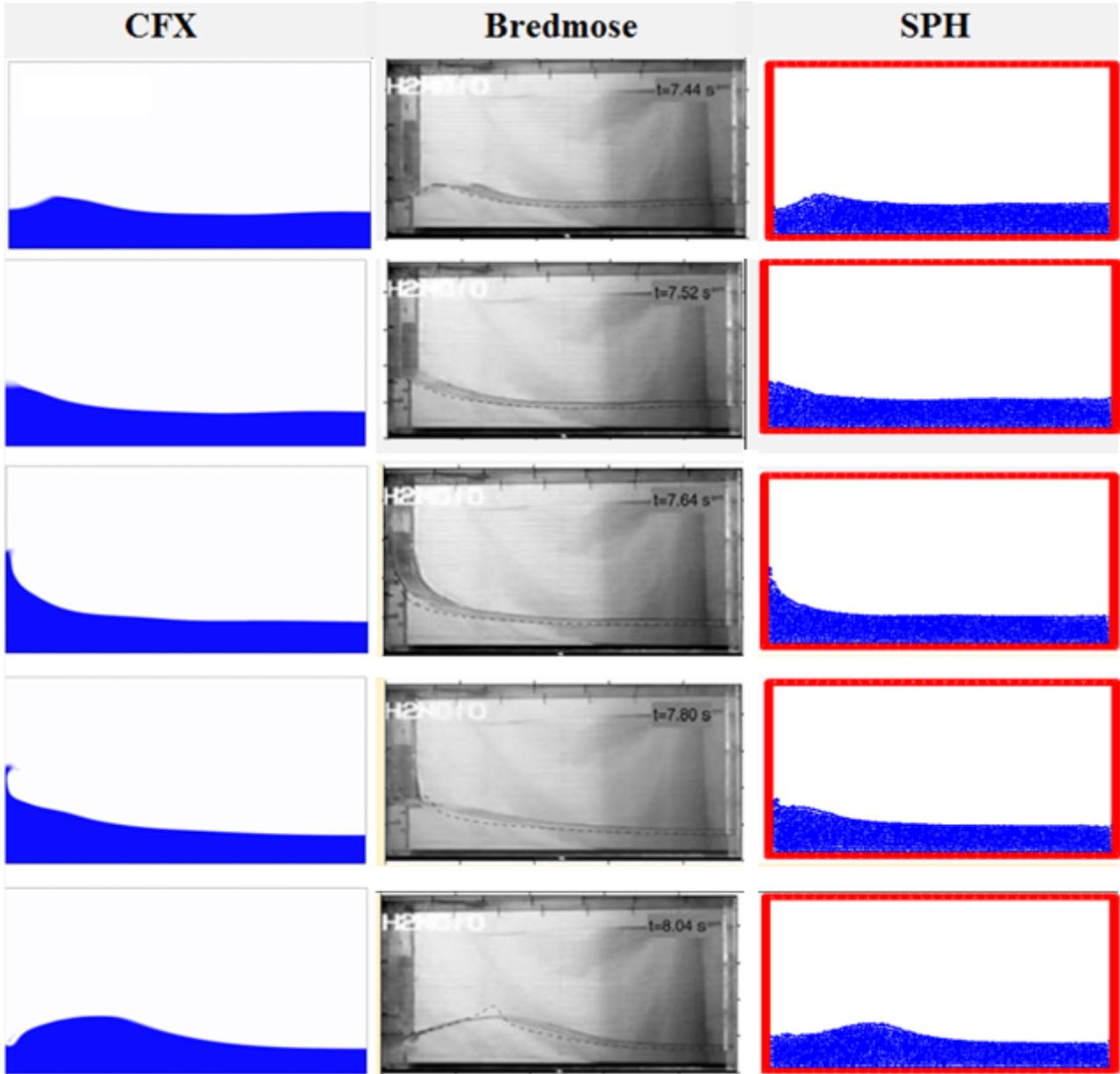
Figure 4. Free-surface snapshots from ANSYS CFX (left column), Bredmose's experiment H10 (center) and SPH method (right column).
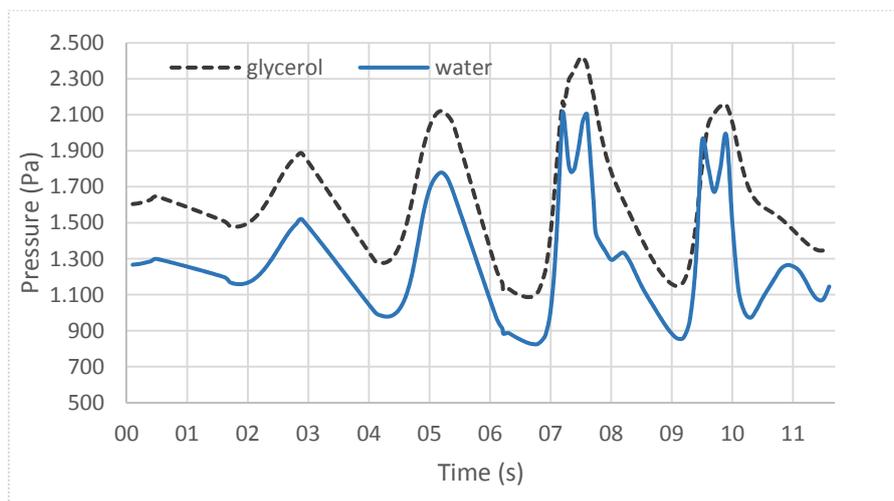

Figure 5. Pressure profiles at the left wall for water and glycerol.

5. CONCLUSIONS

Two different techniques (FVM and SPH) were implemented to model the generation and propagation of water waves in a walled container. Numerical results were compared with experiment H10 developed by Bredmose *et al.* (2003), with a good agreement found for the free-surface evolution and pressure history on the wall's container. During wave impact, CFX results are more consistent with the experiment performed whereas SPH underestimates wave run-up. Significant changes of the pressure profiles are found when water is substituted by glycerol. "Church roof" pressure profiles are no longer observed though higher wave impacts at the wall are still found in this case.

6. REFERENCES

ANSYS CFX 11.0, 2006. ANSYS CFX user's guide. ANSYS Inc.

Batchelor, G.K., 2000. An Introduction to Fluid Dynamics. Cambridge University Press.

Bredmose, H., Brocchini, M., Peregrine, D.H. and Thais, L., 2003. "Experimental investigation and numerical modelling of steep forced water waves". Journal of Fluid Mechanics, Vol. 490, pp. 217-249.

Chen, Y.G., Djidjeli, K. and Price, W.G., 2008. "Numerical simulation of liquid sloshing phenomena in partially filled containers". Computers & Fluids, Vol. 38, pp. 830-842.

Faltinsen, O.D. and Timokha, A.N., 2009. Sloshing. Cambridge University Press.

Fraga Filho, C.A.D. and Chacaltana, J.T.A., 2014. "Numerical study of heat diffusion employing the Lagrangian Smoothed Particle Hydrodynamics method: an analysis of the applicability of the Laplacian operator and the influence of the boundaries on the solution". International Review on Modelling and Simulations, Vol. 7, No. 6, pp. 994-1002.

Gingold, R.A. and Monaghan, J.J., 1977. "Smoothed Particle Hydrodynamics: Theory and Application to Non-spherical stars". Monthly Notices of the Royal Astronomical Society, Vol. 181, pp. 375-389.

Godderidge, B., Turnock, S., Tan, M. and Earls, C., 2009. "An investigation of multiphase CFD modelling of a lateral sloshing tank". Computers & Fluids, Vol. 38, pp. 183-193.

Ibrahim, R.A., 2005. Liquid Sloshing Dynamics: Theory and Applications. Cambridge University Press.

Jiang, M.R., Ren, B., Wang, G.Y. and Wang, Y.X., 2014. "Laboratory study on the hydrodynamic and structural characteristic of violent sloshing in elastic tanks". In Proceedings of the 7th International Conference on Thin-Walled Structures. Busan, Korea. Paper no. ICTWS2014-0801.

Kim, Y., 2007. "Experimental and numerical analyses of sloshing flows". Journal of Engineering Mathematics, Vol. 58, No. 1, pp. 191-210.

Lucy, L.B., 1977. "Numerical approach to testing the fission hypothesis". Astronomical Journal, Vol. 82, pp. 1013-1024.

Maliska, C.R., 2004. Transferência de calor e mecânica dos fluidos computacional. LTC.

Moreira, R.M., Thorley, R.F.D., Costa, D.G., Nicolato, P.C., Bacchi, R.D.A. and Chacaltana, J.T.A., 2012. "Modelling the dynamics of waves in reservoirs". In Proceedings of the 7th National Congress of Mechanical Engineering. São Luís, Brazil. Paper no. CONEM2012-0927.

Nicolato, P.C. and Moreira, R.M., 2009. "Numerical modelling of water wave impact on reservoirs". In Proceedings of the 20th International Congress of Mechanical Engineering. Gramado, Brazil. Paper no. COB09-0354.

Versteeg, H.K. and Malalasekera, W., 1995. An introduction to computational fluid dynamics: the finite volume method. Pearson Prentice-Hall.

7. RESPONSIBILITY NOTICE

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