Sph Modelling of Long-term Sway-Sloshing Motion in a Rectangular Tank

Murat Özbulut[1], Nima Tofighi[1], Mehmet Yıldız[1,], Ömer Gören[2]*
1) Faculty of Engineering and Natural Sciences, Sabanci University, Istanbul, Turkey
2) Faculty of Naval Architecture and Ocean Engineering, Istanbul Technical University, Istanbul, Turkey

ABSTRACT

This work aims to model long-term simulations of sway-sloshing motion in a partially filled rectangular tank with different water depths and enforced motion frequencies. The lateral motion frequency of the tank is chosen so as to coincide with the lowest theoretical natural frequency for the corresponding beam of the tank and initial depth of water reserve. A truly meshless method, Smoothed Particle Hydrodynamics (SPH) is used to discretize and solve the governing equations. It is shown that numerical results of the proposed SPH scheme are in good agreement with experimental and numerical findings of the literature.

KEY WORDS: SPH method; correction algorithms; Sway-Sloshing problem; Violent free-surface flows.

INTRODUCTION

The general stability and the motion characteristics of ships can be distorted significantly by the inner free-surface flows inside huge liquid cargo tanks. Ships like LNG, LPG and chemical tankers carry very large amounts of liquid products and are always subjected to the sloshing motion throughout their life cycle. The enforced motion frequency and fullness ratio of the tank determine the complexity of the fluid motion. Long-term simulations are required in critical motion frequencies and fullness ratios in order to understand the physical behaviors of the fluid motion clearly.

In the context of hydrodynamics, the enforced surge or pitch motion of the liquids in partially filled tanks has been attracting the attention of engineers and scientists hence leading to several experimental, theoretical and numerical studies in literature (Faltinsen and Timokha, 2001, Iglesias, et. al. 2011). As the free surface profiles change rapidly and the rate of its deformation is quite high during the evolution of the fluid flow, the non-linear effects in the mathematical modeling increases dramatically. Faltinsen and Timokha (2001) stated that the direct numerical methods like finite difference, finite volume and boundary elements, experience some deficiencies in volume and energy conservation as well as in the accurate description of fluid impact on the tank walls during the long term simulations. They modeled 2D surge and pitch excited sloshing problem by an adaptive multi-modal method which assumes that the flow is irrotational and there is no overturning waves. Although they obtained quite compatible results with experimental measurements, their model had a sub-limitation of water depth to tank beam ratio of 0.24. To further extend the scope of their theory, Faltinsen and Timokha (2002) presented an asymptotic approximation that can handle the sloshing phenomenon for the depth/beam ratios between 0.1 and 0.24. Besides theoretical methods, there are many attempts to model liquid sloshing by numerical methods like Marker and Cell (MAC), Volume of Fluid (VOF), Level Set (LS) and combined VOF-LS. The pros and cons of these methods are presented in the study of Chen et. al. (2009) briefly. They noted the shortcomings of these numerical techniques in capturing the topology of the free surface and the low numerical accuracy in acquiring impact pressures and forces. In addition to mesh-based numerical methods, there are also studies with mesh-free methods like the recent work of Khayyer and Gotoh (2013) which implements Moving Particle Semi-implicit (MPS) on the multi-phase modeling of sloshing flows considering the entrapped air.

After the extension of Smoothed Particle Hydrodynamics (SPH) method into the incompressible fluid flow problems by Monaghan (1992), the research studies on the modeling of violent free surface flows by using SPH increased enormously because of the Lagrangian nature of the method which captures the free surface intrinsically. It has been applied to various types of violent free surface hydrodynamics problems like dam-break (Marrone, et.al. 2011), waves on beaches (Monaghan and Kos, 1999), ship bow waves produced by certain ship hulls (Marrone et.al. 2012) and water entry of a free-falling object (Shao, 2009) problems. Sloshing problem includes highly non-linear free surface deformations, instantaneous pressure loads on walls and variable velocity fields in problem domains, which requires the implementation of coupled numerical schemes enabling accurate and precise modeling of the physical phenomenon. Hence, one may find several other studies on modeling sloshing problem which used different SPH numerical schemes (Gotoh, et.al. 2014, Delorme, et.al. 2009, Chowdhury and Sannasiraj, 2014).

This work aims to model lateral motion of a partially filled rectangular tank by Weakly Compressible SPH approach together with well-known artificial viscosity term (Monaghan, 1999). Additional numerical correction algorithms namely, density correction and hybrid Velocity-updated XSPH (VXSPH) and Artificial Particle Displacement (APD) (Ozbulut et. al. 2014) are also added into the numerical scheme which are required for the high accuracy and stability of the solution. One of the objectives of this study is to draw the framework of the utilization of APD algorithm for the free surface problems. In long-term simulations, slight errors may accumulate during the solution procedure and lead to a completely different behavior at later stages of the fluid motion. Conservation of volume is of utmost importance in long-term simulations because the total volume of the fluid inside the tank is directly related with the natural frequency of the tank which affects the dynamics of the flow. The common practice of utilizing the maximum velocity inside computational domain as the APD velocity coefficient produces significant increase in fluid volume. This was not encountered in short-term simulations of the free-surface flows (Ozbulut et. al. 2014) or flow problems having bounded domains (Shadloo et.al. 2011b). During the utilization of global velocity coefficient, the particles which hit to the side walls have much higher velocities and the APD velocity coefficient is chosen as this high velocity for the whole domain at that instant. As a result of this choice, the particles away from the side walls gain larger APD values and gradually start to move towards to free surface which leads to an increase in the total volume of the fluid domain. In order to avoid the artificial increase in volume, APD velocity coefficient is computed through velocity variance for each particle, referred to as local velocity coefficient. The total volumes of both global and local velocity coefficients are compared for long term simulation of dam-break problem, where significant improvement in volume conservation is observed for APD algorithm with local velocity coefficient.

In the following section, the governing equations of the fluid motion, their discretization by SPH method and the numerical treatment algorithms used during simulations will be presented. The comparative results of the total volume by utilizing local and global APD term will be shown in the same section. After introducing the entire numerical scheme, the simulation results of the free-surface elevations obtained at the left wall of the tank will be compared with the experimental and numerical findings of the literature in both time and frequency domains in the third section. The final comments and concluding remarks will be drawn in the last section.

MATHEMATICAL FORMULATION

Governing Equations

The sway-sloshing problem is solved using Euler's equation of motion and continuity coupled with Lagrangian particle advection. Sway-sloshing problem is characterized as a violent free surface problem wherein in general, viscous effects are deemed to be negligible. Allowing for fluid element rotation, the governing equations may be specified as

$$\rho D\vec{\mathbf{u}} / Dt = -\nabla p + \rho \vec{\mathbf{g}} \tag{1}$$

$$D\rho / Dt = -\rho \nabla \cdot \vec{\mathbf{u}} \tag{2}$$

$$\vec{\mathbf{u}} = D\vec{\mathbf{r}} / Dt \tag{3}$$

where *D/Dt* is the material time derivative, ∇ is the Nabla operator, $\vec{\mathbf{u}}$, $\vec{\mathbf{r}}$ and $\vec{\mathbf{g}}$ are position, velocity and gravitational acceleration vectors, p and ρ denote pressure and density.

WCSPH approach is utilized for the discretization of the governing equations. It uses an artificial equation of state which couples density and pressure variations through a coefficient which is most commonly known as speed of sound. The state equation enables one to calculate the pressure for computing pressure gradient in the Euler's equation of motion. There are numerous forms of state equation within the scope of WCSPH while the one proposed by Monaghan & Kos (1999) is used during the simulations of this study:

$$p = \frac{\rho_o c_o^2}{\gamma}\left[\left(\rho / \rho_o\right)^{\gamma} - 1\right] \tag{4}$$

In Eq. 4, c_o is the reference speed of sound, γ is a coefficient and is taken to be equal to 7 while ρ_o is the reference density of water which is equal to 1000 (kg/m^3). To keep the density variations below %1 with respect to the reference density, the reference speed of sound parameter should be at least 10 times higher than the expected maximum velocity in the fluid domain (Monaghan, 1992) which leads c_0 taken as 40 (m/s) in the simulations of this study.

SPH Discretization of the Governing Equations

The SPH method interpolates any arbitrary continuous function, $A\left(\vec{\mathbf{r}}_{\mathbf{i}}\right)$ or concisely denoted as $A_{\mathbf{i}}$ in the following manner:

$$A_{\mathbf{i}} \cong \left\langle A\left(\vec{\mathbf{r}}_{\mathbf{i}}\right)\right\rangle \equiv \int_{\Omega} A\left(\vec{\mathbf{r}}_{\mathbf{j}}\right) W(r_{\mathbf{ij}}, h) d^3\vec{\mathbf{r}}_{\mathbf{j}} \tag{5}$$

Here, h is the smoothing length which determines the compact support area of each particle in the domain and $r_{\mathbf{ij}}$ represents the magnitude of the distance vector given as $\vec{\mathbf{r}}_{\mathbf{ij}} = \vec{\mathbf{r}}_{\mathbf{i}} - \vec{\mathbf{r}}_{\mathbf{j}}$ for a particle of interest and its neighboring particles, denoted by using boldface subscripts **i** and **j**, respectively while $\vec{\mathbf{r}}_{\mathbf{i}}$ and $\vec{\mathbf{r}}_{\mathbf{j}}$ are the position vectors for the particles. $W(r_{\mathbf{ij}}, h)$ is the kernel function which weighs the contribution of each neighboring particles according to their distance with respect to the particle of interest. A piecewise quintic kernel function is employed in the simulations of this study because of its accuracy and stability characteristics, (Shadloo et. al. 2011).

By using SPH approach given in Eq. 5, Euler's equation of motion and the mass conservation may be discretized by the SPH method hence leading to the following relations:

$$D\vec{\mathbf{u}}_{\mathbf{i}} / \mathrm{Dt} = -\sum_{\mathbf{j}=1}^{N}\left(p_{\mathbf{i}} / \rho_{\mathbf{i}}^2 + p_{\mathbf{j}} / \rho_{\mathbf{j}}^2 + \Pi_{\mathbf{ij}}\right)\nabla_{\mathbf{i}} W_{\mathbf{ij}} \tag{6}$$

$$D\rho_{\mathbf{i}} / Dt = \rho_{\mathbf{i}} \sum_{\mathbf{j}=1}^{N}\left(m_{\mathbf{j}} / \rho_{\mathbf{j}}\right)\left(\vec{\mathbf{u}}_{\mathbf{i}} - \vec{\mathbf{u}}_{\mathbf{j}}\right) \cdot \nabla_{\mathbf{i}} W_{\mathbf{ij}} \tag{7}$$

Here, the mass is denoted by $m_{\mathbf{j}}$, $\nabla_{\mathbf{i}}$ is the gradient operator, N is the total neighbor particle number of the particle of interest and $\Pi_{\mathbf{ij}}$ is the artificial viscosity term which is calculated as:

$$\Pi_{\mathbf{ij}} = \begin{cases} -\alpha\mu_{\mathbf{ij}} \dfrac{c_{\mathbf{i}} + c_{\mathbf{j}}}{\rho_{\mathbf{i}} + \rho_{\mathbf{j}}}, & \vec{\mathbf{u}}_{\mathbf{ij}} \cdot \vec{\mathbf{r}}_{\mathbf{ij}} < 0 \\ 0, & \vec{\mathbf{u}}_{\mathbf{ij}} \cdot \vec{\mathbf{r}}_{\mathbf{ij}} > 0 \end{cases} \tag{8}$$

where,

$$\mu_{\mathbf{ij}} = h \frac{\left(\vec{\mathbf{u}}_{\mathbf{i}} - \vec{\mathbf{u}}_{\mathbf{j}}\right) \cdot \left(\vec{\mathbf{r}}_{\mathbf{i}} - \vec{\mathbf{r}}_{\mathbf{j}}\right)}{\left\|\vec{\mathbf{r}}_{\mathbf{i}} - \vec{\mathbf{r}}_{\mathbf{j}}\right\|^2 + \theta h^2} \tag{9}$$

The purpose of adding the artificial viscosity term to the linear momentum balance is to advance the numerical stability of the system because of using finite number of particles to represent the continuum domain. It gives a small amount of diffusion to the fluid as the flow evolves where it will has the form of Navier-Stokes viscosity as the particle number goes to infinity (Monaghan & Kos, 1999). The level of such damping terms should be optimized to minimize its impact on the solution while preserving the stabilizing effect.

Here, θ is a constant (taken as 0.05 in this work) which is added to the denominator of the Eq. 9 to prevent any singularity and α parameter is the coefficient which determines the amount of the artificial diffusion in the numerical solution. The value of α parameter is taken as 0.06 in this work which provides satisfactory results in the simulations of the present work. In Eq. 8, the local speed of sound values of each particle is calculated by $c_{\mathbf{i}} = c_0 \left(\rho_{\mathbf{i}} / \rho_o\right)^{(\gamma-1)/2}$.

The time integration procedure utilizes a predictor-corrector scheme. Free-slip boundary condition is applied for all of the bounded walls through ghost particle technique. Dynamic free surface condition is satisfied through assigning zero pressure to all free surface particle as or, equivalently, setting the densities of the free surface particles to reference density. Further details of numerical scheme and boundary condition implementations may be found in our previous study (Ozbulut et. al. 2014).

Corrective Numerical Treatments and APD Algorithm with Local Velocity Variance

SPH method has many advantages on the modeling of violent free surface hydrodynamics problems; however it is also known that it produces highly oscillatory pressure fields due to calculating pressures by using density variations (Molteni & Colagrossi, 2009, Antuono et al. 2010). In order to limit these pressure field fluctuations, a density smoothing/correction and hybrid VXSPH+APD algorithms are included into the numerical scheme. Density smoothing helps to eliminate the noise in pressure. On the other hand, APD term provides a homogeneous particle distribution as the flow evolves and hinders particle clustering which is of detrimental effect on the accuracy of pressure field. Shadloo et.al. (2011) added a small artificial displacement term to the position vectors of each particle in flow simulations with bounded domains since this term encourages uniform particle distribution in the flow field by breaking string like particle arrangement. There are also other approaches for the regularization/ homogenization of particle distribution (Xu et.al. 2009, Lind et.al. 2012). In our previous work (Ozbulut et. al.2014), hybrid VXSPH+APD treatment was applied to the free surface flow problems where the fluid domain was divided into two regions, namely, fully populated and free surface fluid regions. VXSPH algorithm was applied to a very thin layer which acts like a surface tension force thus keeping the particles together on the free surface. The readers are referred to our recent study for the implementation of the VXSPH treatment in free surface flows (Ozbulut et. al.2014).

For the fully populated regions of the fluid flow, APD term was employed and it provided uniformity in the distribution of the particles thus improving the robustness and the accuracy of the numerical scheme. The APD term is defined as follows:

$$\delta\vec{\mathbf{r}}_{\mathbf{i}} = \beta \sum_{\mathbf{j}=1}^{N} \frac{\vec{\mathbf{r}}_{\mathbf{ij}}}{r_{\mathbf{ij}}^3} r_{\mathbf{i},o}^2 u_{\max} \Delta t \tag{10}$$

where, β is a problem dependent constant which is generally taken between the range of 0.01-0.05. $r_{\mathrm{i,o}} = \sum_{\mathbf{j}} r_{\mathbf{ij}} / N$ is the average particle distance for a given particle $\mathbf{i}$ and $u_{\max}$ is magnitude of the maximum velocity in the whole problem domain. The APD algorithm based on the magnitude of the maximum particle velocity in the whole problem domain is hereafter referred to as *global maximum velocity APD.*

During the course of long-term simulations conducted in this study, significant increase in volume has been observed. However, such an increase was not noticed in short term simulations of violent free-surface flows conducted in our previous studies (Ozbulut et. al. 2014). The increase in the total volume arises from the $u_{\max}$ term in the APD algorithm which was originally taken as the maximum of all fluid particle velocities within the computational domain. The fact that short-term simulations have been successful with global maximum velocity coefficient shows that the errors due to such treatment are not large, although when accumulated in time may have adverse effects on the simulation results. In violent free-surface flows such as dam-break and sloshing problems, the variation in the velocity field of the flow is high, especially upon and after the impact of the water reserve on the tank walls. Thus, applying the same global $u_{\max}$ value to all particles generates large APD values, hence pushing the particles towards the free-surface and resulting in a gradual increase in total fluid volume. To ensure the total volume conservation of the fluid domain, instead of using global maximum velocity term, an averaged local velocity variance is employed in the APD algorithm and computed for all fluid particles except the free surface as follows

$$\delta\vec{\mathbf{u}}_{\mathbf{i}} = \frac{\sum_{\mathbf{j}=1}^{N} (\vec{\mathbf{u}}_{\mathbf{i}} - \vec{\mathbf{u}}_{\mathbf{j}})\, \mathrm{W}_{\mathbf{ij}}}{\sum_{\mathbf{j}=1}^{N} W_{\mathbf{ij}}}, \qquad u_c = \left|\delta\vec{\mathbf{u}}_{\mathbf{i}}\right| \tag{11}$$

In order to differentiate the local maximum velocity from the global maximum velocity in Eq.(10), u_{max} term is denoted as u_c in Eq.(11). Another advantage of using local averaged velocity variance APD (which will be referred as *Local Velocity APD* from now on) is the elimination of constant β parameter in Eq. 10.

To show the effect of global and local velocity term in the APD algorithm during the simulations of violent free surface flow problem, dam-break problem is simulated until the water reserve calms down. The simulation conditions and the parameters are taken the same as the one presented in the study of Ozbulut et. al. (2014). The straight black line shows the water level where free-surface profile should reside as t→∞ to conserve the initial volume. As observed from the Fig. 1, at t = 25(s), global velocity coefficient leads significant increase in volume which has been effectively avoided by the use of local velocity coefficient.

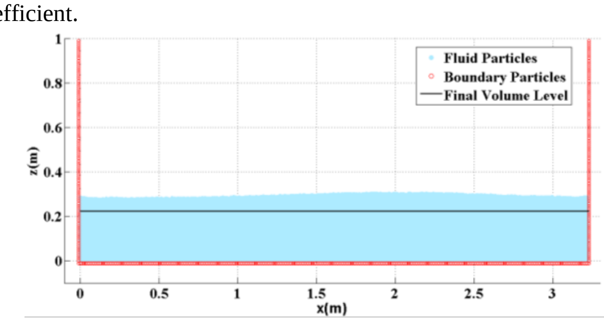


Figure 1: (a) Final fluid volume of the dam-break modeled using the APD algorithm with the global velocity coefficient at *t* = 25 (s).

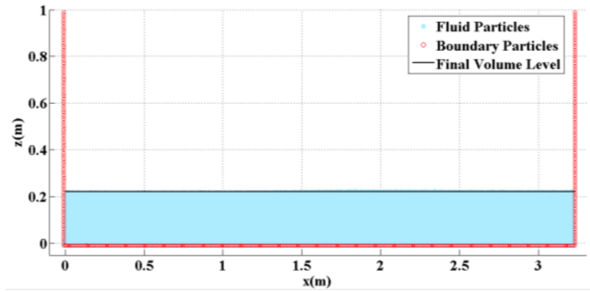


Figure 1: (b) Final fluid volume of the dam-break modeled employing the APD algorithm with local velocity coefficient at t = 25 (s).

NUMERICAL RESULTS

This work focuses on the simulation of long-term sway sloshing problem for two different fullness ratio of the container which has a rectangular cross section. The harmonic sway motion of the tank is induced by a sinusoidal function such that $x(t) = A \sin(\omega t)$ where x is the horizontal position of the tank, A is the amplitude and ω is the angular frequency of the motion. The representative geometry of the problems is given in Fig. 2 and dimensions, amplitudes and angular frequencies of the test cases are tabulated in Tab. 1.

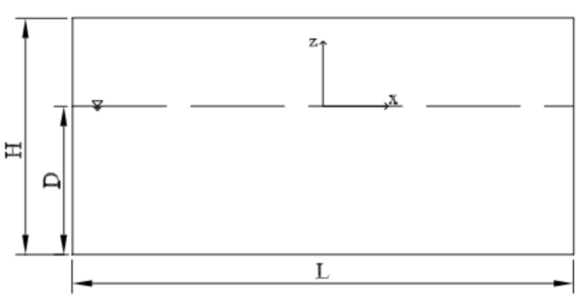


Figure 2: The general geometry of the sway-sloshing simulations.

The test cases scrutinize the kinematic characteristics of the fluid motion where the free surface deformations are compared with the numerical and experimental findings of the work of Pakozdi (2008). The angular frequency of the enforced harmonic sway motion is chosen close to the first theoretical natural frequency of the sloshing motion (ω 1 in Tab. 1). The theoretical n^{th} natural frequency at a certain water depth (D) and tank width (L) is given as (Chen et.al. 2009, Pakozdi, 2008):

$$\omega_n = \sqrt{\frac{ng\pi}{L}\tanh\left(\frac{n\pi}{L}D\right)} \tag{12}$$

Table 1: Dimension, amplitude and frequency values in two cases.

	H (m)	D (m)	L (m)	A (m)	ω (rad/s)	ω/ω_1
Case 1	1.05	0.30	1.73	0.08	3.488	1.173
Case 2	1.05	0.50	1.73	0.08	3.696	1.032

During the simulations, density and gravitational acceleration are taken as 1000 (kg/m^3) and 9.81 (m/s^2). The equidistant initial particle spacing in both x and z directions is utilized, namely, $\Delta x = \Delta z$ =0.01 (m) and the time step value is Δt = 0.00015 (s). The smoothing length parameter, h, is taken as $h=1.33\Delta x$. The particles are in rest and have only hydrostatic pressure at t = 0 (s). The numerical data collection/writing frequency is 11.11 (Hz).

The free surface at the probe location (i.e., x = -0.815 (m)) is determined by averaging the elevation of the free surface fluid particles between x = -0.805 (m) and x = -0.825 (m), which corresponds to the physical region occupied by the wave probe used in the reference experimental study.

The examination of the free surface profile variations during the horizontal motion of a partially filled container gives a clear foresight to understand the physics behind the chaotic motion of fluid in the tank. In this work, the full-time histories and the corresponding frequency domain analysis of the free surface elevations are scrutinized. As the enforced tank motion has a constant frequency, the water reserve in the tank is expected to respond with the similar frequency in free surface elevations at the side walls of the tank. The obtained simulation data of the present work is compared with the results of the numerical and experimental findings of Pakozdi (2008) where the numerical solution of the reference work employs also WCSPH approach with periodical density re-initialization to increase the accuracy of the pressure field, particle refinement in the corners of the tank, XSPH correction and Runge-Kutta time integration scheme for the time marching of the flow.

The time histories of the fluid motion and the frequency domain analyses are given in Fig. 3 and Fig. 4, respectively. The periods and the amplitudes of the free surface elevation in time histories graphics are in a very good agreement with the experimental measurements and numerical findings of the reference work except for a very few instances. This compatibility can also be seen from the frequency domain analysis where the first three modes of the frequencies are matching with a good accuracy. On the other hand, one can notice that there are some discrepancies at the maximum points of the time series and frequency domain plots. These mismatches arise from the different approximations of each study during the determination of the free-surface region when the water reserve reaches to the maximum level at the wall. The discrepancies may be attributed to inherent differences in the determination of free-surface location (on vertical walls via particle positions) compared to the data acquired by the probe used in experiments. Additionally, the water splashes after it impact on side walls, and this brings another difficulty on detecting the free surface level at that time. In the simulations of this study, the average neighboring particle number is around 40 in each time-step and the particles less than 25 neighbor particles are considered as a free surface particle.

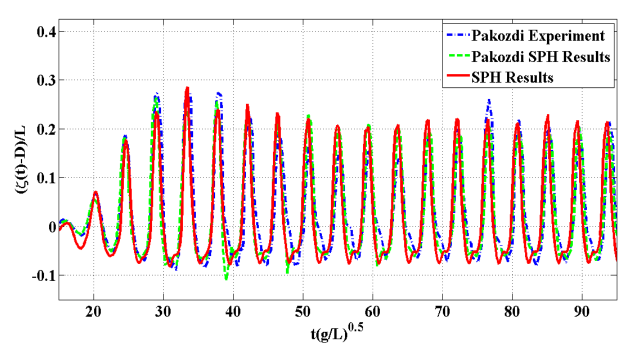


Figure 3: (a) Comparison of the time histories of the free-surface elevations at x = −0.815 (m) for Case 1.

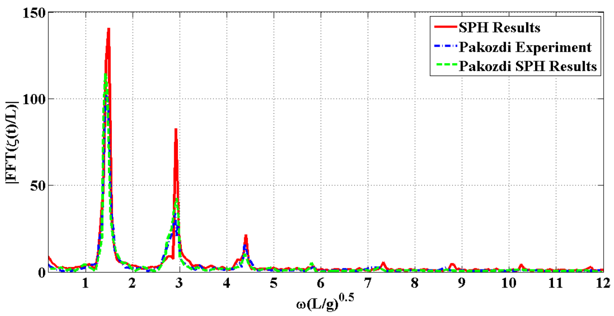


Figure 3 (b): Comparison of the frequency domain analysis of the free-surface elevations at x = −0.815 (m) for Case 1.

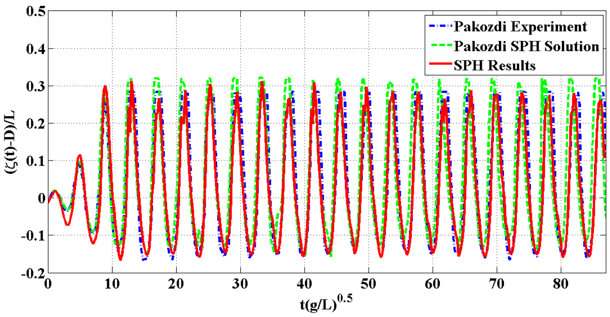


Figure 4: (a) Comparison of the time histories of the free-surface elevations at x = −0.815 (m) for Case 2.

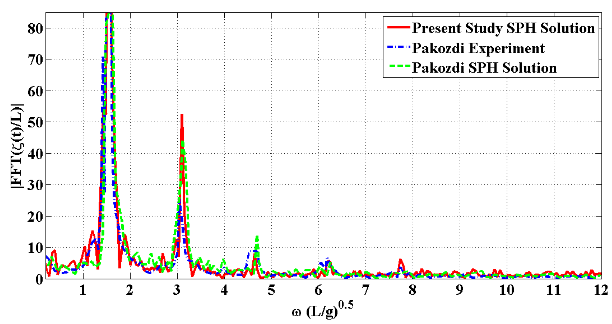


Figure 4: (b) Comparison of the frequency domain analysis of the free-surface elevations at x = −0.815 (m) for Case 2.

An illustrative figure which shows the free surface and the fully populated fluid domains therefore represents the effective regions for VXSPH and APD algorithms is given in Figure 5. As can be seen from this figure, the free surface region where the VXSPH algorithm is defined as a very thin layer and does not alter the physics of the flow.

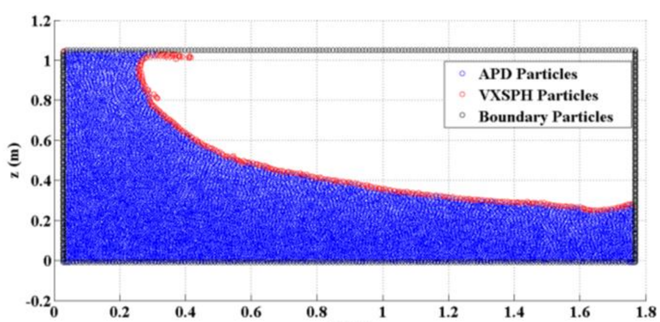
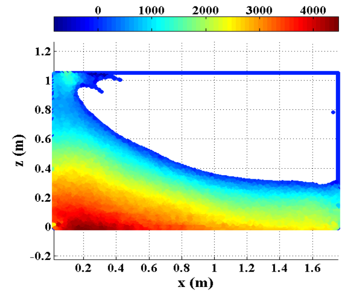
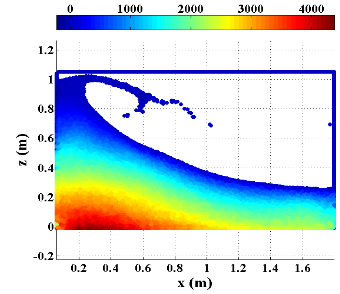


Figure 5: The sketch of free surface (red) and fully populated (blue) regions during the evolution of sway-sloshing motion.

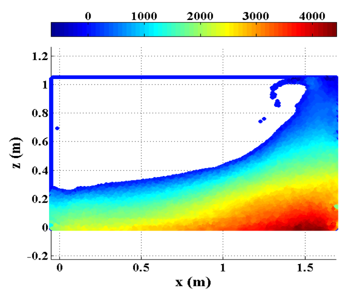
In Figure 6 are shown pressure fields, free-surface profiles and particle distribution at four different instants of test case 1. As seen from these snapshots, the computed pressure fields during the sloshing motion are non-oscillatory and smooth due to the homogeneous particle distribution and precise free-surface profiles.



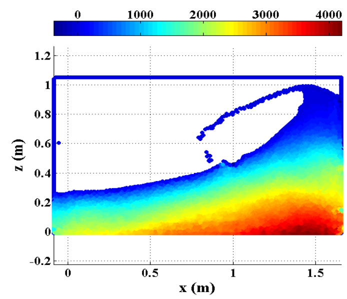
(a) Snapshot at t=30.015(s)



(b) Snapshot at t=30.375 (s)



(c) Snapshot at t=31.455(s)



(d) Snapshot at t=31.725 (s)

Figure 6: Pressure fields, particle distribution and free-surface profiles at four different instants.

CONCLUDING REMARKS

Two-dimensional long-term sway-sloshing motion of a partially filled rectangular tank is investigated in this work. The simulation of fluid motion is carried on until the free surface elevations on the side walls reach a harmonic steady state. Euler's equation of motion and continuity equation are discretized utilizing WCSPH approach and integrated in time via a predictor-corrector scheme. As the conventional SPH method requires some numerical treatments in order to have accurate and robust numerical solutions, density correction and hybrid VXSPH+APD algorithms are incorporated into the conventional SPH numerical scheme.

The hybrid VXSPH+APD treatment was first employed in our previous work (Ozbulut et.al 2014) where the APD algorithm was applied to the fully populated fluid domain and VXSPH was utilized for the free surface region. It is observed that in the long-term modeling of violent free surface flows where the velocity field changes rapidly in whole fluid domain, the APD algorithm with global maximum velocity term causes a significant volume increase as the flow evolves in time. Using the same global maximum velocity coefficient for all fluid particles causes fluid particles to move upward which leads to an increase in the total volume of the fluid gradually. To avoid this problem, local velocity coefficient is proposed and it is shown that the total volume of the fluid domain is exactly conserved in the long-term simulation of dam-break problem.

After showing the effectiveness of the proposed local velocity coefficient in APD algorithm, the kinematic characteristics of the sway-sloshing problem is scrutinized quantitatively. A rectangular tank with two different fullness ratios is enforced to move laterally by a motion frequency which is close to the first theoretical natural frequency in corresponding water depth. The free-surface elevations on the left wall of the tank are compared with experimental and numerical solutions available in literature, showing good agreement in both time and frequency domains. Considering the results obtained for the test cases, it is possible to infer that the proposed WCSPH scheme and related corrective numerical treatments are able to capture the long-term evolution of sway-sloshing motion.

Recall that the current study is intentionally limited to the investigation of the kinematical characteristics of the sway-sloshing problem to reveal the effectiveness of the proposed scheme whereby we have considered only a single enforced motion frequency, which is set close to the first theoretical natural frequency.
In future study, a frequency domain analysis will be performed to construct the wave response curves of the problem, which will enable us to further test the capabilities of proposed numerical scheme and also study the complete physics of the problem. Additionally, the dynamic characteristics of the problem will be compared with available literature data referring to pressure loads on the walls.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge financial support provided by the Scientific and Technological Research Council of Turkey (TUBITAK) for project number 112M721.

REFERENCES

Antuono, M, Colagrossi, A, Marrone, S, Molteni, D (2010). “Free-surface flows solved by means of SPH schemes with numerical diffusive terms”, *Computer Physics Communications*, Vol. 181, 532-549.

Chen, YG, Djidjeli, K, and Price, DJ (2009). “Numerical simulation of liquid sloshing phenomena in partially filled containers”, *Computers & Fluids,* Vol 38, pp 830-842.

Chowdhury, SD, Sannasiraj, SA, (2014), “Numerical simulation of 2D sloshing waves using SPH with diffusive terms”, *Applied Ocean Research,* Vol. 47, pp 219-240.

Delorme, L, Colagrossi, A, Souto-Iglesias, A, Zamora-Rodriguez, R, Botia-Vera,E, (2009). “A set of canonical problems in sloshing, Part I: Pressure field in forced roll—comparison between experimental results and SPH”, *Ocean Engineering,* Vol. 36, pp 168-178.

Faltinsen, OM, and Timokha, AN, (2001) “An adaptive multimodal approach to nonlinear sloshing in a rectangular tank”, *Journal of Fluid Mechanics*, Vol 432, pp 167-200.

Faltinsen, OM, and Timokha, AN, (2002). “Asymptotic modal approximation of nonlinear resonant sloshing in a rectangular tank with small fluid depth”, *Journal of Fluid Mechanics*, Vol 470, pp 167-200.

Gotoh, H, Khayyer, A, Ikari, H, Arikawa, T, Schimosako, K, (2014). “On enhancement of Incompressible SPH method for simulation of violent sloshing flows”, *Applied Ocean Research*, Vol. 46, pp 104-115.

Iglesias, AS, Botia-Vera, E, Marin, A, and Perez-Arribas, F (2011). “A set of canonical problems in sloshing. Part 0: Experimental setup and data processing”, *Ocean Engineering,* Vol 38, pp 1823-1830.

Khayyer, A, Gotoh, H (2013). “Enhancement of performance and stability of MPS mesh-free particle method for multiphase flows characterized by high density ratios”, *Journal of Computational Physics”*, Vol 242, pp 211-233.

Lind, SJ. Xu, R. Stansby, PK. Rogers, BD. (2012). “Incompressible smoothed particle hydrodynamics for free-surface flows: A generalised diffusion-based algorithm for stability and validations for impulsive flows and propagating waves”, *Journal of Computational Physics,* Vol. 231, pp. 1499-1523.

Marrone, S, Antuono, M, Collagrossi, A, Collichio, G, Le Touze, D. Graziani, D, (2011). “δ-SPH model for simulating violent impact flows”, *Computational Methods in Applied Mechanical Engineering,* Vol 200, pp 1526-1542.

Marrone, S, Bouscasse, B, Collagrossi, A, Antuono, (2012). “Study of ship wave breaking patterns using 3D parallel SPH simulations”, *Computers & Fluids*, Vol 69, pp 54-66.

Molteni, D, and Colagrossi, A, (2009). “A simple procedure to improve the pressure evaluation in hydrodynamic context using the SPH”, *Computer Physics Communication*, Vol 180, pp 861-872.
Monaghan, JJ, and Kos, A (1999). “Solitary Waves on Cretan Beach”, *Journal of Waterway, Port, Coastal and Ocean Engineering*, Vol 125, No 3, pp 145-154.

Ozbulut, M, Yildiz, M. Goren, O, (2014). “A numerical investigation into the correction algorithms for SPH method in modeling violent free surface flows”, *International Journal of Mechanical Sciences*, Vol 79, pp 56-65.

Pakozdi, C, (2008). “A smoothed particle hydrodynamics study of two-dimensional nonlinear sloshing in rectangular tanks”, PhD Thesis, *Norwegian University of Science and Technology.*

Shadloo, M, Zainali, A, Yildiz, M. Suleman, A, (2011). “A robust weakly compressible SPH method and its comparison with an incompressible SPH”,. *International Journal of Numerical Methods in Engineering,* Vol 89, No 8, pp 939-956.

Shadloo, M, Zainali, A, Sadek, SH, Yildiz, M, (2011b). “Improved Incompressible Smoothed Particle Hydrodynamics method for simulating flow around bluff bodies”, *International Journal of Computational Methods in Applied Mechanical Engineering,* Vol. 200, pp 1008-1020.

Shao, S, (2009). “Incompressible SPH simulation of water entry of a free-falling object”, *International Journal for Numerical Methods in Engineering*, Vol 59, pp 91-115.

Xu, R. Stansby, P. Laurence, D. (2009) “Accuracy and stability in incompressible SPH (ISPH) based on the projection method and a new approach”, *Journal of Computational Physics,* Vol. 228, pp. 6703-6725.