SPH SIMULATION OF SLOSHING DUE TO HORIZONTAL TANK EXCITATION

D.C. SWAPNADIP* AND S.A. SANNASIRAJ†

Department of Ocean Engineering
Indian Institute of Technology Madras (IIT Madras)
Chennai 600036, India
e-mail: dcswapnadip@googlemail.com*, sasraj@iitm.ac.in†, www.oec.iitm.ac.in

Key words: Smoothed particle hydrodynamics (SPH) Simulation, Dam break, Sloshing.

Abstract. Smoothed particle hydrodynamics (SPH), a true mesh less method has been used to simulate the sloshing problem. The tank motion is restricted to regular horizontal excitation. A number of issues have been addressed while making attempts for simulating with higher h/l ratio. The principle approaches undertaken in order to understand the effectiveness of the standard method have been mostly laid upon different boundary modeling techniques and the artificial viscosity models. Selected method for the time integration scheme has been found to play a significant role. The classical dam break problem has been revisited based on those findings. Model predictions for this case have been compared with few other different CFD techniques as well as experimental data as available in literature. Simulation of tank sloshing has been compared with FEM simulation where the entire mode of investigation is different. The fine tuned values of different controlling parameters have been reported.

1 INTRODUCTION

The SPH method was independently developed by Lucy [1] and Monaghan [2] for solving problems in astrophysics. Soon it attracted the research community interested in investigating fluid dynamical problems with its robustness, ease in understanding and inherent non-necessity of following any grid structure. It has been continuously gaining considerable attention from researchers working on particle based methods. Followed by the rigorous works of Monaghan and Kos [3], Dalrymple and Rogers [4] and, Colagrossi and Landrini [5], it has been understood that SPH is an attractive option to simulate water waves when there is a flow separation. Following the successful application of simulating nonlinear free surface waves, SPH has been adopted to simulate sloshing waves . Souto-Iglesias et al. [6] calculated the moment imparted by the inner liquid on the tank wall by using SPH and compared the free surface shape obtained by the same with experimental data. Delorme et al. [7] examined the impact pressure and studied phase lags with respect to the sloshing frequency using both SPH as well as experiments. Both of these last two mentioned studies were performed for the 2D rolling case. Apart from satisfying a number of agreements with experiments which were otherwise would be difficult to observe through conventional methods it triggered certain issues to be investigated further. A number of algorithms were proposed to adopt along with the basic SPH formulation in order to mitigate the deficiencies. The present study examines some of the concepts to successfully simulate sloshing in a container. The tank motion is restricted to regular horizontal excitation.

2 SPH FORMULATION

The foundation of SPH is based on the theory of integral interpolation. It is based on the mathematical representation that any field variable may be decomposed into several differential parts representing the value of the same at that point interpolated from the surrounding set of points. The intensity of an individual interpolation in that process is controlled by a properly chosen kernel function. Therefore in a continuous field, a field variable $f = f(x)$ may be expressed as

$$f(x) = \int_{\Omega} f(x^*) W(x - x^*, h) dx^* \tag{1}$$

Whereas, the derivative of $f(x)$ is written as [8],

$$f'(x) = \int_{\Omega} f(x^*) W'(x - x^*, h) dx^* \tag{2}$$

The field variable f(x) at the point x is obtained from the information of the function at point x^*, h is called the smoothing length which when multiplied by a scale factor k represents radius of influence around the point x. W denotes the kernel function which closely approximate the Dirac-delta function as the smoothing length (h) tends to zero. The choice of the kernel function has been found to be important both in terms of accuracy of the overall interpolation scheme and computational cost. After making few tests on available form of kernel functions, the normalized Gaussian kernel as reported by Colagrossi and Landrini [5] is adopted in the present study based on its better stability properties and efficiency. It is written as,

$$W(R, \delta) = \frac{e^{-(R)^2} - e^{-(\delta/h)^2}}{2\pi \int_0^{\partial} s\left(e^{-(R)^2} - e^{-(\delta/h)^2}\right) ds} \tag{3}$$

where, R = s/h, s = r_{ij}, distance of neighboring particle j measured from the particle in hand i and δ = kh. The computational domain is discretized in terms of particles. Then following the expressions as shown in Eqns. (1) and (2), the continuity and the Euler equation describing a generic flow filed may be written as

$$\left(\frac{d\rho}{dt}\right)_i = \sum_{j=1}^{N} m_j u_{ij} \nabla_i W_{ij} \tag{4}$$

$$\left(\frac{du}{dt}\right)_i = -\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij}\right) \nabla_i W_{ij} + g \tag{5}$$

Here, i is the particle at which the associated properties have been determined, j denotes the neighboring particle, ρ is the density, P is the pressure, m is the mass of a particle, and g(0,-9.8) is the acceleration due to gravity. In both of the above equations, summation is taken over the neighboring particles for a particle for which the calculation of the desired properties is required. Assuming the fluid medium as weakly compressible, the pressure term appearing in Eqn. (5) is obtained from density as [9],

$$p = B\left[\left(\frac{\rho}{\rho_o}\right)^{\gamma} - 1\right] \tag{6}$$

The ordinary differential equations thus obtained are integrated with respect to the time for a given interval and the particle positions are updated in subsequent time steps.

3 MODELLING BOUNDARY

There are a number of procedures available for modeling boundaries in SPH. Here, the treatment on rigid boundaries is discussed. Yet, particles are seldom found to go beyond the prescribed domain and thereby collapsing the simulation. The main inconvenience comes from the lack of particles to be taken into account for the interpolation process for a field variable near the rigid boundary. In the present study, three methods for modeling the boundary have been tested as described below.

3.1 Dalrymple dynamic

This method was firstly proposed by Crespo et al [10]. The rigid boundary around the domain is defined by boundary particles set in a staggered way. Then they are made to interact with the fluid particles with the constraint on the governing equations. But those particles are not evolved in time as predicted by the momentum equation. Either they are kept fixed in their original position or moved according to an independently prescribed motion. In this approach, an opposing force is imparted by a boundary particle on an approaching SPH particle following Eqn. (6). For shorter time simulation, this approach has been found to be suitable to model the rigid boundary. However, for longer time simulation where there exist rapid particle movements around the boundary resulting impact loads, the boundary particles could not resist the interior particles to leave the domain.

3.2 Lennard-Jones type repulsion force

In order to assure no leaking of particles in a true sense, Lennard-Jones type repulsion force, an artificial force on the boundary particles has been imposed. This approach has been mostly followed in SPH simulation. In the present study, the Lennard-Jones type force has been used. It is written as

$$F = \begin{cases} D\left[\left(\frac{r_0}{r_{ij}}\right)^{n_1} - \left(\frac{r_0}{r_{ij}}\right)^{n_2}\right]\frac{x_{ij}}{r_{ij}^{\;2}}, & if\ r_{ij} < r_o \\ 0 & , otherwise \end{cases} \tag{7}$$

The parameters r_o which denotes the limit in distance from the boundary over which this force becomes activated and the constants D, n_1 and n_2 control its effectiveness. The actual values of them have been found to be mostly problem dependent. Too much repulsion would further resulted in the distortion of the flow field.

3.3 Ghost particles

Ghost particles (GP) are mirror reflections of SPH particles appearing in the vicinity of a rigid boundary. The approach taken in order to implement this procedure resembles the same adopted by Colagrossi and Labdrini [5]. The key idea is to arrange the flow properties of the GPs in such a way that while taking summation for a nearby SPH particle using the GPs as neighbors, it leads to an approximate satisfaction of the free slip condition near the boundary. Most of the initial simulation carried out in our study are using GPs. However, similar to the findings of Delorme et al. [7], there have been some event of particle leaking. So, the above mentioned Lennard-jones type repulsion force has been incorporated with properly calibrated value of the control parameters.

4 THE ARTIFICIAL VISCOSITY TERM

The following artificial viscosity term Π appearing in Eqn. (5) is used in the following form [3].

$$\Pi_{ij} = \begin{cases} \dfrac{-\alpha \overline{c_{ij}} \mu_{ij} + \beta \mu_{ij}^2}{\overline{\rho_{ij}}}, & v_{ij} \bullet r_{ij} \leq 0 \\ 0, & v_{ij} \bullet r_{ij} > 0 \end{cases} \tag{8}$$

where, $\mu_{ij} = \dfrac{h v_{ij} r_{ij}}{r_{ij}^2 + \eta^2}$, $r_{ij} = r_i - r_j$, $v_{ij} = v_i - v_j$, $\overline{c_{ij}} = \dfrac{c_i + c_j}{2}$, $\overline{\rho_{ij}} = \dfrac{\rho_i + \rho_j}{2}$, $\eta^2 = 0.01h^2$, and α is a parameter which is generally taken in between the range of 0-0.1. In most of the test cases, the value of β is taken as zero. Ci and Cj are the numerical sound speeds associated with a particle and calculated according to the particle density at that instant. V is the velocity vector corresponding to that particle.

5 DENSITY REINITIALIZATION

It is well known that the pressure field suffers from huge numerical oscillation in SPH simulation. The main reason behind this is the inability of the chosen kernel function to retain a certain range of consistency as the model proceeds in time. Therefore the density field which takes a direct control over the pressure field is periodically initialized as [5],

$$\left(\rho^{new}\right)_i = \frac{\sum_{j=1}^{N} m_j W_{ij}}{\sum_{j=1}^{N} \frac{m_j}{\rho_j} W_{ij}} \tag{9}$$

The above method referred as Shepard filtering helps to damp out numerical noise in the pressure field. But its effectiveness has been found to be limited in long run. Beyond that it hardly plays any role. So, in the simulation reported herein, the maximum termination time has been fixed up to 10s.

6 TIME INTEGRATION SCHEME

Fourth order Ruge-Kutta (RK4) method along with a dynamic choice of time step has been used to integrate the Eqns. (4) and (5) with respect to time. The time step size (dt) is made as given by Colagrossi and Landrini [5] in order to satisfy the Courant Friedrichs Levy-type condition as

$$dt = \theta \min_j \left(\frac{h}{c_{s,i} + \sigma_i} \right) \tag{10}$$

$$\sigma_i = \max_j \left| h \frac{\left(u_j - u_i\right) \bullet \left(x_j - x_i\right)}{\left|x_j - x_i\right|^2} \right| \tag{11}$$

The value of θ is taken as 2.5 for RK4.

7 RESULTTS AND DISCUSSION

7.1 Dam break

The classical dam break problem has been analyzed to understand the influence of various dictating parameters in SPH simulation. The problem is shown schematically in Fig. 1. Here a dam is represented by a vertical column of water retained by a gate initially. Then the gate is released instantaneously. The test case has been carried out for a domain length (L) of 1m. Fig. 2 shows snap shots for particle configurations taken at different instant of time. Table 1 presents the different input parameters adopted in the dam break simulation.

Table 1: Dam break: input data

	Dam break
Maximum expected flow velocity (Vmax)	$2\sqrt{gh}$
α , β as in Eqn. (8)	1.8, 2.03
D as in Eqn. (7)	$0.5\,V_{max}^2$
n_1 , n_2 as in Eqn. (7)	12, 6
r_o in Eqn. (7)	$0.25dx$
Smoothing length (h)	$0.66dx$

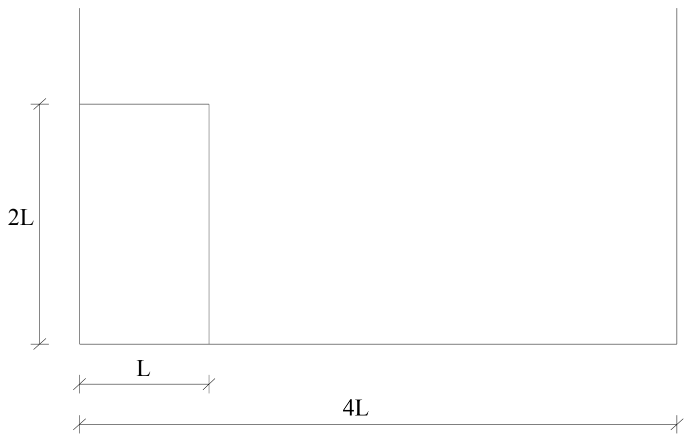

Figure 1: The problem geometry for the dam beak problem

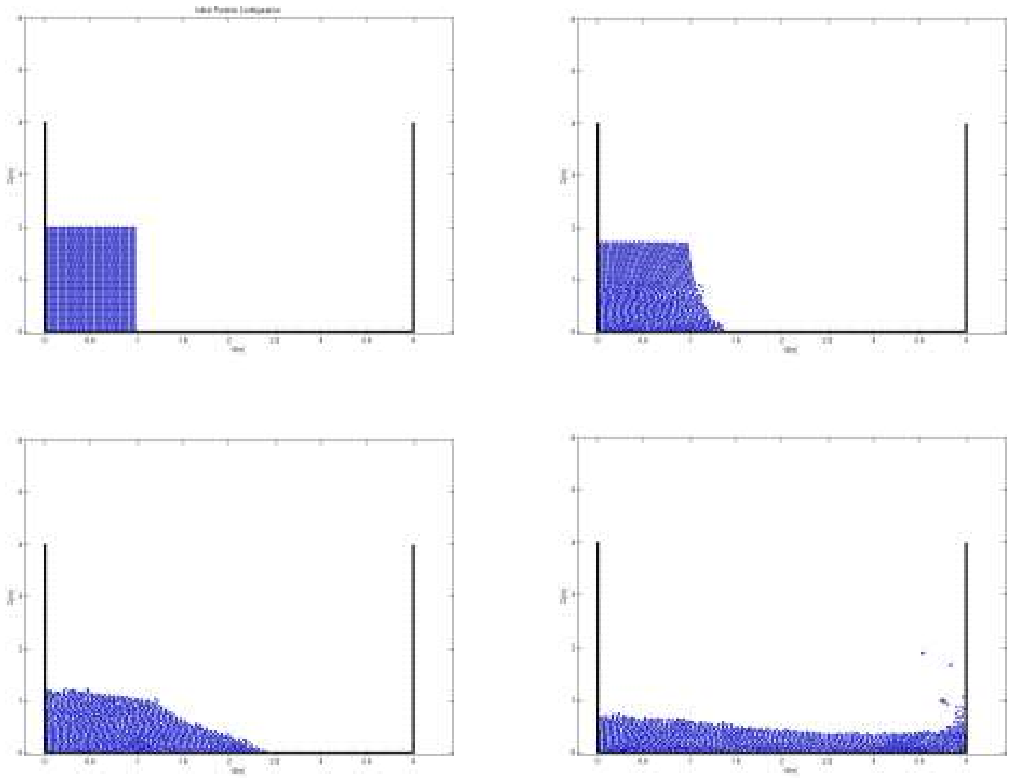

Figure 2: Snap shots showing particle configurations taken at every 0.25 s intervals for the dam break problem

Fig. 3 shows the plot of normalized position of the water front measured along the horizontal axis against normalized time ($t_n = t\sqrt{2g/l}$). It also shows a comparison with MPS [12] and VOF [14] simulation results as well as experimental measurements [13]. The comparison of present SPH simulation with the experimental measurements is found to be good.

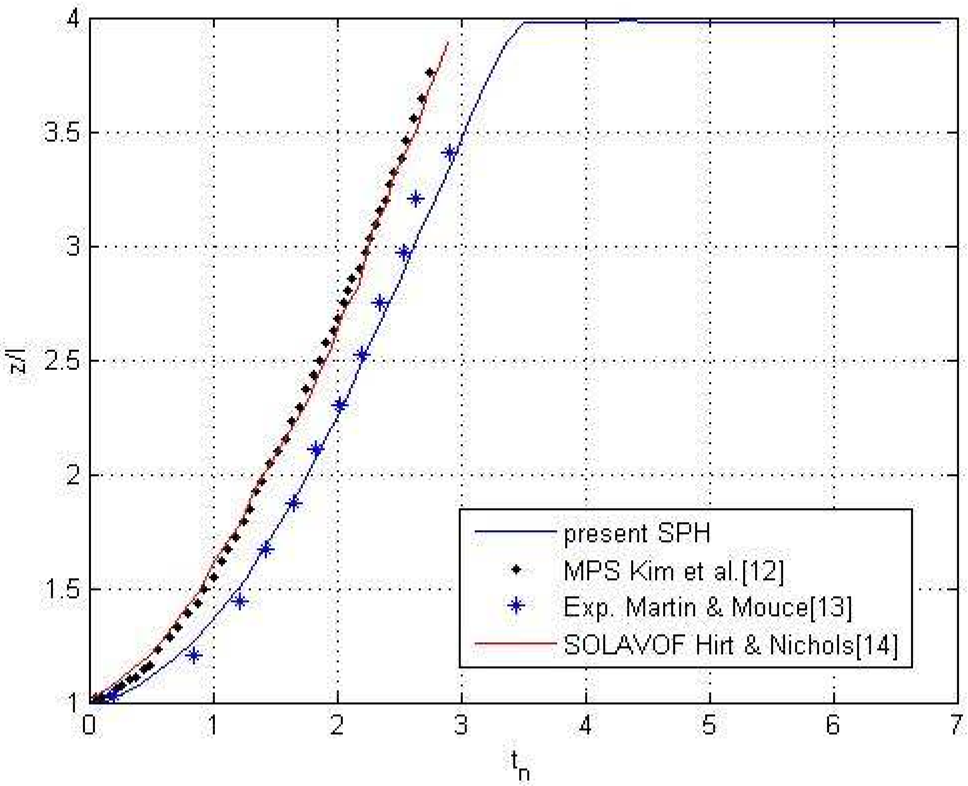

Figure 3: Comparison for the dam beak case with other methods and experiment.

7.2 Tank sloshing The sloshing in a rectangular tank subjected to regular horizontal excitation has been simulated The problem domain is shown in Fig. 4 along with the coordinate system. The tank excitation, X(t) is prescribed by

$$X(t) = a_h \cos(\omega_h t) \tag{12}$$

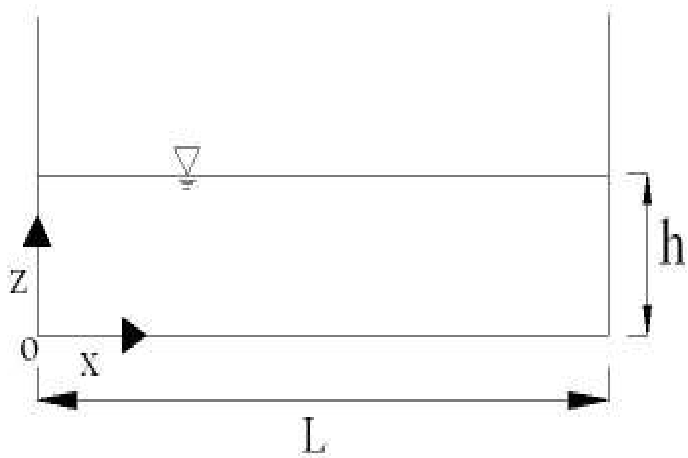

Figure 4: Sloshing tank

Here, a_h is the amplitude of oscillation and ω_h is the excitation frequency. The different input parameters used to follow a particular test case so as to compare with FEM simulation [11] are given in Table 2.

Table 2: Input parameters for the sloshing problem

Length of the tank (L)	2.0 m
Still water depth (h)	1.0 m
Amplitude of oscillation (a_h)	$0.05 \times h$
Excitation frequency (ω_h)	$0.7 \times \omega_1$

Here, ω_1 is the first mode of natural frequency written for the generic modes as

$$\omega_n = \sqrt{gk_n \tanh\left(k_n h\right)} \quad n = 1,2,3.. \tag{13}$$

$k_n = n\pi / L$, is the wave number.

Table 3 presents the different input parameters adopted during SPH simulation. Here *dx* refers to the initial particle spacing.

Table 3: Tank sloshing: input data

	Sloshing
Maximum expected flow velocity (Vmax)	$\sqrt{gh}$
α , β as in Eqn. (8)	0.01, 0.00
D as in Eqn. (7)	$0.05\, V_{\max}^2$
n_1 , n_2 as in Eqn. (7)	12, 6
r_o in Eqn. (7)	0.25*dx*
Smoothing length (h)	1.33*dx*

Fig. 5 shows snap shots of particle configuration taken at every 1.5 s intervals starting from the initial configuration for the above problem.

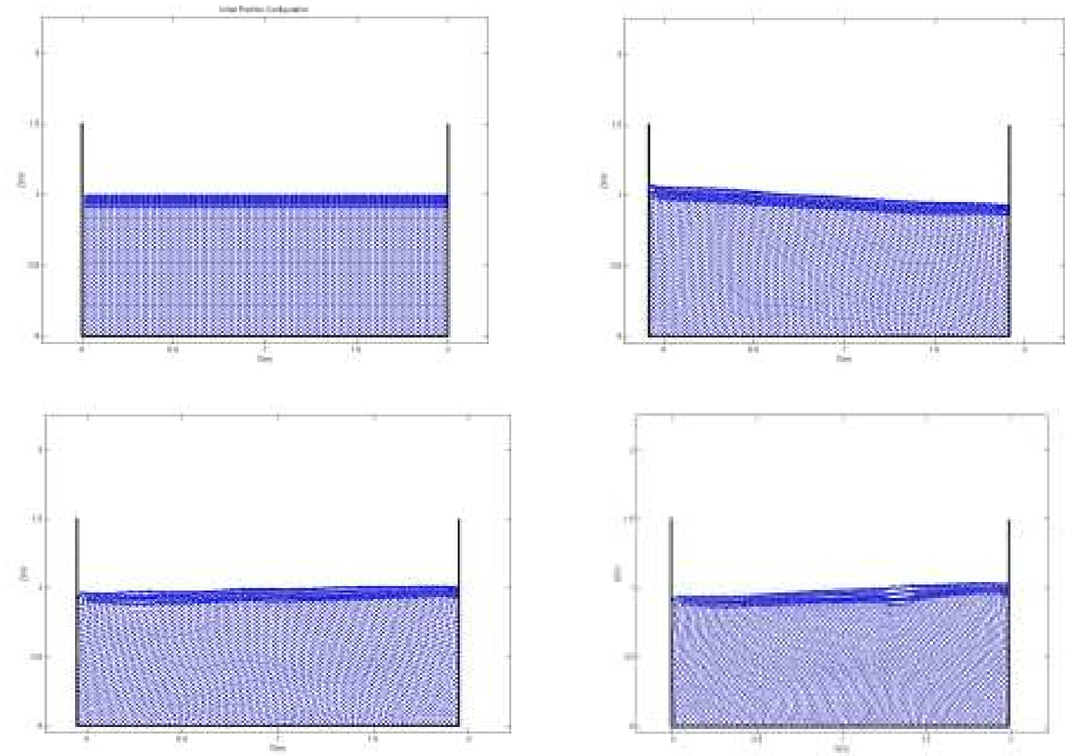

Figure 5: Particle snapshots taken after every 1.5s intervals starting from the initial configuration.

Now, the time history of the free surface elevation at the top left corner of the tank is compared with Sriram et al. [11] which was based on fully nonlinear potential flow theory in Fig. 6.

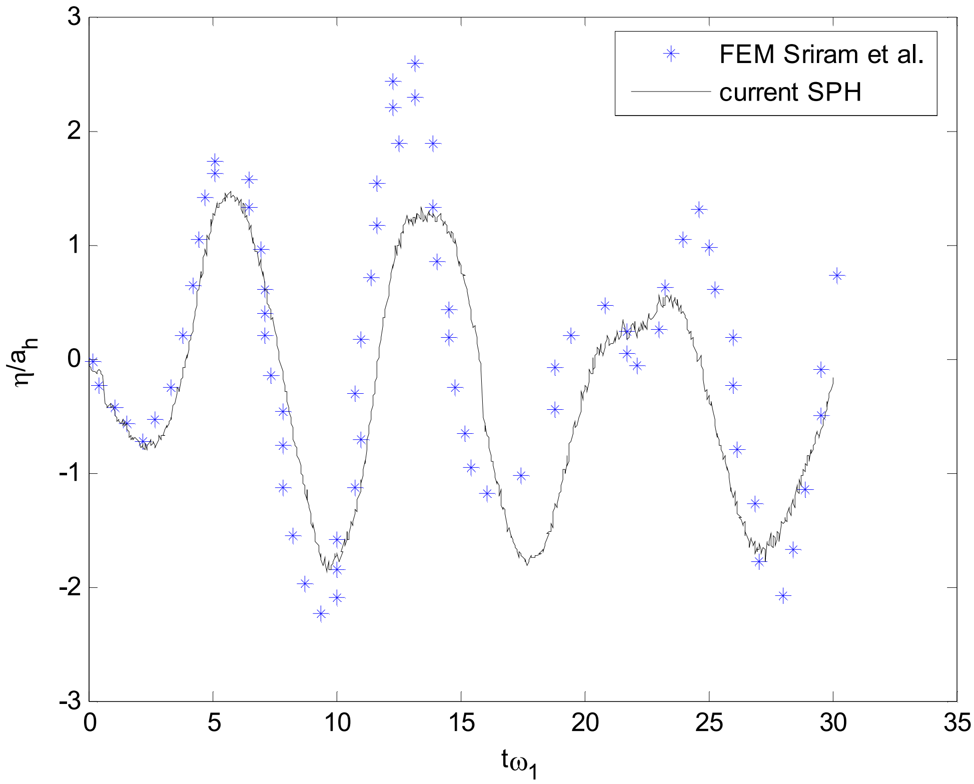

Figure 6: Comparison of time history at the top left corner with the same obtained from FEM

The SPH simulation of free surface time history at the left wall of the tank shows good comparison with the fully nonlinear potential flow theory simulation upto 3s. Beyond that the particles have been found to get scattered to such an extent that affects the local interpolation process for calculating the field variables of a particle. Then the solution sought after the model has been found to diverge from the actual values. In overall, it can be seen that the energy level decreases continuously over the period of simulation. Numerical damping is the main factor behind such variation. Due to the artifical increase of the damping, there is a phase shift. Even though, numerical damping is believed to be a common problem for any numerical method which solves the Navier-Stokes equation, SPH has shortcomings by introducing such numerical damping through various factors such as smoothing length and artificial viscosity. Yet, it motivates the authors to investigate further over these issues.

8 CONCLUSIONS

In recent times it has been well known that the difficulties associated with grid based methods to capture a wave breaking or rapidly changing free surface are dealt through mesh less methods like SPH. But it suffers from lack of accuracy prior to non-breaking as when compared with well established methods. This paper checks SPH in its standard form and reports its output in long time simulations. The present study focuses on the simulation of sloshing in a rectangular tank forced by regular wave excitation using SPH. The various factors that affect the simulation process have been tested. A proper control of input parameters has been found to improve the accuracy. However, a number of unresolved issues have been observed which require further investigations.

9 REFERENCES

[1] Lucy, L. B. Numerical approach to testing the fission hypothesis, *CRC Press,* New Yor*k*, (1977)

[2] Monaghan, J. J. An introduction to SPH. *Computer Physics Communications* (1988) **48**:89-96.

[3] Monaghan, J.J. and Kos, A. Solitary Waves On A cretan Beach. *Journal of Waterway, Port and Ocean Engineering* (1999)**125**:145-153.

[4] Dalrymple, R.A. and Rogers, B.D. Numerical modeling of water waves using the SPH method. *Coastal Engineering* (2006) **53**:141-147.

[5] Colagrossi, L. and Landrini, M. Numerical simulation of interfacial flows by smoothed particle hydrodynamics. *Journal of Computational Physics* (2003)**191**: 448-475.

[6] Souto-Iglesias, A., Delorme, L., Perez-Rojas, L. and Abril-Perez, S. Liquid moment amplitude assessment in sloshing type problems with smooth particle hydrodynamics. *Ocean Engineering* (2006)**33**: 1462-1484.

[7] Delorme, L., Colagrossi, L., Souto-Iglesias, A., Zamora-Rodriguez, R. and Botia-Vera, E. A set of canonical problems in sloshing, Part 1: Pressure field in forced roll-comparison between experimental results and SPH. *Ocean Engineering*(2009)**36**: 168-178.

[8] Liu, G. R. and Liu, M. B. *Smoothed Particle Hydrodynamics-a meshfree particle method*. World Scientific Singapore, (2003).

[9] Batchelor, G. K. *An introduction to fluid dynamics*. Cambridge University Press. U.K, (1974).

[10] Crespo, A. J. C., Gomez, G. M. and Dalrymple, R.A. Boundary conditions generated by dynamic particles in SPH methods. *Computers, materials and continua* (2007) **5(3)**173-184.

[11] Sriram, V., Sannasiraj, S.A. and Sundar, V. Numerical Simulation of 2D sloshing waves due to horizontal and vertical random excitation. *Applied Ocean Research* (2006) **28**19-32.

[12] Lee, B., H., Park, J., C., Kim, M., H., Jung, S., J., Ryu, M., C. and Kim, Y., S. Numerical simulation of impact loads using a particle method. *Ocean Engg.* (2010) **37**:164-173.

[13] Hirt, C.W., Nichols and B.D. Volume of fluid (VOF) method for dynamics of free boundaries. *Journal of Computational* Physics (1981)**39**:201–225.

[14] Martin, J.C. and Mouce, W.J. An experimental study of the collapse of liquid columns on a rigid horizontal plane. *Philosophical Transactions of the Royal Society of London* (1952) **Series A**:244-312.