SPH Simulations of Floating Bodies in Waves

M. DORING, G. OGER, B. ALESSANDRINI, P. FERRANT
Fluid Mechanics Laboratory (CNRS UMR6598),
Ecole Centrale de Nantes
mathieu.doring@ec-nantes.fr

Abstract

Water entry of a solid through a free surface is of interest in ship hydrodynamics applications, namely to study ships behaviour in slamming cases for instance. The study presented here deals with the introduction of a new method based on SPH, and aiming at an accurate numerical prediction of the free motion of a body in a free surface. The validation case exposed here is the water entry of a massive wedge, and comparisons with experimental data are provided. Then simulations involving a box and a pierced box in interaction with waves illustrate the flexibility of the method.

Introduction

Recent advances in the field of free surface flows allowed the computation of breaking and reconnection of interface through the development of interface capturing methods such as Level Set or Volume Of Fluid. Nevertheless the inclusion of a free solid in this kind of approach remains rather complicated, due to the need of a specific treatment of the solid. "Smooth Particles Hydrodynamics" is a new compressible lagrangian method whose flexibility and robustness allow to solve complex free surface flows [1] [2] [3]. Concerning the computation of free solid motion, SPH avoids problems related to mesh managing but a numerical method to evaluate loads on solid boundaries had to be developed. This has been achieved through a new method consisting in calculating forces on the solid from fluid flow characteristic (Pressure, Velocity). To evaluate the accuracy of this new scheme, some results concerning a wedge water entry are provided, including comparison of dynamic condition (accelerations) with experimental data. Once compared, numerical results seem to be in good agreement with experimental data. To illustrate the possibilities of this new scheme, two simulations involving a box and a pierced box in interaction with waves are also presented, confirming the ability of SPH method to simulate complex interactions between water and a free body in large free surface deformations, including coupled interior-exterior fluid-solid computations.

SPH solver

SPH methods are based on a set of interpolating points which are chosen in the medium. Using an interaction function (Kernel function), these points can be used to discretise partial differential equations without any underlying mesh. For free surface flows, the equations to be solved are Navier-Stokes equations (1, 2) and an equation of state for the pressure which is called Tait's equation (3). The use of this equation of state allows to avoid an expensive resolution of Poisson equation. Incompressible flows are obtained as weakly compressible flows: if the Mach number remains below 0.1 during the whole simulation, the flow can be regarded as incompressible.

$$\frac{d\vec{v}}{dt} = \vec{g} - \frac{\nabla P}{\rho} + \overrightarrow{a_{viscous}} \quad (1)$$

$$\frac{d\rho}{dt} = -\rho.\nabla.\vec{v} \quad (2)$$

$$P = \kappa((\frac{\rho}{\rho_0})^7 - 1) \quad (3)$$

$$W(q = \frac{|\vec{r}|}{h}) = C \begin{cases} \frac{2}{3} - q^2 + \frac{1}{2}q^3 & \text{if } 0 \leq q < 1 \\ \frac{1}{6}(2-q)^3 & \text{if } 1 \leq q < 2 \\ 0 & \text{else} \end{cases} \quad (4)$$

The kernel function which tends to a Dirac distribution, is used to discretise previous equations through a convolution with the variables (velocity, pressure, density...) . In the following, the center of the kernel will be identified to a particle. Then the values of a function and its gradient can be determined in the following way:

$$f(\vec{r}) \approx \int_D f(\vec{x}) W(\vec{r} - \vec{x}) d\vec{x} \qquad \nabla f(\vec{r}) \approx \int_D \nabla f(\vec{x}) W(\vec{r} - \vec{x}) d\vec{x} = \int_D f(\vec{x}) \nabla W(\vec{r} - \vec{x}) d\vec{x}$$

The second equation is obtained via integration by parts by neglecting the surface term which is null for interior particles as the chosen kernel in this paper function cancels itself for $\frac{|\vec{r}|}{h} \geq 2$ as is the cubic spline kernel introduced by Monaghan (eq 4), where C is a constant set to ensure $\int W = 1$. This kind of discretisation is second order in space, and to enhance the numerical performance, such as conservation of linear momentum, the formulae are symmetrized [4] leading to the following scheme, where i-subscripted variables correspond to the i^{th} particle:

$$\frac{d\vec{x_i}}{dt} = \vec{v_i}$$

$$\frac{d\vec{v_i}}{dt} = \vec{g} - \Sigma_j\, m_j\, (\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2}) \nabla W(\vec{r_i} - \vec{r_j})$$

$$\frac{d\rho_i}{dt} = -\Sigma_j\, m_j\, (\vec{v_i} - \vec{v_j})\, \nabla W(\vec{r_i} - \vec{r_j})$$

In order to avoid a centered scheme which would lead to numerical instability, an artificial viscosity term is added following Monaghan ([4]). This ordinary differential equation system can be integrated in time by schemes such as Runge-Kutta, Leap-Frog, Predictor-Corrector to ensure at least second order convergence in time. A comparative study with Runge-Kutta methods of different orders proved the advantages of using methods of order greater than two: the lost in CPU time due the increasing number of second member evaluations is canceled by the increasing size of the usable timestep, this resulting in a decreasing total CPU time. In this paper, a third order Runge-Kutta scheme was used.

To this point, a standard SPH scheme has been presented. This scheme has been enhanced to be able to deal with solids in free motion. It can be found in SPH related litterature some applications of SPH scheme to the specific case of wedge water entry [5]. However in the paper [5] the approach is quite different: either the wedge is supposed to be deformable and is modeled by SPH particles with a specific equation of state which describes metal behaviour or the motion of the wedge is imposed. In the case of deformable wedge, the computation of deformations of the body through the motion of solid particles leads to very small and restrictive time steps.

To evaluate efforts on an undeformable body, a numerical method had to be developed. This means :

- evaluation of forces on the solid boundaries: the pressure is interpolated from the water particles which are located in the neighborhood of the solid.
- integration of the pressure effort along the solid boundaries: this is done through a low order trapeze-like rule.
- updating of solid position and velocity: given accelerations on the solid, position and velocity are updated together with flow features using an ODE integrator (third order Runge-Kutta in this paper).

As Euler equations are to be solved, viscous terms are neglected in this paper, thus only pressure efforts are of interest here.

Wedge water entry

These methods will be applied to the standard validation test case of a free-fall impact of a wedge. The experimental device is described in figure 1. This experiment is documented in [6] . At t = 0 s, this free-falling wedge is dropped from 0.61 meters above the free surface with a five degrees clockwise initial heel angle . Its knuckle angles are both fitted with accelerometers dedicated to the measurement of angular and vertical accelerations. After a free falling in the air, the wedge enters the free surface. This impact generates a wide deformation of the free surface and imposes a large vertical deceleration as well as a transverse self-righting of the solid. Experiments are supposed to be realized so that no reflexion of pressure waves interacts with the wedge and the flow can be regarded as two-dimensional. In the SPH simulation, the numerical set-up has been adapted : the tank size has been chosen to ensure no interaction between the wedge and the sound wave generated by the impact.

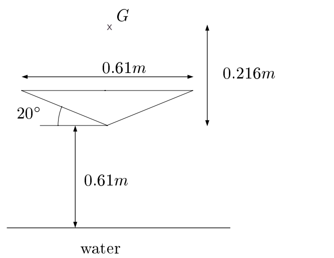

Figure 1: Experimental device

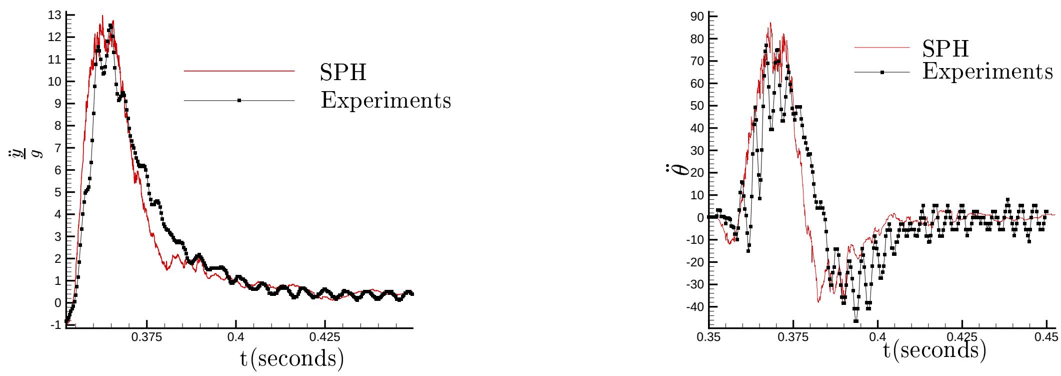

Figure 2: Temporal evolution of the wedge vertical and angular accelerations

As can be seen in figure 2, the temporal evolution of vertical acceleration of the wedge is well predicted in comparison with experimental data. More precisely, the maximum load at t=0.365 s seems to be accurately evaluated in time as well as in amplitude in the SPH simulation. It should be noticed that since air is not taken into account in this SPH simulation, some features of the flow in early stage of the impact are not well captured. Indeed simulations with a finer resolution show some differences in the behaviour of the solid: at the very beginning of the impact, the slope of the temporal evolution of vertical acceleration and its time synchronisation do not converge to experimental result. Nevertheless, early results of biphasic SPH simulations confirm the origin of these differences to be due to air influence. In figure 2, the global behaviour of the solid in terms of angular acceleration is well evaluated in spite of differences between t=0.375 s and t=0.385 s when SPH simulation seems to be in advance in comparison with experiments. This could be due to a lack of resolution in discretisation, leading to a crude description of jets which seems to affect the angular acceleration at this specific instant. Concerning accelerations, some noise can be noticed on both experimental data and SPH simulation. In experiments, this is due to structural vibration [6], whereas in the SPH computation this can be explained by temporal fluctuations of values used to evaluate the loads on the wedge.

Waves in interaction with a box

The results obtained above with this free falling wedge confirm the ability of this method to predict accurately forces on solid boundaries. It seemed to be interesting to try some different test cases using the same scheme,namely the free motion of a box in waves.

Numerical Setup description

The numerical setup is described in figure 7: on the left of the basin, waves can be generated by a piston wavemaker whose motion is prescribed by $X(t) = A\,sin(\omega\,t)$ for $t > t_{delay}$ with $A = 0.03572\,\mathrm{m}$ and $\omega = 6.5222\,rad/s$ waves are damped on the right by a slopping beach. At t=0 s, this squared box is launched right above the free surface. Its density is $500kg/m^3$ and for this condition, a simple stability study shows that the natural equilibrium of the box is 45° turned from its initial position. In the numerical result presented here, the box is first launched in calm water.Then, after a short delay of three periods allowing the box to converge towards its static equilibrium, waves are generated acting on this free solid. The figure on the top left hand corner shows clearly a good convergence towards the final angular position expected (natural equilibrium position). Temporal evolution of angular and linear position due to wave influence seems to be qualitatively correct: no instability has been noticed within the motion of the box.

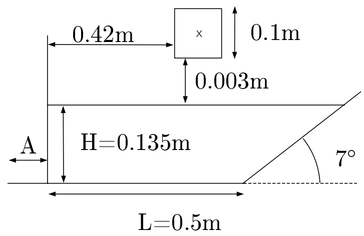

Figure 3: Numerical setup

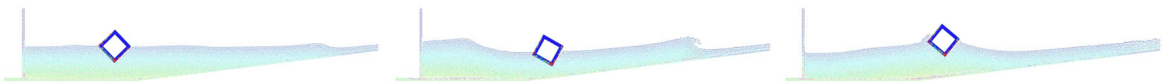

Figure 4: $t = 2.6$ s, $t = 5.9$ s and $t = 6.2$ s

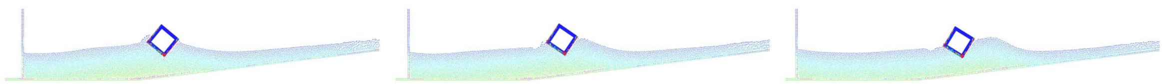

Figure 5: $t = 6.3$ s , $t = 6.4$ s and $t = 6.5$ s

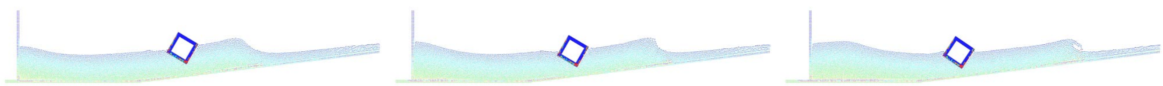

Figure 6: $t = 6.6$ s, $t = 6.7$ s and $t = 6.85$ s

Pierced Box test case

It should be emphasized that this new approach allows the computation of more complex solid fluid coupling: for instance in the pierced box test case, some water will be entrapped within the box, leading to a sloshing motion which will contribute to efforts on the solid. In this simulation, a pierced square is launched right above the free surface, then waves are generated by a piston wavemaker allowing some water to enter the box. To preserve its natural equilibrium to be vertical, this box was preferred to be fitted with a punctual mass (m) located at its bottom (black point in figure 7). The main features are summarized in the table below. As

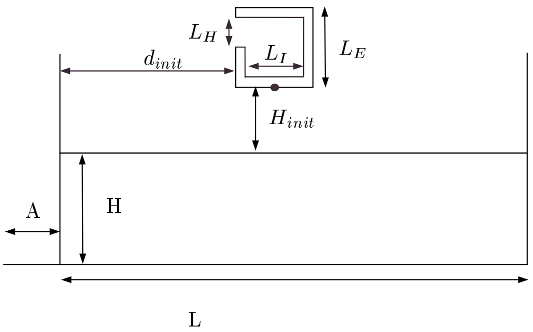

Figure 7: Pierced box numerical setup

A	0.05 m
H	2.0 m
L	7.0 m
H_{init}	0.03 m
d_{init}	2.5 m
L_I	1.0 m
L_E	1.5 m
L_H	0.375 m
m	1050 kg
ω	5.55 rad/s
ρ_{solid}	100 kg/m^3
ω	5.55 rad/s

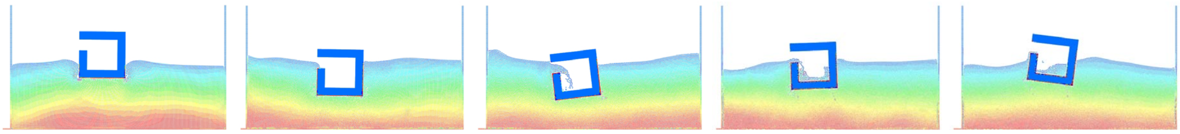

Figure 8: $t = 0.45$ s,$t = 0.95$ s, $t = 1.45$ s, $t = 1.95$ s, and $t = 2.45$ s

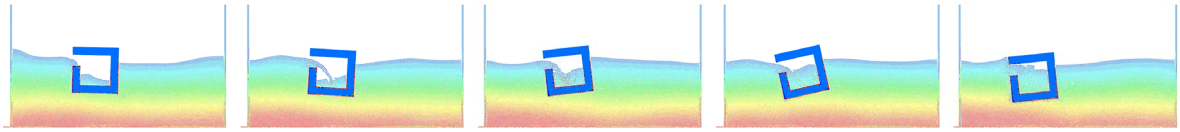

Figure 9: $t = 3.45$ s,$t = 3.95$ s, $t = 4.45$ s, $t = 4.95$ s, and $t = 5.45$ s

in the previous case, this simulation is to be considered as illustrative, since no comparison with experiments is available at the moment. This computation proves the ability of this effort sampling method to treat easily solid-fluid coupling, and opens some new possibilities in the field of free surface flows. Nevertheless in this kind of problems, the presence of air is typically of great influence especially at the very end of the box's filling when an air bubble should be entrapped, leading to modifications of the results.

Conclusion

In this paper, a new method in the field of SPH has been briefly presented which enables the simulation of free motion of solids in interaction with free surface flows. In the standard test case of wedge water entry, results obtained using this approach have been compared to experimental data showing promising agreement. Then, in order to illustrate the possibility of this new method, a simulation involving a box in interaction with waves generated by a piston wavemaker has been shown, including breaking of the waves on a beach. The flexibility of this approach allows coupled interior-exterior fluid-solid simulations as could be seen in the pierced box computation. Further work is still needed concerning the inclusion of air in the SPH solver and the validation of this method on other test cases.

References

[1] J J Monaghan. Simulating free surface flows with sph. *Journal of Computational physics*, 110:399–406, 1994.

[2] Y Andrillon, M Doring, P Ferrant, and B Alessandrini. Simulation d'écoulement à surface libre complexe au moyen de méthodes sph et vof. In *Proceedings des* 9^{emes} *Journées de l'Hydrodynamique*, Mars 2003.

[3] M Doring, Y Andrillon, P Ferrant, and B Alessandrini. Sph free surface flow simulation. In *Proceedings of the* 18^{th} *International Workshop on Water Waves and Floating Bodies*, April 2003.

[4] J J Monaghan. Smoothed particle hydrodynamics. *Annu. Rev. Astron. Astrophys.*, 30:543–574, 1992.

[5] J P Vila and A Rompteaux. Calcul du tossage avec smart fluids un code de calcul particulaire compressible. 1999.

[6] R Petterson L Xu, A W Troesch. Asymmetric hydrodynamic impact and dynamic response of vessels. *Proceedings of 17th International Conference on OFFSHORE MECHANICS and ARCTIC ENGINEERING*, pages 98–320, 1998.

Discusser: Q. Ma

You have presented very impressive results. Could I ask the following questions:

1. how many particles did you use along the depth for problems associated with a wave maker?

2. In the pressure-density model, how to determine the value of K? Is it problem dependent?

Author's reply:

K is defined as $c/7$ where c stands for a chosen nominal sound speed. It is problem dependent, since to approach an incompressible fluid, we have to respect $Ma < 1$. It imposes $(V/c) < 0.1$ (where V is the maximum velocity reached by the fluid). Note that it is equivalent to make the density not to vary more than 5% from the nominal fluid density.

Discusser: M. Kashiwagi

In the last example of a pierced-body motion simulation, what kind of boundary condition did you impose? Is that a solid wall boundary condition or some absorbing boundary?

Author's reply:

We impose solid wall boundary conditions (classical free slip boundary conditions) for all boundaries (water tank as well as pierced-body boundaries).