PRESSURE PREDICTIONS DURING WATER ENTRY OF A 2D RIGID CYLINDER USING SPH METHOD

K.S. Vepa[1], D. Van Nuffel[1], W. Van Paepegem[1]

KameswaraSridhar.Vepa@Ugent.be, diederik.vanNuffel@ugent.be , Wim.VanPaepegem@Ugent.be

[1]Ghent University, Department of Materials Science and Engineering, Sint-Pietersnieuwstraat 41, 9000, Ghent, Belgium

Introduction

Water entry of cylindrical shaped structures is important in the context of loads due to wave impact/slamming on floating buoys used for wave energy conversion [1]. Wave impact or slamming is a phenomenon characterized by high local pressures (10 bar or more) for very short durations (in the order of milliseconds). Slamming loads cause severe damage to the structure [2]. This forms the typical case of fluid-structure interaction between the floating buoys and the water surrounding the buoys [3]. Different numerical approximation methods are available for simulating fluid structure interaction problems. Traditional mesh techniques use nodes and elements for approximating the continuum equations whereas particle methods like smoothed particle hydrodynamics (SPH) approximate the continuum equations using the kernel approximation technique and hence can be used for a wide range of fluid dynamics problems [4].

The SPH method has proven to be successful in modelling the free surface flows with good accuracy [5]. SPH has the characteristics of a lagrangian method. This helps in modelling the breaking of interface accurately. Hence it is applied for modelling the free surface impact problems as in this case [6]. The basic equation solved is the Navier-Stokes equation as given by equation 1. An interpolation scheme as shown in equation 2 can be used for calculating the lagrangian derivatives at each point in the domain D. Finally, numerical integration is done using a quadrature formula like the one shown in equation 3 at every integration point i. Instability is one drawback of this method due to the explicit time integration scheme. To overcome this, an artificial viscosity term is added to the momentum equation. This could be a drawback if used in low-dynamics flows, but for shocks and explosions where, good conservation of energy is not needed, this still gives good results.

$$\frac{\partial V}{\partial t} + (\vec{V}.\nabla)\vec{V} = -\frac{1}{\rho}\nabla P + v\nabla^2\vec{V} \qquad (1)$$

$$f(\vec{r}) \approx \int_D f(\vec{x})W(\vec{r} - \vec{x}, h)d\vec{x} \qquad (2)$$

$$\int_D g(\vec{x})d\vec{x} \approx \sum_i g_i w_i \qquad (3)$$

In this paper, the water entry of two dimensional cylindrical structures is studied using an SPH solver coupled with a Finite Element solver. The SPH solver uses a variable smoothing length that adapts to the varying particle densities. Water is modelled using SPH particles, the cylinder is modelled using finite elements and the contact is modelled using the contact algorithms. Shell elements are used for modelling the cylinder and contact is handled using node to surface contact algorithm. Few models are tested using the coupled solver with different particle sizes with the emphasise on the pressures at the bottom-most point of the cylinder.

Modelling steps

A commercially available explicit code, LS-DYNA is used for modelling the water entry of a cylinder. Figure 1 shows the model and boundary conditions used for this purpose. This code uses a coupled SPH-FE solver for calculating the fluid field variables and also the stresses developed in the cylinder (in case of an elastic cylinder). A bucket sorting method is used for finding the neighbours of a particle. Artificial viscosity is used to take care of the shocks. Time integration is done using the classical first order method. Though it increases the number of particles, a regular SPH mesh is used to reduce the inter-particle discrepancies. The cylinder is allowed to fall under gravity loading with an initial velocity of 4.1 m/s.

Choice of initial velocity is based on the experimental data. Since this is a 2D simulation, the nodal degrees of freedom are constrained in the third direction. The boundaries are also modelled using shell elements and a contact algorithm is employed for modelling the interaction between the container walls and the SPH particles (water).
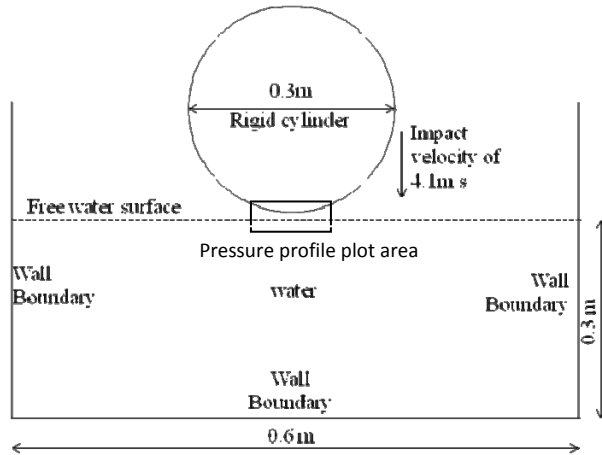

Fig.1: Simulation setup used

Since SPH is a particle based lagrangian method, the water surface is tracked based on the particle position. The smallest particle size of 0.5 mm is used to see if there is any variation from the experimental results and also the velocity of the cylinder in the numerical simulations is compared with that of the experimental data. A total of 1600000 particles are used for the simulation with the particle size of 0.5 mm and the number of particles keep reducing with increasing particle size. Shell elements are used for modelling the rigid cylinder. Shell elements of uniform size (approximately 6 mm) are used for representing the cylinder.

The density of the water is taken as 1000 kg/m^3 and is treated as compressible fluid. Based on the smallest particle size, the time step size is decided using the dimensionless Courant number, which is calculated based on the velocity of the considered.

Results

1.Pressure vs. time

As expected, the peak pressure is found at the bottom of the cylinder and travels along the perimeter of the cylinder. Figure 2 shows the pressure curves as observed in the SPH (water) particles of size 0.5 mm on the top most layer that comes in contact with the cylinder. Initial particle locations are marked in Figure 1.
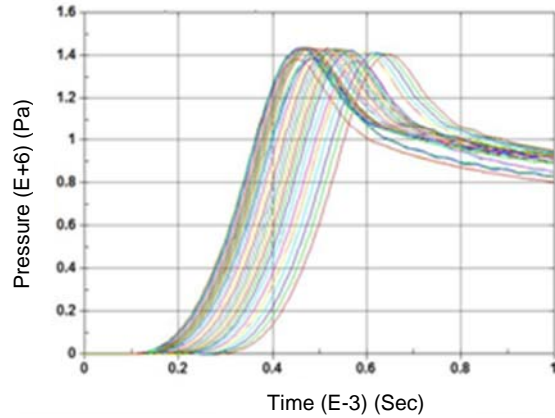

Fig.2: Pressure profiles for SPH simulations with 0.5 mm particle size

Figures 3 and 4 show the pressure profiles for other particle densities viz., 1 mm and 5 mm.
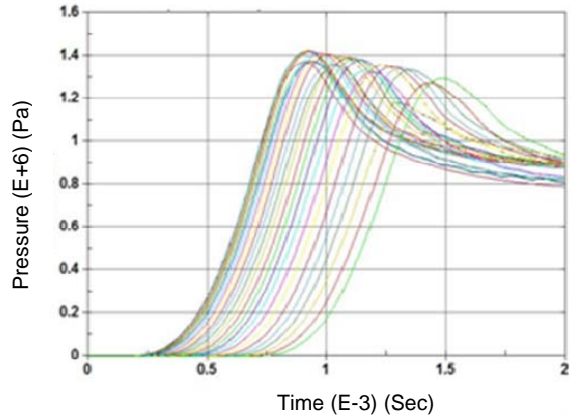

Fig.3: Pressure profiles for SPH simulations with 1mm particle size

It can be observed that there is a large difference between the peak pressures for 5 mm particle size and 1 mm particle size. But the difference reduces from 1 mm to 0.5 mm and approaches the experimental value. Also, it can

be observed that the number of particles in the range reduces with the increasing particle size.
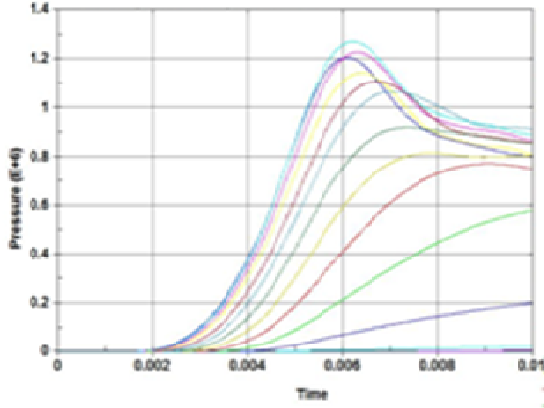

Fig.4: Pressure profiles for SPH simulations with 5 mm particle size

Table 1 below shows the peak pressures as observed in experimental measurements along with the SPH calculations for different mesh sizes.

	Peak pressure (in bar)
Experimental	14.7
SPH (particle size 0.5 mm)	14.22
SPH (particle size 1 mm)	14.1
SPH (particle size 5 mm)	12.8

Table 1: Comparison of peak pressures

The difference in the peak pressure between the SPH simulation with 0.5 mm particle size and the experimental result can be attributed to a number of factors like small difference in density, viscosity and other material properties of water.

2. Computational time and memory

There is a large influence of the particle size or total number of particles on the memory usage and computational time of the SPH simulations. Table 2 below shows the comparative data of the computational time needed for the three different cases under consideration. RAM memory usage is proportional to the number of particles coming within the smoothing length of each particle.

Table 3 shows the RAM memory requirements for the three cases.

In the above three cases, if one selects a particle size of 1 mm instead of 0.5 mm, little difference is observed in the peak pressures calculated but a lot of computational time and RAM memory required is saved.

	Computational time (in hours)
SPH (particle size 0.5 mm)	8.3
SPH (particle size 1 mm)	1.1
SPH (particle size 5 mm)	0.1

Table 2: Comparison of computational times

	RAM Memory usage (in GB)
SPH (particle size 0.5 mm)	4.6
SPH (particle size 1 mm)	1.3
SPH (particle size 5 mm)	0.25

Table 3: Comparison of memory usage

Conclusions

Coupled SPH-FE simulations are carried out to understand the importance of SPH particle size on the computational load as well the accuracy of the results. As expected there is a convergence of pressure results with the increasing mesh sizes and further research is to be done to understand the consistency of these observations at other speeds. Problems faced by traditional CFD hydrocodes to accurately measure the water entry pressure loads is overcome using the coupled SPH-FE simulations. Though SPH simulations take longer calculation times, techniques like variable smoothing length and optimum particle size help in controlling the calculation times without losing the accuracy of the results.

References

1. Zhao, R. and O. Faltinsen, *water entry of 2-dimensional bodies.* Journal of Fluid Mechanics, 1993. 246: p. 593-612.
2. Sebastiani, L., et al., *a theoretical / experimental investigation of the slamming pressures on fast monohull vessels.* FAST-2001, 2001.
3. Agamloh, E.B., A.K. Wallace, and A. von Jouanne, *Application of fluid-structure interaction simulation of an ocean wave energy extraction device.* Renewable Energy, 2008. 33(4): p. 748-757.
4. Monaghan, J.J., *Smoothed particle hydrodynamics.* Reports on Progress in Physics, 2005. 68(8): p. 1703-1759.
5. Oger, G., et al., *Two-dimensional SPH simulations of wedge water entries.* Journal of Computational Physics, 2006. 213(2): p. 803-822.
6. Fang, J., et al., *Improved SPH methods for simulating free surface flows of viscous fluids.* Applied Numerical Mathematics, 2009. 59(2): p. 251-271.