Fluid Flow Modeling with SPH in LS-DYNA®

Edouard Yreux
Livermore Software Technology Corporation

Abstract

A new Smoothed Particle Hydrodynamics formulation for fluid flow modeling has been added in LS-DYNA. A density smoothing algorithm based on kernel density estimation is implemented to correct for the well-known pressure oscillation issue that arises with traditional SPH schemes when modeling fluids. A Weakly-Compressible equation of state is adopted to ensure reasonable timestep restrictions while minimizing the compressibility effects of the fluid. The resulting formulation is particularly suitable for free surface flows and fluid-structure interaction problems. Two and three dimensional validation problems are presented, as well as qualitative comparisons with incompressible CFD results obtained with the ICFD solver of LS-DYNA.

Introduction

While a substantial amount of early SPH structural applications were dedicated to high velocity [1, 2] and hypervelocity [3] impact simulations, there has been a growing interest in fluid flow modeling in recent years. Two common approaches are the incompressible SPH (ISPH) method [4], based on a classical projection method, and the weakly-incompressible SPH method, where the pressure is explicitly computed based on a specific equation of state [5] designed to keep density variations to a minimum. The latter method has been widely employed for free-surface flows simulations [6, 7], liquid sloshing [8], and wave-structure interactions [9], among others. A density reinitialization technique [10] was also developed to alleviate strong pressure oscillations observed in early fluid flow simulations. In this paper, we present the fundamentals of weakly-compressible SPH modeling, how it is implemented in LS-DYNA, and how to use this feature in the software. Numerical examples are provided and comparisons are made with simulations performed in the ICFD solver of LS-DYNA, and with experimental values when available.

Weakly-Compressible SPH

To model fluid flow with SPH in LS-DYNA, only a couple of points differ from regular structural analysis. Following [5], the Murnaghan equation of state [11] can be used to enforce quasi-incompressibility while allowing for a reasonable timestep in explicit calculations. The pressure p at any point in the fluid is expressed as

$$p = k_0 \left[\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right] \tag{1}$$

where ρ_0 is the density of the fluid at rest, γ is a parameter often set to 7, and k_0 is selected such that

$$c_0 = \sqrt{\frac{\gamma k_0}{\rho_0}} \geq 10 v_{\max} \tag{2}$$

where $v_{\max}$ is the maximum expected fluid flow velocity. Satisfying this criterion will allow for minimal variations in density, preserving the quasi-incompressible characteristics of the fluid.

Traditional SPH formulations can exhibit very substantial pressure oscillations when modeling fluid flows. A common approach to overcome this problem is to use a density reinitialization scheme [10]. A Shepard filter is constructed with the SPH kernel functions, and applied to the density field such that

$$\rho_I^{\text{new}} = \sum_J \rho_J \tilde{W}_{IJ} \frac{m_J}{\rho_J} = \sum_J \tilde{W}_{IJ} m_J, \tag{3}$$

where

$$\tilde{W}_{IJ} = \frac{W_{IJ}}{\sum_K W_{IK} m_K / \rho_K}. \tag{4}$$

The resulting density field is much smoother, which in turns produces a smoother pressure field through the equation of state.

A final point of attention is the artificial viscosity employed in the simulation. The momentum equation in SPH can be written as

$$\frac{\mathrm{d}\boldsymbol{v}_I}{\mathrm{d}t} = -\sum_J m_J \left(\frac{\boldsymbol{\sigma}_I}{\rho_I^2} + \frac{\boldsymbol{\sigma}_J}{\rho_J^2} + \Pi_{IJ} \right) \cdot \boldsymbol{\nabla} W_{IJ}, \tag{5}$$

where $\boldsymbol{\sigma}_I$ is the stress tensor at particle I in Voigt notation, $\boldsymbol{v}_I$ is the velocity at particle I, and $\boldsymbol{\nabla} W_{IJ} = \boldsymbol{\nabla} W\left(|\boldsymbol{x} - \boldsymbol{x}_J|/h\right)|_{\boldsymbol{x}\to\boldsymbol{x}_I}$ is the gradient of the kernel function associated with particle J evaluated at particle I. Π_{IJ} is the artificial viscosity term, expressed as:

$$\Pi_{IJ} = \begin{cases} \dfrac{-Q_2 \bar{c}_{IJ} \mu_{IJ} + Q_1 \mu_{IJ}^2}{\rho_{IJ}} & \text{if } \boldsymbol{v}_{IJ} \cdot \boldsymbol{x}_{IJ} < 0 \\ 0 & \text{otherwise} \end{cases} \tag{6}$$

with $\boldsymbol{x}_{IJ} = \boldsymbol{x}_I - \boldsymbol{x}_J$, $\boldsymbol{v}_{IJ} = \boldsymbol{v}_I - \boldsymbol{v}_J$, and

$$\mu_{IJ} = \frac{\bar{h}_{IJ} \boldsymbol{v}_{IJ} \cdot \boldsymbol{x}_{IJ}}{x_{IJ}^2 + 0.01 \bar{h}_{IJ}}, \tag{7}$$

where $\bar{h} = \frac{h_I + h_J}{2}$, and $Q1$ and $Q2$ are user-defined parameters. The default value of these two parameters is suitable for many solid applications, but is too dissipative for most fluid flow simulations, especially for low-viscosity fluids such as water. More fitting values for $Q1$ and $Q2$ are suggested in the following section.

Typical LS-DYNA Input Deck

The density filtering presented in the previous section is implemented in LS-DYNA as a new formulation in *Control_SPH, by setting FORM = 15. The Murnaghan equation of state is also available as a new keyword: *EOS_Murnaghan. Any SPH model part is therefore represented by a *MAT_Null card, describing the density and viscosity of the material, and an *EOS_Murnaghan card, describing the pressure-density relationship.

The default artificial viscosity parameters being too dissipative to model low-viscosity fluids accurately, it is suggested to overwrite their value, using either a *Control_Bulk_Viscosity card, which applies to the entire domain, or an *Hourglass card for each SPH fluid part. It is recommended to set the quadratic bulk viscosity coefficient Q_1 to a value between 0.01 and 0.1, and the linear bulk viscosity coefficient Q_2 to a value of 1.0×10^{-12}, (essentially zero, but if zero is entered, LS-DYNA replaces the coefficient by its default value, which is 0.06). For more violent flows verging on impact simulations, it might be necessary to increase these values to maintain stability of the method.

Numerical Examples

Dam break evolution with wet bottom

In this 2D simulation, a volume of water is initially contained behind a gate, which is opened at $t = 0$. On the other side of the gate, a shallow layer of water is also presented, essentially being impacted by the body of water on the left side of the gate. The problem geometry is given in Figure 1, and some experimental data is available from [12]. Studying the wave formation and propagation proves quite interesting, as the initial wave from the collapsing body of water of the left impacts and collides with the shallow layer of water on the right. Figure 2 shows a comparison between the experiment, an ICFD simulation ran in LS-DYNA, and an SPH simulation using the fluid formulation. Snapshots are shown for $t = 0.156$ s, $t = 0.219$ s, $t = 0.281$ s, $t = 0.343$ s, $t = 0.406$ s, $t = 0.468$ s and $t = 0.531$ s. The agreement between the two numerical methods and with the experiment is very satisfactory. Both ICFD and SPH simulations are color coded by pressure field, and correlate very well between both methods.

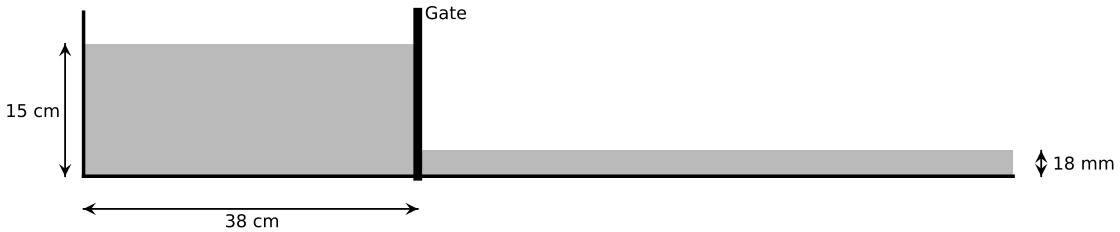

Figure 1: Dam-break evolution, problem geometry.

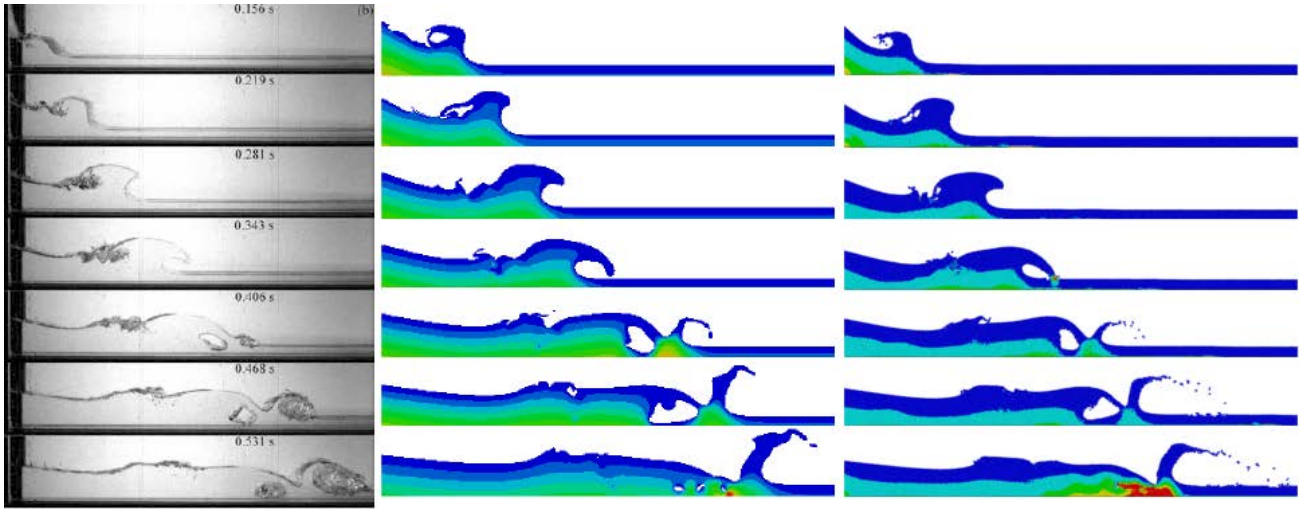

Figure 2: Dam-break evolution, comparison between experiment [12] (left), LS-DYNA ICFD simulation (center), and LS-DYNA SPH simulation (right).

Wave-Structure Interaction

This 3D simulation involves a wave forming, propagating and impacting a square column. The main object of this study is the total force exerted by the fluid on the square structure, and to compare the numerically obtained force to experimental values available in [13]. The geometry of the problem is given in Figure 3, and a snapshot of the simulation is illustrated in Figure 4. The total force experienced by the structure in both the experiment and the numerical simulation are shown in Figure 5. Again, the agreement is quite good, as the simulation properly captures not only the initial impact of the wave, but also the more complex interaction as the wave travels to the back wall, rebounds and flows around the structure. Qualitative comparison with the ICFD solver in LS-DYNA also showed a very similar flow between the two methods.

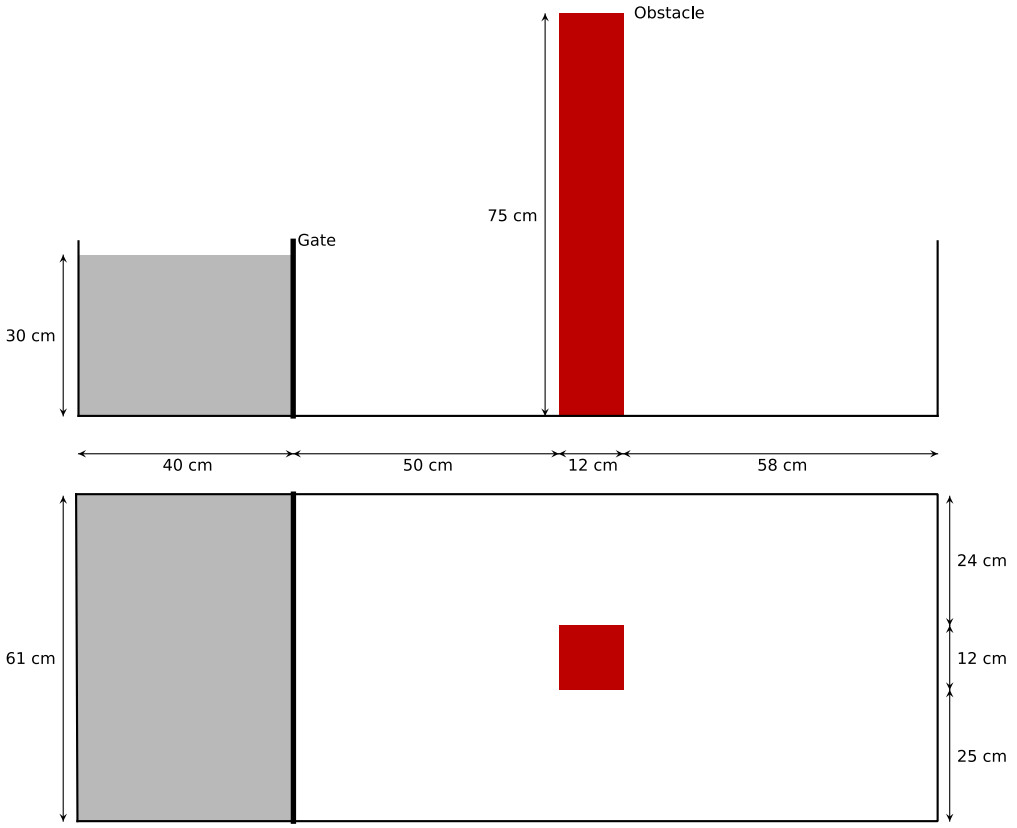

Figure 3: Wave-structure interaction, problem geometry.

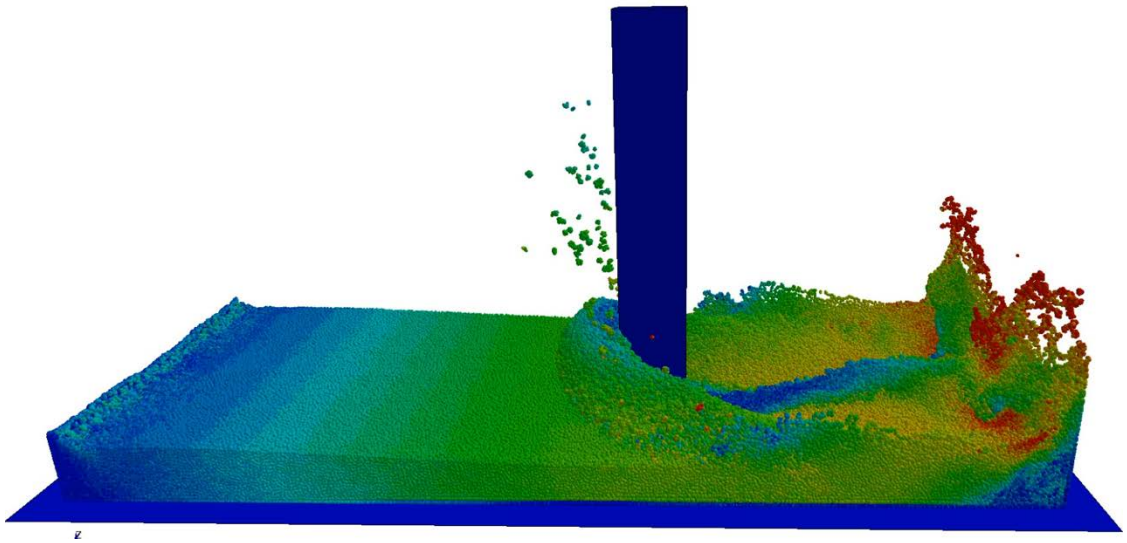

Figure 4: Wave-structure interaction, snapshot of the SPH simulation during impact.

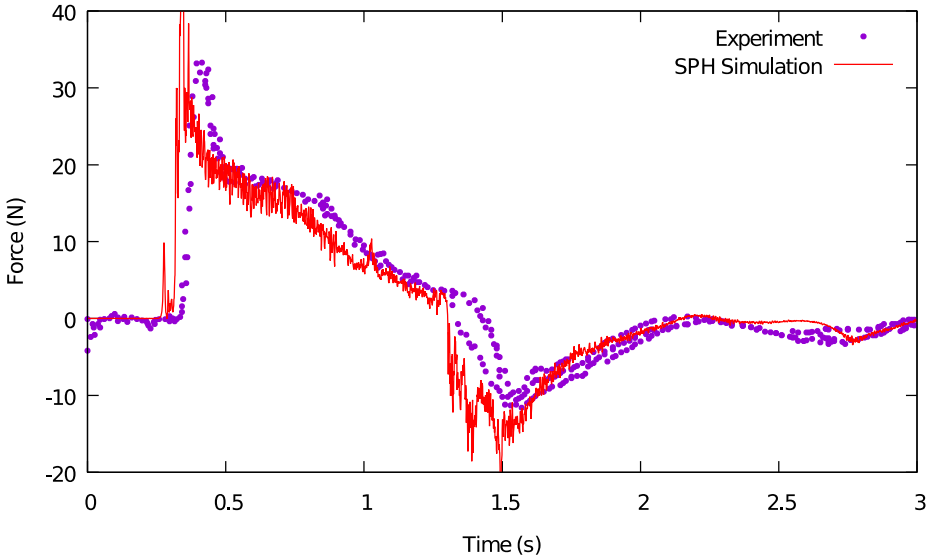

Figure 5: Wave-structure interaction, total force exerted by the water on the structure, comparison between experimental value and SPH simulation.

Conclusion

A new formulation was implemented in LS-DYNA, aimed at modeling fluid flow with the SPH solver. A weakly compressible framework was developed, involving the use of a specific equation of state for efficiency, and of density filtering for accuracy. Practical choices for artificial viscosity parameters were also discussed. The method was validated both in 2D and 3D, and showed good agreement with available experimental data. Comparisons with the ICFD solver in LS-DYNA were also shown.

References

[1] Gordon R. Johnson, Eric H. Petersen, and Robert A. Stryk. “Incorporation of an SPH option into the EPIC code for a wide range of high velocity impact computations”. In: International Journal of Impact Engineering 14.1 (1993).

[2] Gordon R. Johnson, Robert A. Stryk, and Stephen R. Beissel. “SPH for high velocity impact computations”. In: Computer Methods in Applied Mechanics and Engineering 139.14 (1996).

[3] Larry D. Libersky et al. “Recent improvements in SPH modeling of hypervelocity impact”. In: International Journal of Impact Engineering 20.610 (1997).

[4] Sharen J Cummins and Murray Rudman. “An SPH Projection Method”. In: Journal of Computational Physics 152.2 (1999).

[5] J Monaghan and Andrew Kos. “Solitary Waves on a Cretan Beach”. In: Journal of Waterway Port Coastal and Ocean Engineering 125 (May 1999).

[6] Moncho Gomez-Gesteira et al. “State-of-the-art of classical SPH for free-surface flows”. In: Journal of Hydraulic Research 48.sup1 (2010).

[7] Damien Violeau and Benedict D. Rogers. “Smoothed particle hydrodynamics (SPH) for free-surface flows: past, present and future”. In: Journal of Hydraulic Research 54.1 (2016).

[8] J.R. Shao et al. “An improved SPH method for modeling liquid sloshing Hydrodynamics”. In: Computers & Structures 100101 (2012).

[9] M. Gomez-Gesteira et al. “SPHysics development of a free-surface fluid solver Part 1: Theory and formulations”. In: Computers & Geosciences 48 (2012).

[10] Andrea Colagrossi and Maurizio Landrini. “Numerical simulation of interfacial flows by smoothed particle hydrodynamics”. In: Journal of Computational Physics 191.2 (2003).

[11] FD Murnaghan. “The compressibility of media under extreme pressures”. In: Proceedings of the National Academy of Sciences 30.9 (1944).

[12] Imre M. Jánosi et al. “Turbulent drag reduction in dam-break flows”. In: Experiments in Fluids 37.2 (2004).

[13] Todd Silvester and Paul Cleary. “Wave-Structure in Interaction Using Smoothed Particle Hydrodynamics”. (2006)