# Numerical simulation of wedge water entry based on two-dimensional two phase SPH model

Kai GONG, Benlong WANG, Hua LIU
Department of Engineering Mechanics, Shanghai Jiao Tong University, Shanghai 200240, China
Email: hliu@sjtu.edu.cn

The hydrodynamic problem of a two-dimensional wedge entering water is studied based on SPH model. A non-reflection boundary treatment for SPH method is proposed to reduce the size of computational domain. The details of water entry and enclosing are simulated using both single and multi phase SPH model. Good agreement is obtained comparing experimental data.

## Introduction

Water entry is part of the general fluid-structure impact problem in the field of naval architecture. SPH method is attractive on simulating the violent wave impact problems, e.g. Oger et al (2006). Since 2004 some progresses have been made in our group, applying SPH method on water entry, dambreaking, sloshing and breaking of solitary waves for both 2D, axisymmetry 2D and 3D problems, such as Gong et al (2007). For water entry problems, the air cavity enclosed by the water may significantly affect the local free surface profile and flow field, and then the hydrodynamics loads. Correctly simulating the multiphase flows may improve the force prediction.

## 1 Mathematical Modeling

The momentum and kinematics equations for fluid particles are

$$\begin{aligned}\frac{\mathrm{d}\vec{v}_a}{\mathrm{d}t} &= \vec{g} - \frac{\nabla p_a}{\rho_a} + \nu_0 \nabla^2 \vec{v}_a \\ &= \vec{g} - \sum_b \frac{m_b}{\rho_a \rho_b}(p_a + p_b)\nabla_a W_{ab} + \sum_b m_b \frac{2\nu_0}{\rho_a + \rho_b} \frac{(\vec{v}_a - \vec{v}_b)(\vec{r}_a - \vec{r}_b)\cdot\nabla_a W_{ab}}{\left|\vec{r}_a - \vec{r}_b\right|^2}\end{aligned} \tag{1}$$

$$\frac{\mathrm{d}\vec{x}_a}{\mathrm{d}t} = \hat{\vec{v}}_a = \vec{v}_a - \varepsilon \sum_b m_b \frac{\vec{v}_{ab}}{\bar{\rho}_{ab}} W_{ab} \tag{2}$$

For weak compressible method, the pressure is calculated by state equation that used by Monaghan (1994) and based on Batchelor (1974) $P = P_0\left[\left(\rho/\rho_0\right)^{\gamma} - 1\right]$, where $\gamma = 7$ and $\rho_0 = 1000$ for water. The parameter $P_0$ is chosen to have maximum density oscillations of order of $O(1\%)$ around the reference density $\rho_0$. In practice, the constant $P_0$ refers to the sound speed $c_0^2 = \mathrm{d}P/\mathrm{d}\rho$ which is ten times or larger than the highest fluid velocity expected in the physical problem, that is $P_0 = \rho_0 c_0^2/\gamma$. For weak compressible flows, the density is calculated by integrating:

$$\frac{\mathrm{d}\rho_a}{\mathrm{d}t} = -\rho_a \nabla\cdot\vec{v}_a = \sum_b m_b (\vec{v}_a - \vec{v}_b)\cdot\nabla_a W_{ab} \tag{3}$$

## 2 Code verifications

To demonstrate the incompressible SPH code is correctly programmed, verification is conducted to verify the weak compressible SPH code and various boundary implementations. A symmetric wedge vertical impacting the free surface is simulated. To save the computational cost, a non-reflection boundary is designed using the damping function to reduce the reflection of sound wave. The boundary pressure is obtained using an improved coupling boundary treatment approach. Two snapshots of pressure distribution during water entry are illustrated in Fig. 1. Computational results were compared with the experimental and analytical results given by Zhao & Faltinsen (1993), as shown in Fig.2. The pressure distribution fits well the analytical results.

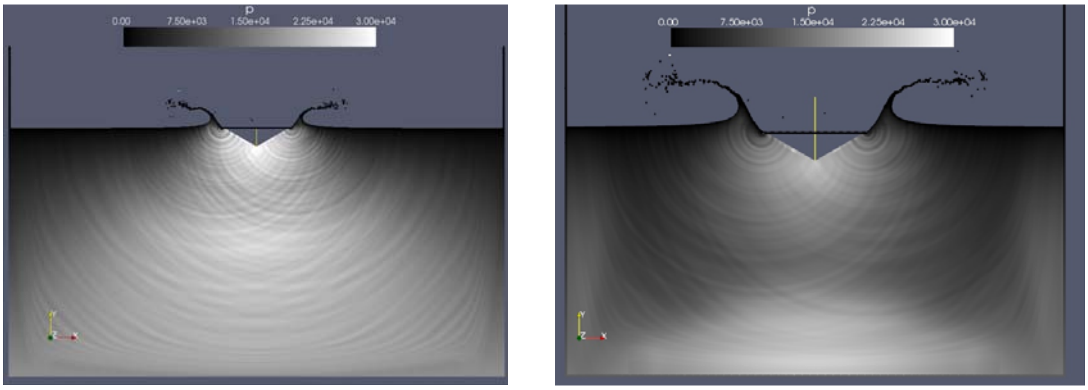


**Figure 1. Pressure distribution of the whole flow field for smaller computational domain at** $t = 0.0248\mathrm{s}$ **and** $t = 0.031\mathrm{s}$

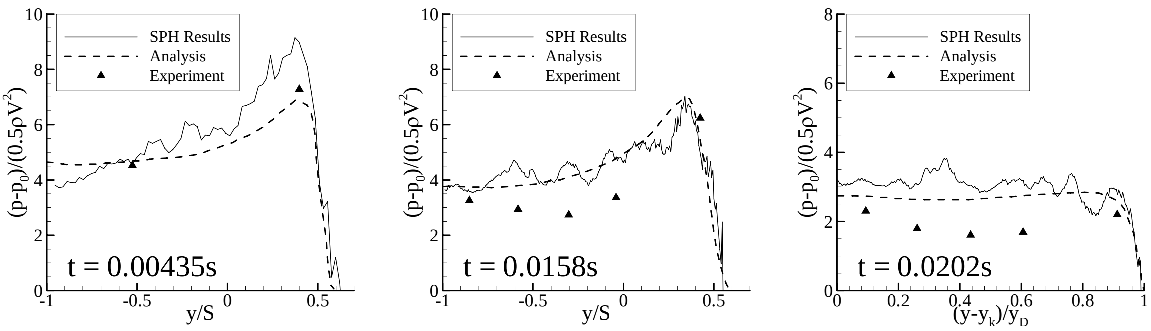


**Figure 2. Comparison of the pressure distribution on the wedge.**

## 3 Parallel weak compressible flow solver

The parallel environment is a DELL T5400 workstation, with two quad-core XEON E5420 2.5GHz processors and 16G full-buffered RAM. The most time consuming part is neighbor searching although a link-list method has been used. The particle pair information is saved in different array for each CPU. Other parts, such as force and density calculation, XSPH could be easily parallelized with OpenMP. The number to total fluid particles is around 4.5 million for the water entry problem shown in previous subsection. The total pair's number is around 100 million. The parallel efficiency is 0.75n as shown in Fig.3. Good computational performance is obtained when four threads are created in the same processor. The speedup decays apparently when additional threads are spawned in the second processor due to additional system time costs. Computational experiences suggest that the parallel strategy at the level of fluid particles is suitable for multi-core processors.

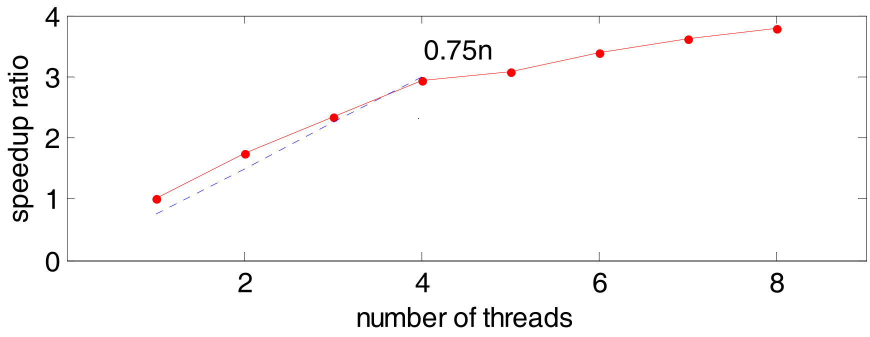


**Figure 3. Speedup of the parallelized weak compressible SPH code.**

## 4 Enclosing of water entry cavity

In some circumstances, the single phase model could not predict the physical process after enclosing, wherein the relative pressure is zero even in the enclosed cavity. However, with two phase SPH model, the flows of entrapped air could be well simulated after enclosing of the cavity. Comparison between numerical results and physical experiments found a good agreement, as the particles distribution shown in Fig.4. Only water particles are shown in the figure.

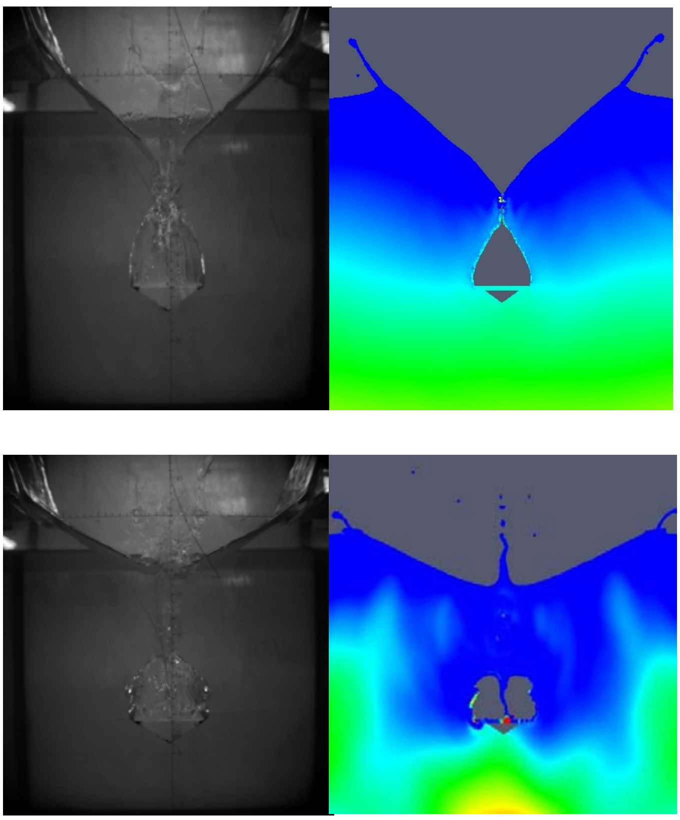

**Figure 4 Comparison between experimental results and computed results of water phase.**

To save computational efforts, absorbing boundary is implemented to remove the sound disturbance from the computational domain, details in Gong et al (2009). By this approach, the computational time can be extended without the limitation of sound wave's reflection from solid boundaries, which is a common drawback of weak compressible SPH method.

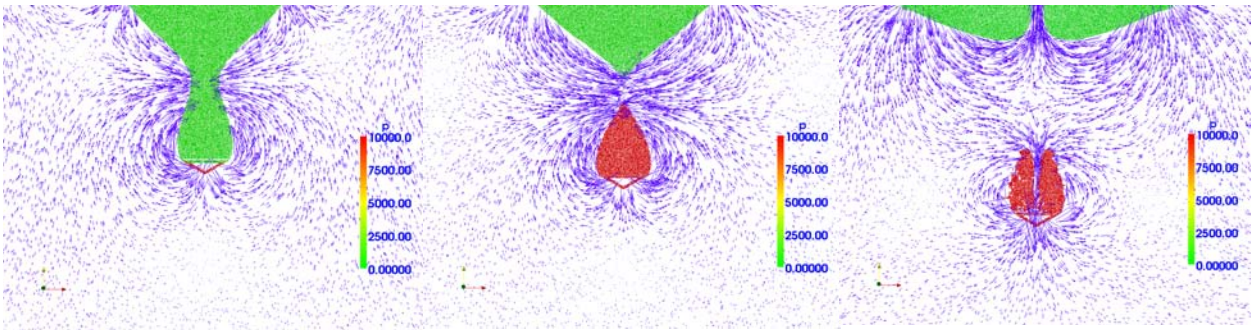


**Figure 5 Pressure (in colour) and velocity vector distribution before and after cavity closure.**

The flow field and pressure are shown in Fig.5, in which the pressure of vented cavity equals the atmosphere pressure in terms of relative value. After enclosing, the pressure in the sealed cavity increases rapidly and. the re-entrant jet is formed after the deep enclosure.

## 5 Concluding remarks

To simulate the enclosing process after water entry, a two phase SPH method is applied. With high spatial resolution of fine particle distribution, details of water entry, including deformation of the free surface, pressure distribution and total force etc, could be well predicted. The enclosing of the water entry cavity running with a wedge is successfully simulated with the proposed SPH model, providing a powerful Lagrangian approach for numerical simulation of the violent free surface flow.

### Bibliography

[1] Batchelor G.K. (1974). *Introduction to Fluid Dynamics*. Cam. Univ. Press, Cambridge, U.K..

[2] Monaghan J.J. and Kocharyan A. (1995). SPH simulation of multi-phase flow. *Computer Physics Communication*, 87: 225-235.

[3] Oger G.., Doring M., Alessandrini, B. and Ferrant, P. (2006). Two-dimensional SPH simulations of wedge water entry, *J. Comp. Phys.* 213: 803-822.

[4] Gong K., Liu H. (2007). Numerical simulation of circular disk entering water by an axisymmetrical SPH model in cylindrical coordinates, *Proceeding of the Fifth International Conference on Fluid Mechanics*, Aug., Shanghai, China.

[5] Gong K., Liu H., Wang B.L. (2009). Water entry of a wedge based on SPH model with an improved boundary treatment, *Journal of Hydrodynamics*, 21(6):750-757.

[6] Zhao R., Faltinsen O. (1993). Water entry of two-dimensional bodies. *J. Fluid Mech.*, 246: 593-612.