

Numerical simulation of fluid-structure interaction using a Level-Set Immersed Boundary Method

Author Names: Wei Bai, Chengzhong Huo

Department of Civil and Environmental Engineering, National University of Singapore, Singapore, (w.bai@nus.edu.sg, ceehc@nus.edu.sg)

1 Introduction

Investigation of the interaction between free surface flows and marine structures is a classical hydrodynamic problem and has a wide range of applications in many offshore engineering problems. It is difficult to study numerically complex free-surface evolutions and irregular boundaries. The challenge is even harsher if the structure is in motion. For example, a flow singularity occurs when a body impacts the free surface which gives rise to a high pressure peak localized at the spray root and makes water entry and exit problems difficult. In recent years, the Level-Set Method (LSM) has become a popular tool for the modeling of two-phase fluid flows due to its simple representation of the surface curvature and the ease of its implementation, See Osher and Fedkiw (2001) for a review. Due to its standard form does not guarantee the overall conservation of fluid, especially in regions of high curvature, the Hybrid Partial Level Set Method (HPLS) of Enright et al. (2002) has been proposed to improve its conservation properties and accuracy. On the other hand, the Immersed Boundary Method (IBM) is popularly used to mimic the boundary immersed in the fluid flow. In the IBM, a body force is introduced to the momentum equations to enforce the boundary condition of the structure in the fluid (Fadlun et al. 2000). The IBM has the advantage of simplifying the grid generation and its inherent simplicity to study moving body (Mittal et al. 2005) on fixed Cartesian grids. Furthermore, it is very convenient to compute forces acting on a body namely lift and drag force, because of its appropriate treatment in the IBM. These advantages suggest that it is well suited to study problems involving a moving body with the free surface flow.

Here, we investigate the applicability of the particle Level-Set Immersed Boundary Method (LS-IBM) for the simulation of breaking waves with obstacle

and water entry problem. The incorporation of an immersed boundary method with a free surface capture scheme implemented in a Navier-Stokes solver allows the interaction between fluid flow with free surface and moving bodies of almost arbitrary shape to be modeled. Dam break past a rectangular obstacle is modeled using the present model and the simulation results using LS-IBM agree well with the experimental results in the literature. The LS-IBM is also applied to study breaking waves with obstacle of different shape for different purpose. In this paper only the dam break past a circle obstacle is shown for the purpose of demonstration.

2 Mathematical Model

2.1 Governing equations

In the study of 2D wave-structure interactions, the incompressible viscous fluid flow is governed by the Navier-Stokes equations:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{\rho(\phi)} \left(-\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \right) + f_i \quad (2.1)$$

and the continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2.2)$$

where Cartesian tensor notation is used, ($i = 1, 2$), u_j , p and x_j are the velocities, pressure and spatial coordinates respectively, f_i represents momentum forcing components, ρ is the fluid density and τ_{ij} are the viscous stress components given by

$$\tau_{ij} = \mu(\phi) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (2.3)$$

where μ is the fluid viscosity.

2.2 Free surface equations

We investigate the motion of two incompressible fluids and track the movement of the free surface implicitly by the Level Set Method (LSM). In LSM a scalar quantity ϕ , known as the level set function, is specified throughout the domain to represent the location of grid cells relative to the surface. Here, we define ϕ to be a signed distance function, which measures the shortest distance from the grid cell to the free surface (*i.e.* $|\nabla\phi| = 1$) and is positive in one fluid phase and negative in the other. The evolution of the level set function ϕ is governed by

$$\frac{\partial\phi}{\partial t} + u_i \frac{\partial\phi}{\partial x_i} = 0 \quad (2.4)$$

In the governing equations, both ρ and μ vary dependant on the local fluid phase properties and are smoothed over a small distance $\varepsilon = 2\Delta x$ across the surface by use of a Heaviside function H to avoid numerical instabilities caused by the sharp gradients present. We calculate ρ and μ by

$$\begin{aligned} \rho(\phi) &= \rho_{air} + H(\phi)(\rho_{water} - \rho_{air}) \\ \mu(\phi) &= \mu_{air} + H(\phi)(\mu_{water} - \mu_{air}) \end{aligned} \quad (2.5)$$

where,

$$H(\phi) = \begin{cases} 0 & \phi < -\varepsilon \\ \frac{1}{2} + \frac{\phi}{2\varepsilon} + \frac{1}{2\pi} \left(\frac{\pi\phi}{\varepsilon} \right) & -\varepsilon \leq \phi \leq \varepsilon \\ 1 & \phi > \varepsilon \end{cases} \quad (2.6)$$

At present, we neglect the influence of surface tension since we are currently interested in the large scale dynamics of gravity waves.

3 Numerical Approach

3.1 Navier-Stokes solver

The Navier-Stokes equations are discretized on a staggered grid with p , ρ and μ defined at grid cell centers and the velocity components at cell faces. We discretize the temporal gradient with a second order Runge Kutta Total Variation Diminishing (RK-TVD) scheme. Eqs. (2.1) and (2.2) are discretized using a fractional step method.

$$u_i^{**} = u_i^n + \Delta t \left(\frac{1}{\rho^n(\phi)} \left(\frac{\partial \tau_{ij}^n}{\partial x_j} \right) - u_j^n \frac{\partial u_i^n}{\partial x_j} + g_i + f_i \right) \quad (3.1)$$

$$\frac{\partial}{\partial x_i} \left(\frac{1}{\rho^n(\phi)} \frac{\partial p^n}{\partial x_i} \right) = \frac{1}{\Delta t} \left(\frac{\partial u_i^{**}}{\partial x_i} \right) \quad (3.2)$$

$$u_i^{n+1} = u_i^{**} - \frac{\Delta t}{\rho^n(\phi)} \frac{\partial p^n}{\partial x_i} \quad (3.3)$$

where u^{**} is the predicted velocity, f_i is the momentum forcing used to enforce the desired boundary conditions on an immersed boundary interface and Δt is the time step. In discretizing the convective term in (Eq. 2.1) it is essential to avoid the introduction of numerical instabilities due to the sharp density gradient at the surface. We employ first order upwinding to ensure stability. The diffusive term in (Eq. 2.1) is discretized with a second order central difference. The time step is restricted by the CFL condition and gravity as discussed in Kang et al. (2000). The CFL number for our simulations is kept below 0.5.

3.2 Free surface solver

Accurate solution of the level set equation (Eq. 2.4) is crucial to capture the correct surface physics. Here, we discretize ϕ at cell centers and calculate velocity gradients with a fifth order HJ-WENO scheme (Jiang and Peng, 2000). Temporal gradients are resolved with a third order RK-TVD scheme. Since only the location of the surface is of interest, ϕ can be solved in a narrow band close to the interface. For the HJ-WENO scheme the narrow band occupies six cell widths either side of the surface in all directions (Peng *et al.* 1999). As ϕ is evolved in time it may deviate away from being a signed distance function (*i.e.* $|\nabla\phi| \neq 1$) requiring reinitialisation. Here, we reinitialize ϕ with an efficient fast marching technique at every time step, see Sethian (1996) for details.

3.3 Immersed boundary treatment

It is noted that the term f_i must be determined prior to the computation of the predicted velocities u^{**} . This term is prescribed at each time step to establish the desired boundary moving velocity V_{ib} . For a time-stepping scheme, this force can be expressed as

$$\frac{u_i^{n+1} - u_i^n}{\Delta t} = RSH_i + f_i \quad (3.4)$$

where RSH_i includes the convective, viscous, pressure gradient and body force of the governing equations. If the forcing f_i must yield $u_i^{n+1} = V_{ib}^{n+1}$ on

the immersed boundary where V_{ib}^{n+1} is the Dirichlet boundary condition at the immersed boundary, the forcing is given from the equation below,

$$f_i = \frac{V_{ib} - u_i^n}{\Delta t} - RSH_i \quad (3.5)$$

This forcing is direct in the sense that the desired boundary condition can be satisfied at every time step but only holds when the immersed boundary coincides with the grid. In general, Eulerian grid nodes almost never coincide with the immersed boundary in practical applications. f_i needs to be computed at grid points near and not exactly on the interface. The value of u_f at the forcing points is not known and has to be reconstructed using the information from the interface and surrounding field. In terms of u_f that has been predicted by the reconstruction, the force on the forcing points can be expressed as

$$f_i = \frac{u_f - u_i^n}{\Delta t} - RSH_i \quad (3.6)$$

4 Results

4.1 Dam break past a rectangular obstacle

Fluid flow past a fixed rectangular obstacle is simulated to verify that the present LS-IBM accurately predicts flow phenomena such as separation and wave breaking. In order to compare our results directly with the experiments of Koshizuka et al. (1995) and numerical simulations of Larese et al. (2008) and Gao Mimi (2011), we consider a rectangular column of water of initial height $2L$ and width L within a computational domain of $4L \times 4L$ (height \times length) where $L = 0.072\text{m}$. An obstacle block with geometry $h \times 2h$, where $h = 0.024\text{m}$ is used in the experiment and numerical simulation, is located at the middle of the tank. A schematic view of the problem is shown in Figure 1. In this case, we set the immersed boundary coincide with the grid line. So the momentum forcing term is specified in a way to make the velocity magnitude to be equal to the immersed boundary velocity at these points. Base on the no-slip condition on immersed boundary, the velocity on these points should be zero. And then we can obtain the forcing term which satisfy the desired boundary conditions on the immersed boundary.

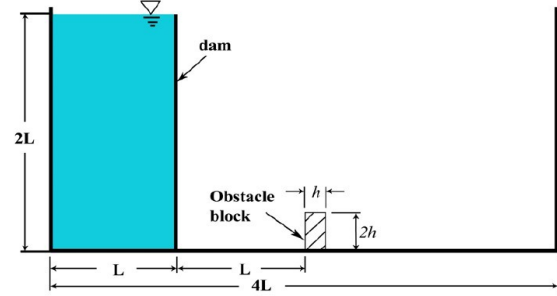


Figure 1 Sketch of dam break with a rectangular obstacle

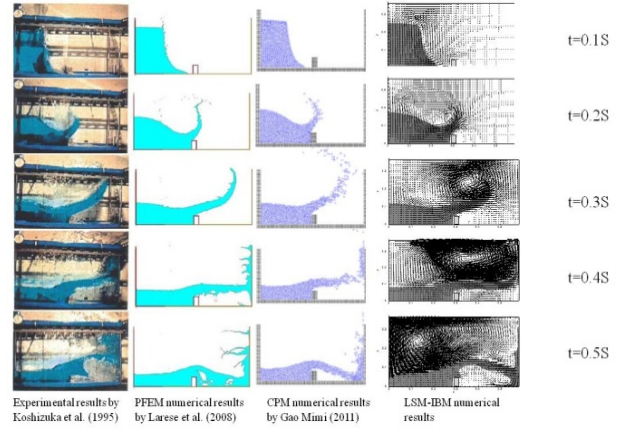


Figure 2 comparison of snapshot for the dam break with a rectangular obstacle

Figure 2 shows the water profiles at several time instants after dam breaks. The numerical results using LS-IBM agree well with the experimental results and look better than the other two numerical simulations.

4.2 Dam break past a circular obstacle

To further demonstrate the capability of LS-IBM in simulation of arbitrary bodies, another dam break example is presented in this section. We choose a circular obstacle that consists of many line segments of different slopes to verify the capability of the interpolation method. The computational domain is $4L \times 4L$ (height \times length) and dam dimension is $2L \times L$ (height \times width) where $L = 0.6\text{m}$. The radius of the circle is 0.19m and the position of the center is 0.6m away from the right side boundary and 0.26m above the bottom. A schematic view of the problem is shown in Figure 3.

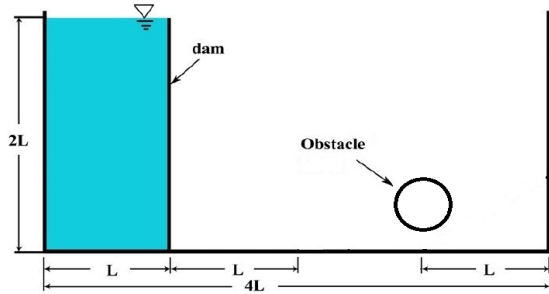


Figure 3 Sketch of dam break with a circular obstacle

In this case, the immersed boundary is not aligned with the grid plane which makes the problem more complicated. The momentum forcing term will act only on the points nearest to the immersed boundary. In order to render a velocity that is approximately equal to immersed boundary velocity for the forcing term, an interpolation for the momentum forcing is therefore required.

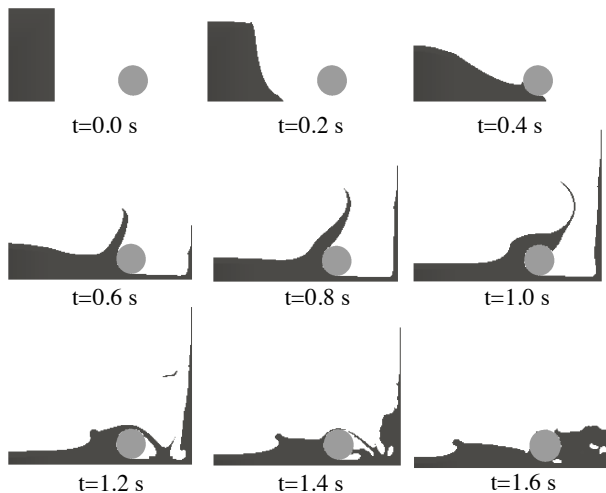


Figure 4 Water profiles at several time instants for dam breaking with a circle obstacle

Snapshots of this case are shown in Figure 4. Good performance can be seen. The results indicate that the interpolation method is correct and it can be implemented successfully for the fluid-structure interaction problem.

5 Conclusions

The Level-Set Immersed Boundary Method (LS-IBM) technique has been successfully implemented to simulate dam breaking with complex-shape obstacles. The results perform well and are in good

agreement with the available experimental results, which show that the IBM is very robust for complicated geometries. Now, water entry of a moving body is being investigated, and some preliminary results have been obtained.

6 References

- Enright, D, Fedkiw, R, Ferziger, J, Mitchell, I (2002). "A Hybrid Particle Level Set Method for Improve Interface Capturing," *J. Comput. Phys.* Vol 183, pp 83-116.
- Fadlun E.A., Verzicco R., Orlandi P., Mohd-Yusof J., 2000. Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations. *J. Comput. Phys.* 61, 35-60.
- Jiang, G-S, and Peng, D (2000). "Weighted ENO schemes for Hamilton-Jacobi Equations," *SIAM J. Sci. Comput.* Vol 21, No 6, pp 2216-2143.
- Kang, M, Fedkiw, R.P. and Liu, X-D (2000). "A boundary condition capturing method for multiphase incompressible flow," *J. Sci. Comput.* Vol 15, pp 323-360.
- Koshizuka, S., Tamako, H. and Oka, Y. (1995). A particle method for incompressible viscous flow with fluid fragmentation, *Computational Fluid Dynamic Journal*, 4(1), 29-46.
- Larese, A., Rossi, R., Onate, E. and Idelsohn, S.R. (2008). Validation of the particle finite element method (PFEM) for simulation of free surface flows, *Engineering Computations*, 25, 385-425.
- Mittal R. and Iaccarino G., 2005. Immersed boundary methods. *Annual Review of Fluid Mechanics* 37, 239-261.
- Osher, S, and Fedkiw, R.P (2001). "Level set methods: An overview and some recent results," *J. Comput. Phys.* Vol 169, pp 463-502.
- Peng, D, Merriman, B, Osher S, Zhao, H, Kang, M (1999). "A PDE-Based Fast Local Level Set Method," *J. Comput. Phys.* Vol 155, pp 410-438.
- Sethian, J.A. (1996). A fast marching level set method for monotonically advancing fronts. *Proceedings of the National Academy of Science USA.* **93**: 1591-1595.