



Citation for published version:

Chen, Q, Zang, J, Kelly, D, Williams, C & Dimakopoulos, A 2015, 'Particle-In-Cell Numerical Solver for Free Surface Flows with Fluid-Solid Interactions' 30th International Workshop on Water Waves and Floating Bodies, Bristol, UK United Kingdom, 12/04/15 - 15/04/15, .

Publication date:
2015

Document Version
Publisher's PDF, also known as Version of record

[Link to publication](#)

University of Bath

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

Particle–In–Cell Numerical Solver for Free Surface Flows with Fluid–Solid Interactions

by Q. Chen^{1,*}, J. Zang¹, D. M. Kelly^{2,3}, C. J. K. Williams¹ and A. Dimakopoulos²

¹Department of Architecture and Civil Engineering, University of Bath, BA2 7AY, U.K.

²H R Wallingford, Wallingford, Oxon, OX10 8BA, U.K.

³International Hurricane Research Center, Florida International University, Miami FL 33199, USA

Email: chenqiang913@hotmail.com

Highlights:

- This paper presents a novel numerical approach based on the Particle–In–Cell (PIC) technique for the solution of the incompressible Navier–Stokes equations with emphasis on free surface deformation and two–way fluid–solid interactions. As a hybrid Eulerian–Lagrangian approach, this method has the flexibility of the Smoothed Particle Hydrodynamic (SPH) method as well as the efficiency of an Eulerian method.
- Two–way fluid–solid interaction simulation has been integrated inside the numerical model by adopting the Distributed Lagrangian Multiplier (DLM) technique proposed in Patankar [7].

1 Introduction

In the past few decades Computational Fluid Dynamics (CFD) techniques have been widely used for both academic research and commercial engineering applications. CFD techniques have become more and more popular as computational power has continued to increase. For the solution of the Navier–Stokes equations three principal approaches are typically employed these being: Eulerian methods, Lagrangian methods and hybrid Eulerian–Lagrangian methods. While grid based Eulerian methods perform well in terms of equation discretization, enforcing incompressibility and improving computational efficiency[3], they have drawbacks with regards to integration of the advection term and require more effort in handling the free surface boundary especially when it is subject to extensive deformation. From this point of view, purely Lagrangian techniques such as the SPH method and Moving Particle Semi Implicit (MPS) schemes seem to be more suitable for free surface fluid problems as they can handle large free surface deformation easily (e.g.[1]). In addition Lagrangian methods can integrate the advection term relatively trivially through advecting the discretized fluid elements. However, pure Lagrangian methods tend to be extremely demanding in terms of CPU time as millions of particles may be used for high accuracy (e.g.[1]).

Our work is motivated by the idea of developing a hybrid Eulerian–Lagrangian approach based on the PIC framework which exhibits both the flexibility of the SPH method in terms of ability to simulate complex problems and the computational efficiency of Eulerian methods. The PIC method was originally devised for compressible flows by Francis Harlow [5] in 1955. The idea being that particles, which carry and advect fluid mass and momentum, are seeded on an underlying mesh on which the main process of solving the Navier Stokes equations is undertaken. Information between particles and the grid is transferred via interpolations. The *classic* PIC method (e.g. [5]) suffered from high numerical dissipation due to the direct velocity transfer at each time step. Brackbill and Ruppel [2] suggested incrementing the particle velocity by the change of velocity on the grid, which reduced the dissipation significantly. In a previous work the authors applied the PIC technique to various complicated 2D flow problems including full two–way fluid–solid interactions can be found in [6]. In this paper we will present a 2D validation case and an example case from the 3D version of the PICIN numerical model which is currently under development.

2 Numerical model

The numerical model employs the framework of PIC methodology and solves the incompressible Navier Stokes equations for a Newtonian fluid in both 2 and 3 spatial dimensions. Two–way fluid–solid

interaction model is integrated inside this model using the approach proposed in [7], which forms part of the overall governing equations :

$$\nabla \cdot \vec{u} = 0 \quad \text{in } \Omega, \quad (1)$$

$$\mathbf{D}[\vec{u}] = 0 \quad \text{in } \mathbb{S}, \quad (2)$$

$$\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} = \vec{f} - \frac{1}{\rho} \nabla p + \nu \nabla^2 \vec{u} \quad \text{in } \mathbb{F}, \quad (3)$$

$$\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} = \vec{f} - \frac{1}{\rho_S} \nabla p + \nabla \cdot \mathbf{\Pi} \quad \text{in } \mathbb{S}, \quad (4)$$

with boundary conditions:

$$\vec{u} = \vec{u}_\Gamma(t) \quad \text{on } \Gamma(t) \quad (5)$$

and:

$$\vec{u} = \vec{u}_i \quad \text{and} \quad (\mathbf{\Pi} - p\mathbf{I}) \cdot \vec{n} = \vec{T} \quad \text{on } \partial\mathbb{S}(t), \quad (6)$$

where Ω , \mathbb{F} and \mathbb{S} denote the overall computational domain, fluid region and solid region. Γ and $\partial\mathbb{S}$ represent the overall solid boundary and solid region boundary. \vec{u} is the velocity field, p represents the pressure, \vec{f} is the gravity force, ν accounts for the fluid dynamic viscosity coefficient, and ρ and ρ_S are the fluid density and solid density, respectively. $\vec{u}_\Gamma(t)$ and \vec{u}_i represents boundary velocity and \vec{T} is the traction force of the fluid on the solid. As an incompressibility constraint, equation (1) ensures a divergence free velocity field in the whole domain and gives rise to pressure field as a Lagrange Multiplier. Similarly, Equation (2), which represents a rigid body constraint, enforces a deformation free velocity field. Here $\mathbf{\Pi}$ is the extra deformation stress in addition to pressure and it is nothing but distributed Lagrange Multiplier due to rigid body constraint [7]. It is noted that for any vector \vec{u} , $\mathbf{D}[\vec{u}] = [\nabla \vec{u} + (\nabla \vec{u})^T] / 2$, which measures the spatial deformation of \vec{u} .

The overall solution has been divided into two major steps, i.e. Eulerian step and Lagrangian step. In the Eulerian step, the pressure projection technique of Chorin [4] has been adopted to solve for fluid motion on the MAC grid. A second-order accurate technique has been employed here for the Dirichlet-type free surface boundary condition and the non grid-aligned solid boundary is resolved via a cut-cell approach. In the Lagrangian step the particles, which are initially seeded inside the fluid cells, are advected. They carry with them the updated divergence free velocity field interpolated from the grid. Thus, the advection term of Navier-Stokes equation is integrated with respect to time. We note here that the velocity field will be transferred back to the grid for the next time step computation after the particles are advected. The two-way fluid solid interaction is treated in a manner such that the solid objects are firstly solved as if they were fluid and then a velocity correction inside the solid region is made by taking account of the density difference between solid and fluid. The rigid body restraint is finally enforced by finding a unique solid velocity considering all the momentum contributions from solid region. This technique is straightforward to implement and can treat floating bodies very efficiently. More details of the numerical model can be found in Kelly et al. [6].

3 Case study

3.1 Case 1 : Movement of caisson breakwater

A 2D case of caisson breakwater movement was used for the validation of our 2D numerical model. This case was previously investigated experimentally in Wang et al [9] and numerically in Rogers et al [8] by SPH method. Figure 1 depicts the simulation sketch of a wave paddle generating waves that travel to impact the caisson breakwater which was placed on a fixed foundation and allowed to move horizontally . The generated wave has a period of 1.3s and a height of 0.15m. The caisson was given a density of 1440 kg/m^3 and the numerical fiction force between caisson and the foundation was adopted following the method proposed in [8], where the fiction force was switched from static force and dynamic force based on the relationship between caisson velocity and a threshold velocity which we found had a very sensitive effect on the motion of caisson. Here, following [8], the caisson foundation is considered to be impermeable. The simulation presented here used a cell size of $\Delta x = \Delta z = 0.013m$ with a total number of about 64,000 particles. The CFL number was set at 0.5 to adapt the time step and it took about 1.4hrs for 15 seconds of simulation on an Intel(R) i5-3470 CPU@3.2GHz core.

Figure 2 shows snapshots of our simulation at time instants roughly similar to those presented in [8]. A comparison of results shows that the wave behavior is similar to that observed in [8]. Figure 3 presents comparisons of the caisson displacement and overall horizontal wave force between experimental data and numerical results. It is noted here that the experimental data has some limitations for a comprehensive comparison as reported in [8]. We started to measure the displacement data around the time instant when the caisson was just activated to move backward after a slight forward motion due to seaward water level decline because of the first wave propagation and the wave force was compared from our first relatively stable wave force. It can be seen from Figure 3(a) that the numerical fiction force sensitively influences the motion of caisson though the displacement magnitude is captured by the numerical model. Also, in Figure 3(b), the agreement of impact peak wave force and lowest wave force are acceptable though slight phase difference occurs. Overall, the numerical model quantitatively captures the principal characteristics of caisson movement due to wave actions.

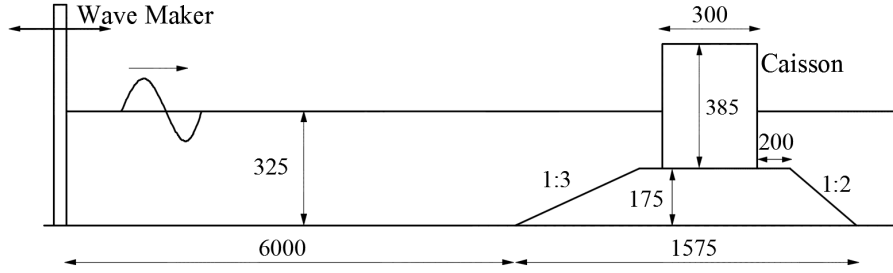


Figure 1: Numerical model set-up for caisson breakwater (Units: mm).

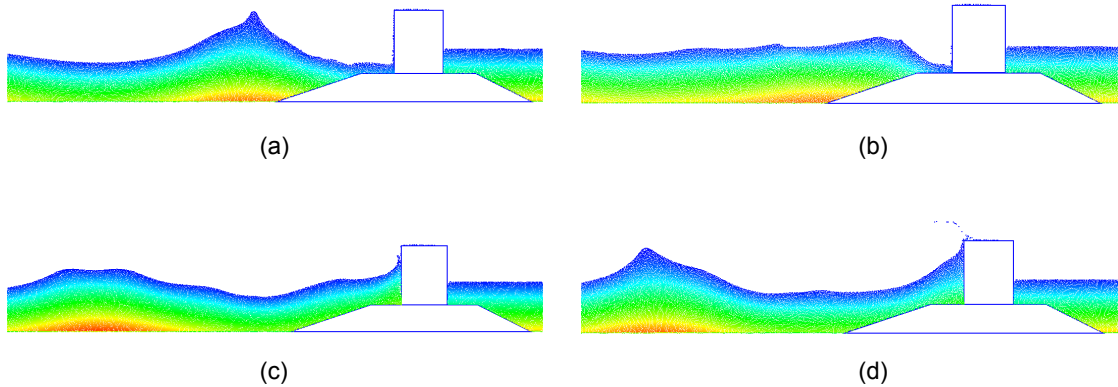


Figure 2: Snapshots of numerical simulation for caisson movement. The colour represents pressure field ranging from min value 0 kPa(blue) to max value 4 kPa (red).

3.2 Case 2 : 3D solid impacting water surface

A falling missile-shaped object impacting water surface case was used to test our 3D version PIC solver. Figure 4 shows snapshots of the numerical simulation – an object falls into water within a tank and causes water surface evolution due to the impact. This is an on-going study using our recently developed 3D version of the PIC solver, we will present more results in the workshop.

Acknowledgments

The authors acknowledge with thanks the financial support of the University of Bath and HR Wallingford.

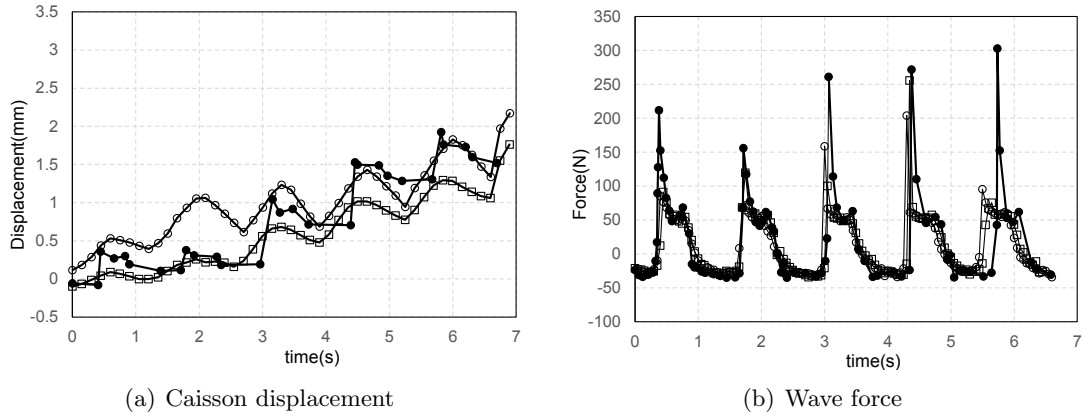


Figure 3: Comparisons between experimental data and numerical results of moving caisson under wave actions. Line with “●”: Experimental data; Line with “□”: Numerical results, threshold velocity set at 1mm/s; Line with “○”: Numerical results, threshold velocity set at 2mm/s .

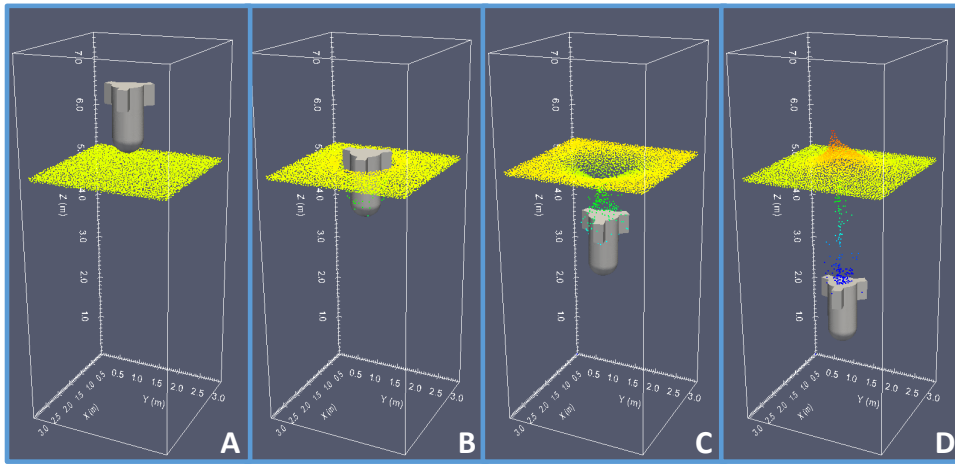


Figure 4: Snapshots of numerical simulation for solid impacting free surface. The colour represents the height of free surface particle position; the warmer colour means the higher position.

References

- [1] B. Bouscasse, A. Colagrossi, S. Marrone, and A. Souto-Iglesias. Viscous flow past a circular cylinder below a free surface. In J. M. Smith, editor, *Proceedings of the ASME 2014 33rd Int. Conf. on Ocean, Offshore & Arctic Engineering.*, San Fransisco, USA, 2014. OMAE.
- [2] J. U. Brackbill and H. M. Ruppel. FLIP: A method for adaptively zoned, Particle-In-Cell calculations of fluid flows in two dimensions. *J. Comp. Phys.*, 65:314–343, 1986.
- [3] R. Bridson. *Fluid Simulation for Computer Graphics*. A K Peters, Ltd, 2008.
- [4] A. J. Chorin. Numerical solution of the Navier-Stokes equations. *Mathematics of computation*, 22:745–762, 1968.
- [5] F. H. Harlow. A machine calculation method for hydrodynamic problems. Technical Report LAMS-1956, Los Alamos Scientific Laboratory, Los Alamos, 1955.
- [6] D. M. Kelly, Q. Chen, and J. Zang. PICIN: A particle-in-cell solver for incompressible free surface flows with two-way fluid solid coupling. *Submitted to SIAM Journal*, 2014.
- [7] N. A. Patankar. A formulation for fast computations of rigid particulate flows. *Center for Turbulence Research Annual Research Briefs*, 2001:185–196, 2001.
- [8] B. D. Rogers, R. A. Dalrymple, and P. K. Stansby. Simulation of caisson breakwater movement using 2-D SPH. *Journal of Hydraulic Research*, 48(S1):135–141, 2010.
- [9] Y.-Z. Wang, N.-N. Chen, and L.-H. Chi. Numerical simulation on joint motion process of various modes of caisson breakwater under wave excitation. *Communications in numerical methods in engineering*, 22(6):535–545, 2006.