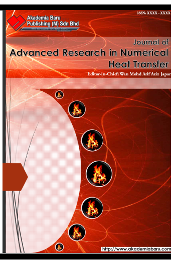
Penerbit
Akademia Baru

Journal of Advanced Research in
Numerical Heat Transfer

Journal homepage:
http://www.akademiabaru.com/submit/index.php/arnht/index
ISSN: 2735-0142



A 3D Mesh-Less Algorithm for Simulating Complex Fluid Structure Interaction (FSI) Problem involving Free Surface

K. C. Ng[1,*]

[1] Department of Mechanical, Materials and Manufacturing Engineering, University of Nottingham Malaysia, Jalan Broga, 43500 Semenyih, Selangor, Malaysia

ABSTRACT

In this preliminary work, we would like to demonstrate some flow applications of our in-house 3D Fluid Structure Interaction (FSI) simulation software that we have developed at University of Nottingham Malaysia (UNM). This mesh-less FSI simulation software is developed based on the Smoothed Particle Hydrodynamics (SPH) and Volume Compensated Particle Method (VCPM) for fluid and solid modelling respectively. A force coupling algorithm has been developed to couple the fluid and solid body solvers. Recently, the algorithm has been implemented in an open-source C++ code (i.e. DualSPHysics) so that complex large-scale 3D problem can be simulated within a reasonable time frame by making use of the Graphical Processing Unit (GPU) technology. A few simulation cases are highlighted in order to show some potential applications of this FSI method.

Keywords:
Smoothed Particle Hydrodynamics (SPH); Fluid Structure Interaction (FSI); DualSPHysics; Lattice Particle Method (LPM); Lattice Spring Models (LSMs); Free Surface

1. Introduction

Mesh-less method is a powerful simulation technique that has been adopted to solve many complex engineering problems such as those related to heat transfer [1-3], mixing [4-6], multiphase flow [7], etc. It has been proven to be able to address some limitations of mesh-based methods (e.g. Finite Volume Method, Finite Element Method) such as convective instability [8-9], numerical diffusion due to convective discretization, difficulty in modelling interfacial flows, sensitivity to mesh quality, etc. Recently, there are on-going efforts being developed to couple SPH with other structural solvers in order to model another class of interesting yet challenging engineering problem which is better known as Fluid-Structure Interaction (FSI) problem [10]. Classical FSI simulation using SPH involves the solution of complex constitutive equation in the solid body [11]. Very recently, we have developed a simple yet effective FSI approach known as the SPH-VCPM method where no complex constitutive equation is required for solid modelling [12-14].

In the current work, we intend to showcase some applications that we have simulated using our SPH-VCPM approach.

---

[*] *Corresponding author.*
E-mail address: KhaiChing.Ng@nottingham.edu.my

2. Method

2.1 Fluid body

The flow governing equations are discretized using the Weakly Compressible Smoothed Particle Hydrodynamics (WCSPH) approach. The discretized mass balance equation of particle *i* can be written as:

$$\langle\frac{d\rho_i}{dt}\rangle = \rho_i \sum_j V_j\left(\mathbf{v}_i - \mathbf{v}_j\right) \cdot \nabla_i W_{ij} \qquad (1)$$

where ρ_i is the fluid density of local particle *i*, V_j is the volume of neighbouring particle *j*, $\nabla_i W_{ij} = \frac{dW_{ij}}{dr}\frac{\mathbf{r}_{ij}}{\|\mathbf{r}_{ij}\|}$ and $\mathbf{r}_{ij} = \mathbf{r}_i - \mathbf{r}_j$ is the displacement vector. The relative velocity vector between particles *i* and *j* is $\mathbf{v}_i - \mathbf{v}_j$. W_{ij} is the kernel function with compact support radius of $r_c = 2h$. The smoothing length *h* is set to be 1.5*Dp*, where *Dp* is the initial particle spacing. For WCSPH, the pressure of a fluid particle *i* can be calculated as:

$$P_i = \frac{\rho^F\left(c^F\right)^2}{\gamma}\left(\left(\frac{\rho_i}{\rho^F}\right)^{\gamma} - 1\right), \qquad (2)$$

with γ is set as 7 and ρ^F is the initial fluid density. c^F is the sound speed, normally taken as at least ten times larger than the maximum anticipated flow speed.

The momentum equation can be discretized as:

$$m_i \frac{d\mathbf{v}_i}{dt} = -\sum_j V_i V_j\left(P_i + P_j\right)\nabla_i W_{ij} + \sum_j m_i m_j \frac{4\nu^F}{\rho_i+\rho_j}\frac{\nabla_i W_{ij}\cdot\mathbf{r}_{ij}}{\left(\left\|\mathbf{r}_{ij}\right\|^2+0.01h^2\right)}\mathbf{v}_{ij} + \sum_j m_i m_j\left(\frac{\bar{\bar{\tau}}_i}{{\rho_i}^2} + \frac{\bar{\bar{\tau}}_j}{{\rho_j}^2}\right)\nabla_i W_{ij} + m_i\mathbf{g} \qquad (3)$$

where $\mathbf{F}_i$ is the net force vector acting on a fluid particle *i*. Here, m_i and P_i are the mass and fluid pressure of particle *i*, respectively. The turbulent stress tensor $\bar{\bar{\tau}}$ is modelled using the Large Eddy Simulation (LES) approach. For the modelling of laminar viscous force, the fluid kinematic viscosity ν^F is prescribed.

2.2 Solid body

The 3D Volume Compensated Particle Method (VCPM) is implemented to model the deformation of linear elastic solid bodies. It uses 1D bond-level force-elongation relationship; hence, the use of complex constitutive equation as needed in continuum-based approach such as FEM is unnecessary. The equation of motion for a solid particle *I* can be written as:

$$m_I \frac{d\mathbf{v}_I}{dt} = \mathbf{F}_{S,I} + \mathbf{F}_{F\to S,I}, \qquad (4)$$

where $\mathbf{F}_{S,I}$ is the spring force (acting between solid particles) and $\mathbf{F}_{F\to S,I}$ is the fluid force acting on solid particle *I* (acting between fluid and solid particles). For the detailed derivation of $\mathbf{F}_{S,I}$ and $\mathbf{F}_{F\to S,I}$, readers can refer to our earlier work [13]. Very recently, an improved multi-resolution scheme has been developed in order to enhance the computational efficiency of the current FSI solver [12].

3. Result

Here, we intend to demonstrate the capability of our current FSI solver in simulating two 3D FSI cases involving free surfaces.

The first case involves an elastic plate clamped at the bottom of a tank undergoing rolling motion, in which experimental data is available [15]. The material properties of the elastic plate are: ρ^S = 1100 kgm^{-3}, E = 6 MPa, and v^S = 0.45. The schematic diagram of this flow case is shown in Figure 1. Figure 2 shows the free surface patterns captured using the 2D and 3D FSI approaches. As seen, the free surface predicted using the 3D approach is less rigorous (coming closer to experimental observation) that that predicted using the 2D approach. This could be due to the flow damping effect as the side walls are taken into account in the 3D model. In general, as compared to the 2D model, the 3D free surface pattern compares quite well with that of the experimental photo at different time frames.

The second case involves a very recent benchmark test case proposed by Yilmaz and co-workers [16] which was designed to investigate the wave impact on an elastic sluice gate. Figure 3 shows the schematic diagram of this flow case, detailing all the key geometric information. As the fluid is released from the reservoir, the flood water hits the sluice gate and pushes the sluice gate to the right. The displacement of the sluice gate is shown in Figure 4 and it is appealing to note that the displacement values compare well with those measured at different time levels. As the gate retraces, a complex hydraulic jump forms behind the gate as shown in Figure 5. The von-Mises stress developed within the elastic gate is captured as well.

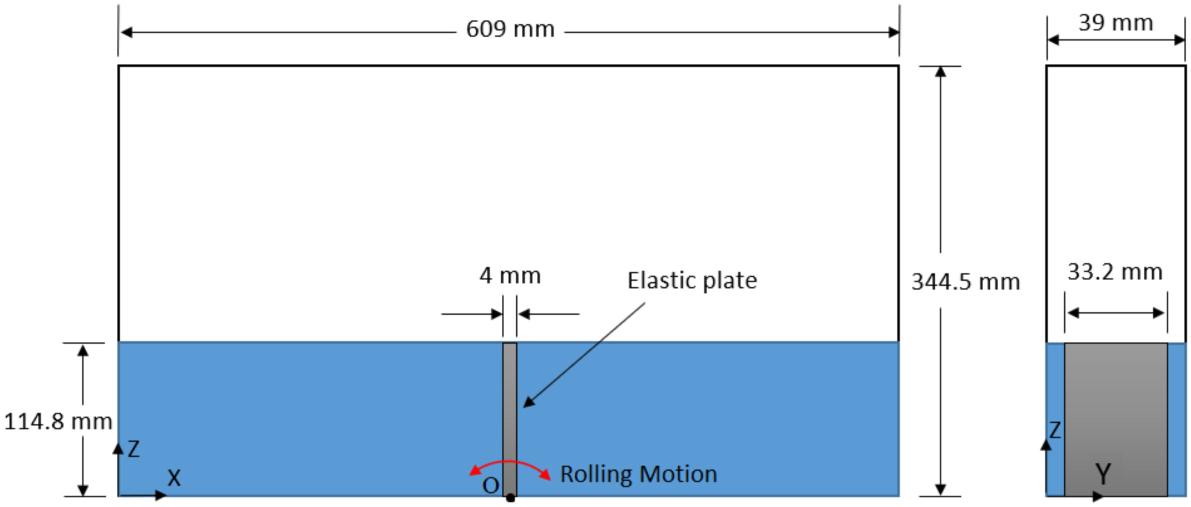


Fig. 1. Geometric description of the 3D rolling tank filled with oil

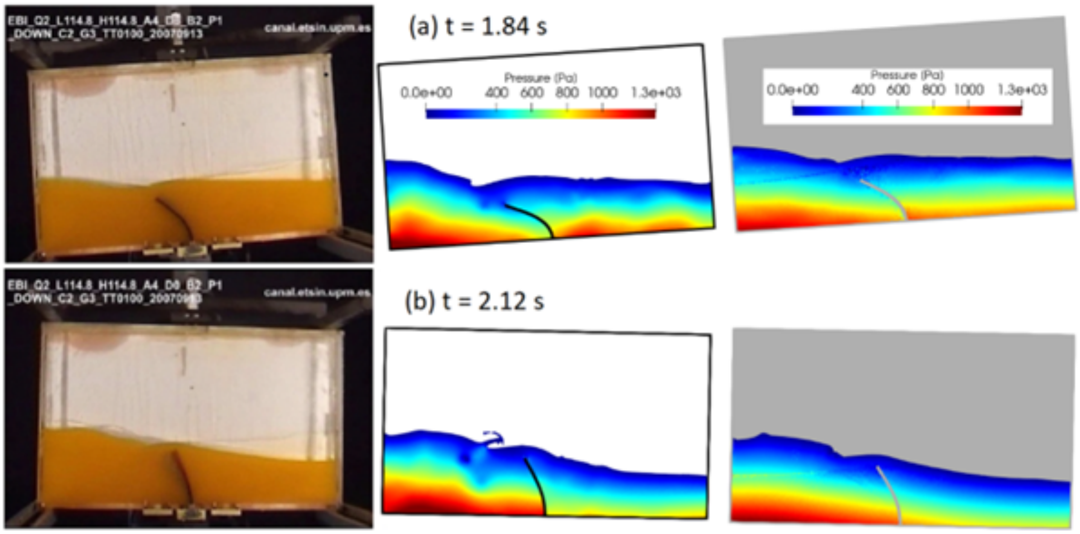


Fig. 2. Free surface patterns at various sloshing stages obtained from experiment [6] (left column), 2D SPH-VCPM method (middle column) and 3D SPH-VCPM method (right column)

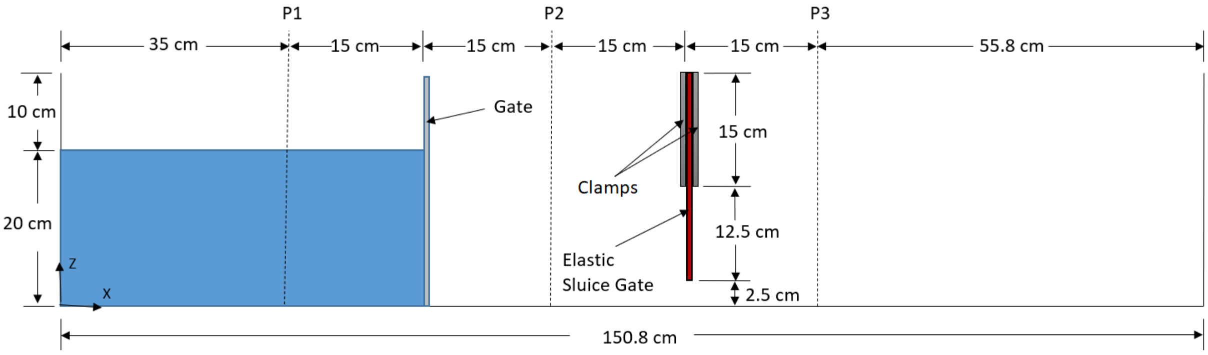


Fig. 3. Schematic diagram of the 3D sluice gate problem

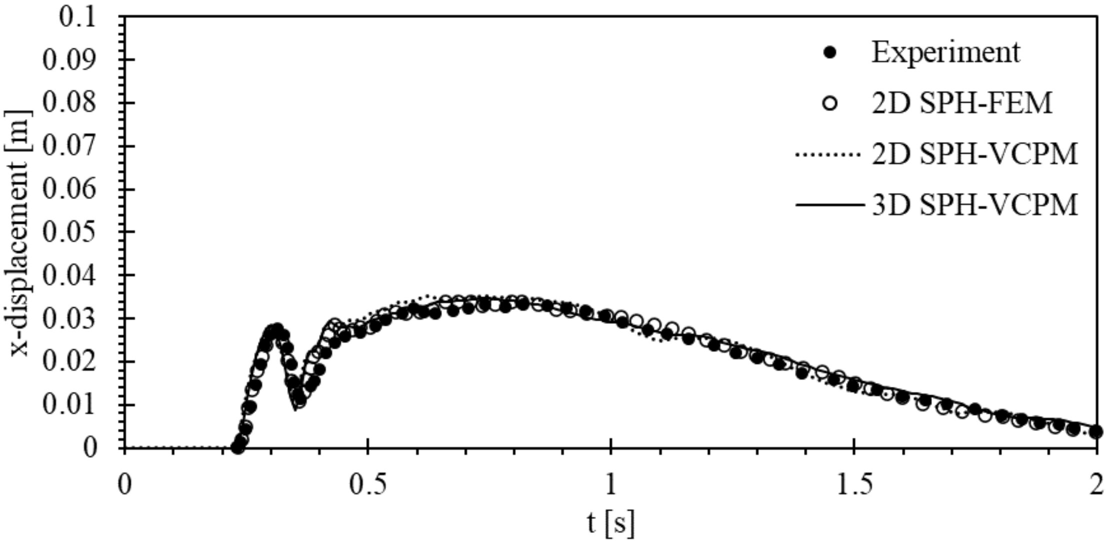


Fig. 4. Displacement of a point on the sluice gate located at 5 cm from the free end

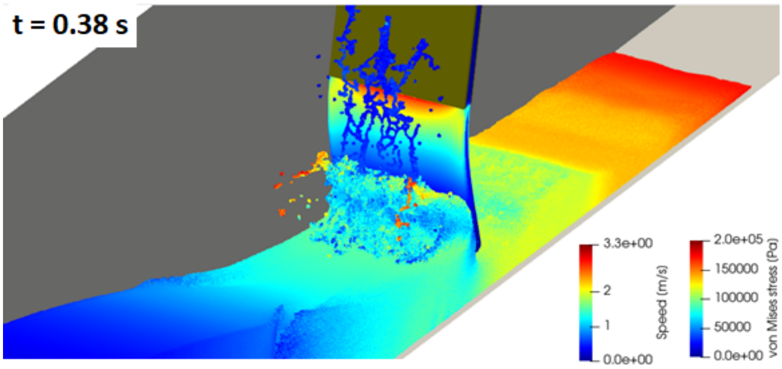


Fig. 5. Splashing of water particles behind the sluice gate

4. Conclusion

The 3D FSI solver has been developed and used to simulate some 3D FSI cases involving free surface and large solid deformation. The numerical results have been found to agree considerably well with the experimental data. In the future, the solver will be further validated and its computational performance will be investigated. As the inherent strength of Volume Compensated Particle Method (VCPM) is in fracture modelling, the immediate research direction would be extending the current 3D FSI solver in handling such cases, by incorporating the bond-breaking rule.

Acknowledgement

The author would like to thank the financial support given by University of Nottingham Malaysia (UNM) and Ministry of Higher Education (MOHE) Malaysia under the Fundamental Research Grant Scheme (FRGS/1/2021/TK0/UNIM/02/6).

Reference

[1] Hosain, Md Lokman, J. M. Domínguez, R. Bel Fdhila, and Konstantinos Kyprianidis. "Smoothed particle hydrodynamics modeling of industrial processes involving heat transfer." *Applied Energy* 252 (2019): 113441.

[2] Ng, K. C., Y. L. Ng, T. W. H. Sheu, and A. Alexiadis. "Assessment of Smoothed Particle Hydrodynamics (SPH) models for predicting wall heat transfer rate at complex boundary." *Engineering Analysis with Boundary Elements* 111 (2020): 195-205.

[3] Ng, Khai Ching, Yee Luon Ng, T. W. H. Sheu, and A. Mukhtar. "Fluid-solid conjugate heat transfer modelling using weakly compressible smoothed particle hydrodynamics." *International Journal of Mechanical Sciences* 151 (2019): 772-784.

[4] Ng, Khai-Ching, Eddie YK Ng, and Wong Hing Lam. "Lagrangian simulation of steady and unsteady laminar mixing by plate impeller in a cylindrical vessel." *Industrial & Engineering Chemistry Research* 52, no. 29 (2013): 10004-10014.

[5] Shamsoddini, Rahim, Mohammad Sefid, and Rouhollah Fatehi. "Incompressible SPH modeling and analysis of non-Newtonian power-law fluids, mixing in a microchannel with an oscillating stirrer." *Journal of Mechanical Science and Technology* 30 (2016): 307-316.

[6] Ng, Khai-Ching, and Eddie YK Ng. "Laminar mixing performances of baffling, shaft eccentricity and unsteady mixing in a cylindrical vessel." *Chemical Engineering Science*104 (2013): 960-974.

[7] Ng, Khai-Ching, Yao-Hsin Hwang, Tony WH Sheu, and Ching-Hao Yu. "Moving Particle Level-Set (MPLS) method for incompressible multiphase flow computation." *Computer Physics Communications* 196 (2015): 317-334.

[8] Ng, Khai-Ching, Tony Wen-Hann Sheu, and Yao-Hsin Hwang. "Unstructured Moving Particle Pressure Mesh (UMPPM) method for incompressible isothermal and non-isothermal flow computation." *Computer Methods in Applied Mechanics and Engineering* 305 (2016): 703-738.

[9] Liu, Kuan-Shuo, Tony Wen-Hann Sheu, Yao-Hsin Hwang, and Khai-Ching Ng. "High-order particle method for solving incompressible Navier–Stokes equations within a mixed Lagrangian–Eulerian framework." *Computer Methods in Applied Mechanics and Engineering* 325 (2017): 77-101.

[10] O'Connor, Joseph, and Benedict D. Rogers. "A fluid–structure interaction model for free-surface flows and flexible structures using smoothed particle hydrodynamics on a GPU." *Journal of Fluids and Structures* 104 (2021): 103312.

[11] Yang, Qing, Van Jones, and Leigh McCue. "Free-surface flow interactions with deformable structures using an SPH–FEM model." *Ocean engineering* 55 (2012): 136-147.

[12] Ng, K. C., A. Alexiadis, and Y. L. Ng. "An improved particle method for simulating fluid-structure interactions: The multi-resolution SPH-VCPM approach." *Ocean Engineering* 247 (2022): 110779.

[13] Ng, K. C., A. Alexiadis, Hailong Chen, and T. W. H. Sheu. "A coupled smoothed particle hydrodynamics-volume compensated particle method (SPH-VCPM) for fluid structure interaction (FSI) modelling." *Ocean Engineering* 218 (2020): 107923.

[14] Ng, K. C., A. Alexiadis, Hailong Chen, and T. W. H. Sheu. "Numerical computation of fluid–solid mixture flow using the SPH–VCPM–DEM method." *Journal of Fluids and Structures*106 (2021): 103369.

[15] Souto Iglesias, Antonio, Sergio Idelsohn, J. Marti, Ricardo Zamora Rodriguez, and Eugenio Oñate Ibáñez. "Modeling of free surface flows with elastic bodies interactions." (2008): 1-17.

[16] Yilmaz, Ada, Selahattin Kocaman, and Mustafa Demirci. "Numerical modeling of the dam-break wave impact on elastic sluice gate: A new benchmark case for hydroelasticity problems." *Ocean Engineering* 231 (2021): 108870.