Journal of Engineering and Applied Sciences 11 (4): 788-794, 2016
ISSN: 1816-949X

Development of Moving Particle Semi-Implicit Method and Simulation of Flow over Flip Buckets

Masoud Arami-Fadafan and Masoud-Reza Hessami-Kermani
Department of Civil Engineering, Shahid Bahonar University of Kerman, Kerman, Iran

Abstract: A modified MPS method has been proposed and implemented in simulations which generated smooth computational pressure fields. The method utilizes of one row wall boundary condition. This property has the ability of defining complex geometries. In this study, flow over flip bucket has been simulated which has curve line in its geometry. The applied method eliminates unphysical pressure fluctuations that occur in original MPS method.

Key words: Pressure fluctuations, one row wall boundary condition, flip bucket, MPS, flow over

INTRODUCTION

Due to advances in computer technology and developments in numerical methods, numerical simulations of various phenomena are in interest. Utilizing of numerical simulations may cause to decrease number of physical modeling which causes to decrease of costs and efforts. Among of various Lagrangian methods, Moving Particle Semi-implicit (MPS) and Smooth Particle Hydrodynamics (SPH) methods are widely used in engineering problems such as astrophysics (Gingold and Monaghan, 1977), continuum solid (Hieber and Koumoutsakos, 2008), fluid mechanics (Koshizuka and Oka, 1996) and heat transfer (Clearly, 1998). Using Lagerangian methods, problem of mesh adaptability and connectivity eliminated because in this method, the state of a system is represented by a set of discrete particles, without a fixed connectivity; hence, such methods are inherently well suited for the analysis of large deformations and fragmentations (Asai *et al*., 2012). Moreover, numerical diffusion is eliminated in Lagarangian methods due to the existence of advection terms in the Navier-stokes equations (Shobeyri and Afshar, 2012).

Furthermore, detecting of the free surface simply identified in Lagrangian methods using a simple criterion. In this study, modified MPS method is introduced and implemented in simulations. MPS method originally proposed by Koshizuka and Oka (1996) is a fully Lagerangian method that is based on particles characteristics and their movement. Characteristics of fluid/material such as density, viscosity, velocity and pressure are assigned to each particle and coordinates of particles are calculated based on these properties and initial conditions using MPS discretization of governing equations. The governing equations are mass and momentum conservation that are solved in Lagrangian form using a two-step fractional method. In this manner, each time step is divided into two basic steps. At the first step or prediction step, the equations governing the flow characteristics of the fluid particles are solved without considering the pressure term and the provisional velocity and position of each particle are obtained. Because of omitted pressure term in first step in governing equations, an unaccepted compressibility occurs in fluids. Hence, in the second step or correction step the incompressibility must be enforced in the calculations through of an equation of state in WC-MPS or Poisson equation of pressure in I-MPS method. Despite of advantages of particles method and numerous successes in MPS simulations in hydraulic problems such as dam break (Koshizoka and Oka, 1996; Asai *et al*., 2012), Solitary wave (Monaghan and Kos, 2000; Ataie-Ashtani *et al*., 2008), open channel flows (Shakibaeinia and Jin, 2009; Husain *et al*., 2014), hydraulic jump (Lopez *et al*., 2010), multi-phase flow (Shakibaeinia and Jin, 2012), sloshing flow (Gotoh *et al*., 2014), fluid-structure interaction problems (Hwang *et al*., 2014), this method is rarely implemented in problems with curved boundaries due to difficulties in assigning particle number density for particles on curved boundaries. In this study, some modifications in I-MPS method are proposed and applied in simulations that cause to obtain more smooth computational pressure fields. Proposed method has the ability of use of just one row wall boundary which simplifies defining. Complex geometry with curved lines.

MATERIALS AND METHODS

Governing equations: The governing equations of viscous fluid flows that are mass and momentum conservation equations are presented in the following:

Corresponding Author: Masoud Arami-Fadafan, Department of Civil Engineering, Shahid Bahonar University of Kerman, Kerman, Iran

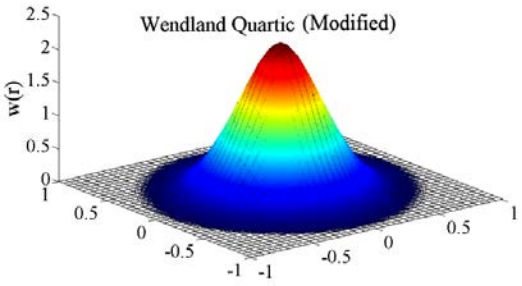

Fig. 1: Modified wendland 2D function

$$\frac{1}{\rho}\frac{D\rho}{Dt} + \nabla u = 0 \tag{1}$$

$$\frac{Du}{Dt} = -\frac{1}{\rho}\nabla P + \frac{\mu}{\rho}\nabla^2 u + \vec{g} \tag{2}$$

Where:

ρ = The density
u = The velocity vector
t = The time
P = The pressure
μ = The dynamic viscosity and is the gravitational acceleration

In Lagrangian coordinates the convection terms are directly calculated by the motion of particles, therefore, they are removed from the left side of the momentum equation. The right side terms expressed by differential operators should be replaced by particle interactions.

Particle interaction models: MPS interaction model is built on a set of disordered points in a continuum without a grid or mesh. A particle (i) interacts with others (j) in its vicinity covered with kernel function $w(r_{ij}, r_e)$ where r_{ij} is the distance between particles i and j and r_e is the cutoff radius of the interaction area. The kernel function is considered as a smoothing function of physical quantities around each particle. In this study, modified Wendland 2D kernel function is employed Fig. 1:

$$w(r_{ij}, r_e) = \begin{cases} \frac{7}{\pi h^2}(1 + 4(\frac{r_{ij}}{r_e}))(1 - (\frac{r_{ij}}{r_e}))^4 & 0 \le r_{ij} \le r_e \\ 0 & r_{ij} > r_e \end{cases} \tag{3}$$

It is possible to apply another kernel function which is often used in literatures such as B-Spline or MPS standard kernel function. A dimensionless parameter that is called particle number density, represents the density of particles and denotes by:

$$\langle n \rangle_i = \sum_{j \neq i} w(|r_j - r_i|) \tag{4}$$

In this Equation, the contribution from particle itself is not considered. Fluid density is proportional to the particle number density and defined as Eq. 5:

$$\langle ? \rangle_i = \frac{m\langle n \rangle_i}{\int_v w(r)dv} \tag{5}$$

m is the mass of each particles and assumed to be the same for all particles of determined fluid. Thus, the continuity equation is satisfied if the particle number density is constant. This constant value is denoted by n^0 (Koshizuka and Oka, 1996). Differential operators such as gradient and Laplacian, are represented by the following particle interaction models using the weight function:

$$\langle \nabla \phi \rangle_i = \frac{d}{n^0} \sum_{j \neq i} \left[\frac{\phi_j - \phi'_i}{|r_j - r_i|^2} (r_j - r_i) w(|r_j - r_i|) \right] \tag{6}$$

$$\langle \nabla^2 \phi \rangle_i = \frac{2d}{n^0 ?} \sum_{j \neq i} \left[(\phi_j - \phi_i) w(|r_j - r_i|) \right] \tag{7}$$

Where:

d = The number of spatial dimensions
ϕ = An arbitrary scalar.

The Laplacian model parameter λ is a correction parameter which cause variance increase be equal to the analytical solution and is defined as:

$$\lambda = \int_v w(r) r^2 dv \Big/ \int_v w(r) dv \tag{8}$$

The current model of Laplacian is conservative since the quantity lost by particle i is just obtained by particles j (Fadafan, 2014).

I-MPS solution algorithm: Implying projection method (Cummins and Rudman, 1999) for time integration, each time step is split into two parts of basic steps namely prediction and correction. Following figures depicts original and modified MPS algorithm (Fig. 2 and 3).

Boundary conitions: Wall solid boundaries are represented by one line of particles. The Poisson equation of pressure is solved on these particles. In the absence of any contrivances, by approaching solid boundaries, the density of particles decreases which causes to recognize the wall particles as free surface particles (Koshizuka and Oka, 1996). Thus, several lines of dummy particles are also

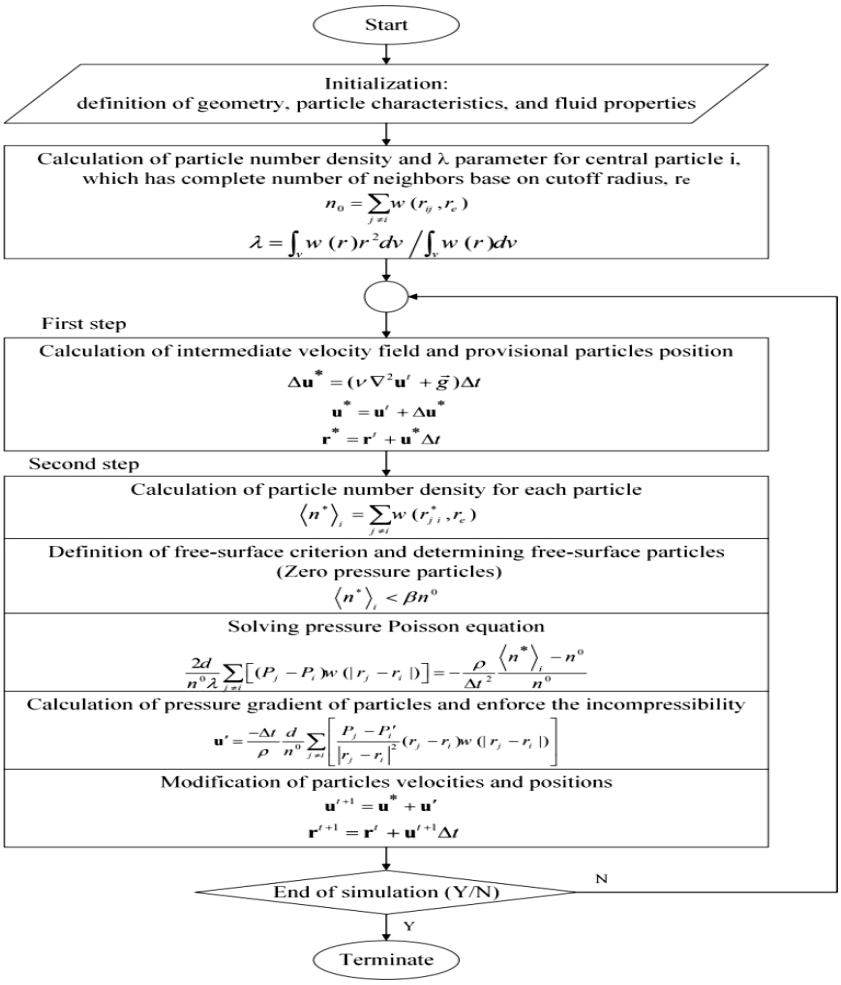

Fig. 2: Original MPS algorithm

placed outside of solid walls in order to keep the fluid density at the wall particles in standard MPS method. The thickness of dummy particles depends on the kernel range (Shao and Lo, 2003). However, in this study, dummy particles are eliminated in our I-MPS method and less value for criterion parameter (β) together with an auxiliary function are used for wall particles. The present method enables us to simulate curved wall boundaries which are of the interest in particle methods.

$$f(i) = \begin{cases} 1 & \text{if number of fluid particle} \\ & \text{in the vicinity of particle i} \geq 1 \\ 0 & \text{otherwise} \end{cases} \quad (9)$$

$$\beta_{wall}(i) = \begin{cases} 0.5\beta_{fluid} & (\text{radius cutoff} = 2.4l_0) \\ \beta_{fluid} & \text{otherwise} \end{cases} \quad (10)$$

Free surface particles: Known pressure boundary conditions are prescribed to the surface particles. Since, no fluid particle exists in the outer region of a free surface, the density of particles decreases by approaching the free surface. A particle which satisfies the following equations is considered as a free surface particle:

$$\begin{aligned} \langle n_{re} \rangle_i^* &< \beta_1 n_{re}^0 \\ \langle n_{re,lap} \rangle_i^* &< \beta_2 n_{re,lap}^0 \end{aligned} \quad (11)$$

where, β_i are the free surface parameters. This parameters are not effective to the calculation results if the calculation proceeds stably (Koshizuka and Oka, 1996). Usually parameter is chosen from 0.8-0.99.

Inflow boundary: Some layers of particles with prescribed velocity equal to inlet velocity are defined at the inflow

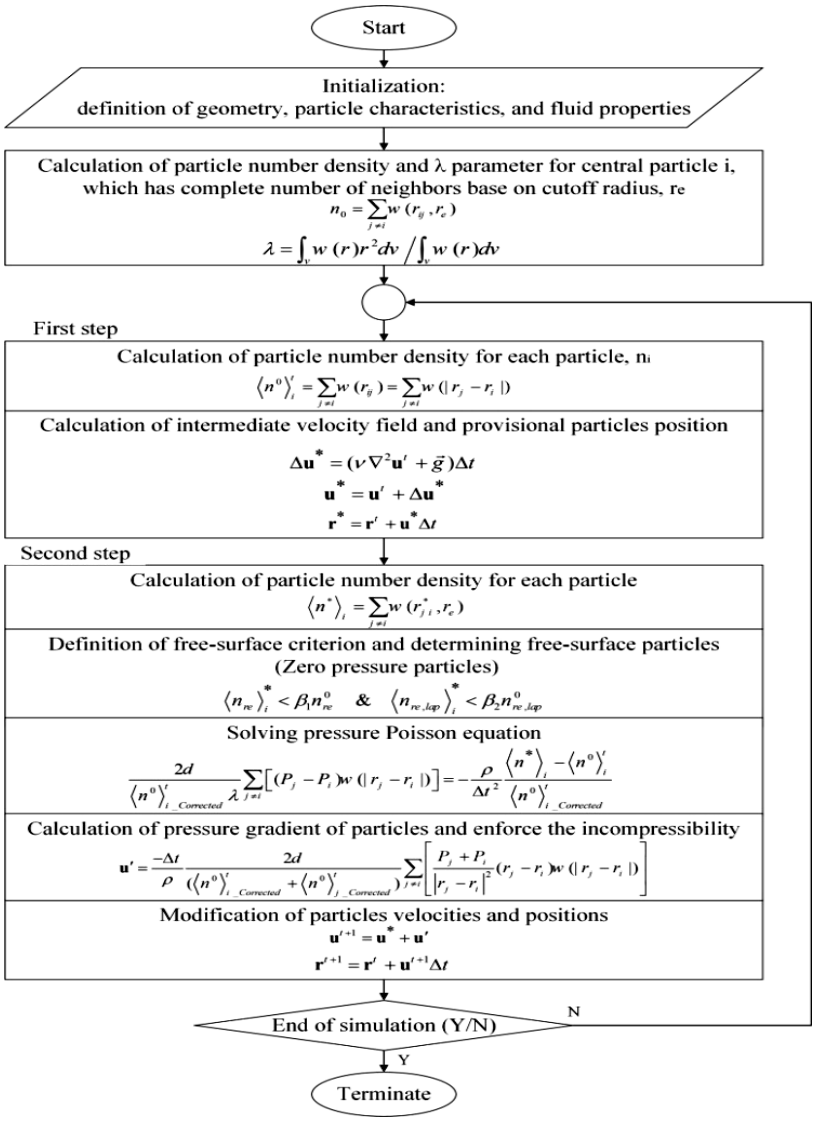

Fig. 3: Modified MPS algorithm

boundary to compensate for density deficiency. The particles on the inner first line of these particles are involved in the pressure calculation. Inlet particles are added to the distance between these layers and the fluid particles. For inlet particles, velocity in the flow direction is set to known velocity inflow until these particles move far from inlet position at least equal to a determined distance such as $1_0.1_0$ is the initial distance between neighboring particles in the initial configuration. Inflow particles are added to inflow boundary each K time step, based on inflow velocity, time step size and distance of particles.

Outflow: Each particle leaves the computational domain, eliminated from the computations.

RESULTS AND DISCUSSION

Model applications: Dam break: Dam break problem experimentally implemented by Hu and Kashiwagi (2004) is simulated using proposed I-MPS to show the accuracy of the method. The geometry of the experimental set up is shown in following Fig. 4-6.

Flow over a flip bucket: In this example, experimental study of (Heller *et al*., 2005) is used to simulated flow over a flip bucket. Their experiments were conducted in a rectangular channel with the total length of 7 m. The flip bucket consisted of a 1 m long horizontal approach channel. Schematic view of the experimental setup is shown in Fig. 7.

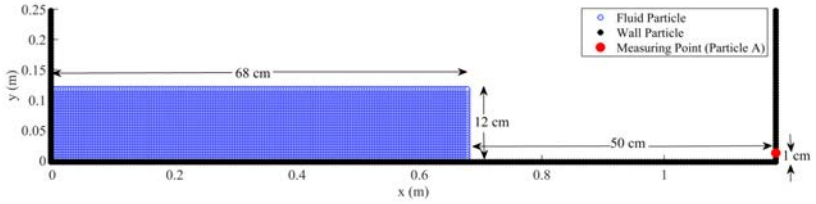

Fig. 4: Schematic view of experimental dam break set up

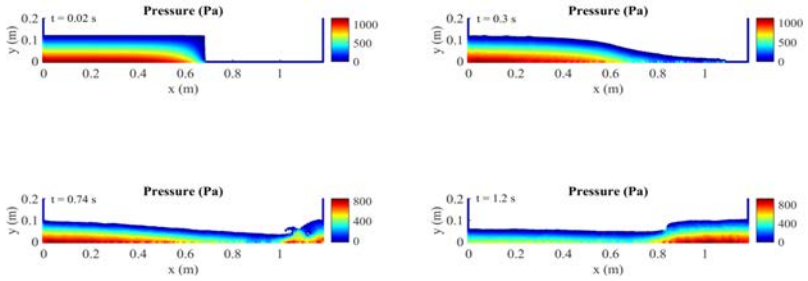

Fig. 5: Computational pressure fields

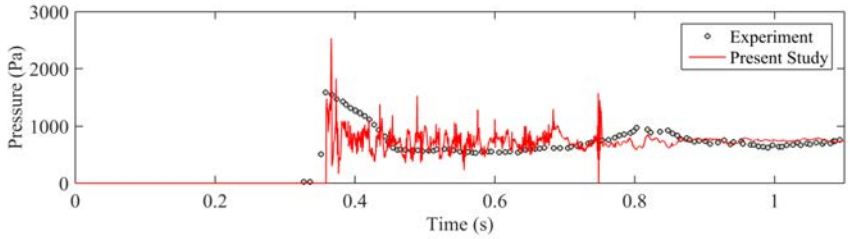

Fig. 6: Comparison of pressure history at the evaluation point (point A in Fig. 4) using present method and experimental results

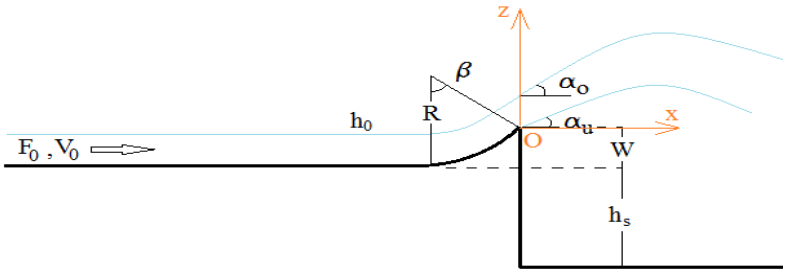

Fig. 7: Schematic view of flip bucket

Heller *et al.* (2005) suggested following equations to evaluate the location of maximum pressure, maximum pressure and pressure distribution on a flip bucket based on 91 case tests on six bucket deflection angles (β), tree bucket Radii and the approach flow depth (h_0) in the range of 0.036-0.095 m:

$$x_{PM} / R\sin\beta = -(1.5^\circ / \beta)^{1/5} \quad \text{for} \quad \beta \geq 15^\circ \tag{12}$$

$$h_{PM} / h_0 = (1/5)(\beta / 40^\circ)F_0^2 \quad \text{if } (h_0 / R)(40^\circ / \beta) \geq 0.20 \tag{13}$$

$$h_p / h_{PM} = [X_{PM}\exp(1 - X_{PM})]^{1.5} \tag{14}$$

where, X_{PM} the horizontal normalized coordinate ($x_{PM} = x/x_{PM}$) and is the location of maximum dynamic pressure head. Approach flow depth of 0.04 m, Froude number of 3, bucket deflection angle of and bucket radius of m is used in the simulation, results are depicted in following Fig. 8 and 9. Since, Eq. 14 not considered the effect of x/(R sin β)with β, following equation (Juon and Hager, 2000) is used to compare the results:

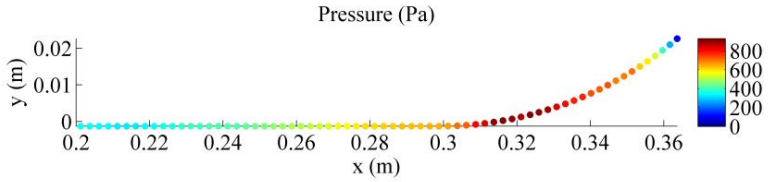

Fig. 8: Calculated pressure on flip bucket invert

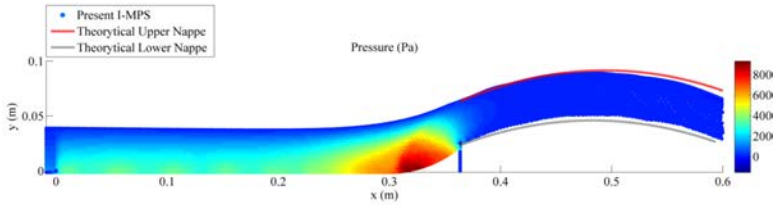

Fig. 9: Comparison of upper nappe and lower nappe together with computational pressure field

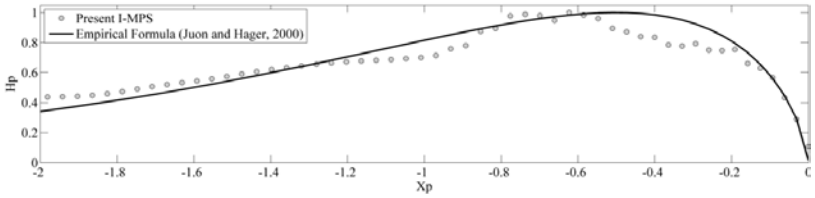

Fig. 10: Comparison of calculated non-dimensional pressure distribution over flip bucket invert and empirical formula

$$H_P = \frac{h_p - h_0}{h_{PM} - h_0} = \left[-2(X_p)\exp(1+2(X_p))\right]^{2/3} \qquad (15)$$

where is X_p = x/R sinβ dimensionless coordinate. Calculated non-dimensional pressure distribution over flip bucket invert and empirical formula result are compared in Fig. 10. As Fig. 10 shows there is good agreement between MPS results and empirical formula. This figure shows the accuracy and the ability of presented I-MPS to simulate the problems that consist of curved lines in their solid boundaries.

Obtained computational pressure field of this problem is in good agreement with pressure field computed by Shirkhani *et al.* (2014) using WC-SPH.

CONCLUSION

MPS method in its original form suffers from unphysical pressure fluctuations and use of this method without any contrivances accompanies serious pressure oscillations and numerical explosions. In this study, modified method has been used and new MPS algorithm solution with different modifications on Poisson pressure equation, gradient and Laplacian operators, wall boundary condition and free surface criteria introduced and implemented in simulations. As results shows pressure oscillations damped and smooth computational pressure field obtained. Modification on wall boundary condition redounded to simplify of utilizing MPS method for simulation of problems with curved lines.

Results of simulation of flow over flip bucket as a problem with curved line in its geometry shows the accuracy and ability of suggested method.

REFERENCES

Asai, M., A.M. Aly, Y. Sonoda and Y. Sakai, 2012. A stabilized incompressible SPH method by relaxing the density invariance condition. J. Applied Math., Vol. 2012. 10.1155/2012/139583

Ataie-Ashtiani, B., G. Shobeyri and L. Farhadi, 2008. Modified incompressible SPH method for simulating free surface problems. Fluid Dynamics Res., 40: 637-661.

Cleary, P.W., 1998. Modelling confined multi-material heat and mass flows using SPH. Applied Math. Model., 22: 981-993.

Cummins, S.J. and M. Rudman, 1999. An SPH projection method. J. Comput. Phys., 152: 584-607.

Gingold, R.A. and J.J. Monaghan, 1977. Smoothed particle hydrodynamics: Theory and application to non-spherical stars. Monthly Notices Royal Astronomical Soc., 181: 375-389.

Gotoh, H., A. Khayyer, H. Ikari, T. Arikawa and K. Shimosako, 2014. On enhancement of Incompressible SPH method for simulation of violent sloshing flows. Applied Ocean Res., 46: 104-115.

Heller, V., W.H. Hager and H.E. Minor, 2005. Ski jump hydraulics. J. Hydraulic Eng., 131: 347-355.

Hieber, S.E. and P. Koumoutsakos, 2008. A Lagrangian particle method for the simulation of linear and nonlinear elastic models of soft tissue. J. Comput. Phys., 227: 9195-9215.

Hu, C. and M. Kashiwagi, 2004. A CIP-based method for numerical simulations of violent free-surface flows. J. Mar. Sci. Technol., 9: 143-157.

Husain, S.M., J.R. Muhammed, H.U. Karunarathna and D.E. Reeve, 2014. Investigation of pressure variations over stepped spillways using smooth particle hydrodynamics. Adv. Water Resour., 66: 52-69.

Hwang, S.C., A. Khayyer, H. Gotoh and J.C. Park, 2014. Development of a fully lagrangian MPS-based coupled method for simulation of fluid-structure interaction problems. J. Fluids Struct., 50: 497-511.

Juon, R. and W.H. Hager, 2000. Flip bucket without and with deflectors. J. Hydraulic Eng., 126: 837-845.

Koshizuka, S. and Y. Oka, 1996. Moving-particle semi-implicit method for fragmentation of incompressible fluid. Nuclear Sci. Eng., 123: 421-434.

Lopez, D., R. Marivela and L. Garrote, 2010. Smoothed particle hydrodynamics model applied to hydraulic structures: A hydraulic jump test case. J. Hydraulic Res., 48: 142-158.

Monaghan, J.J. and A. Kos, 2000. Scott Russell's wave generator. Phys. Fluids, 12: 622-630.

Shakibaeinia, A. and Y.C. Jin, 2009. Lagrangian modeling of flow over spillways using the moving particle semi-implicit method. Proceedings of the 33rd IAHR Congress, Water Engineering for a Sustainable Environment, August 9-14, 2009, Vancouver.

Shakibaeinia, A. and Y.C. Jin, 2012. MPS mesh-free particle method for multiphase flows. Comput. Methods Applied Mech. Eng., 229: 13-26.

Shao, S. and E.Y. Lo, 2003. Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface. Adv. Water Resour., 26: 787-800.

Shirkhani, H., M. Khanpour and A.R. Zarrati, 2014. Development of mesh-free particle technique for simulation of non-hydrostatic free surface flow. Comput. Fluids, 105: 8-15.

Shobeyri, G. and M.H. Afshar, 2012. Corrected discrete least-squares meshless method for simulating free surface flows. Eng. Anal. Boundary Elements, 36: 1581-1594.