MODELING OF FLOW OVER AN OGEE SPILLWAY USING MOVING PARTICLE SEMI-IMPLICIT METHOD

MASOUD ARAMI FADAFAN(1), MASOUD-REZA HESSAMI KERMANI(2)

(1) *Ph.D. Student, Department of Civil Engineering, Faculty of Engineering, Shahid Bahonar University of Kerman, Kerman, Iran, Aramimasoud@eng.uk.ac.ir*

(2) *Assistant professor, Department of Civil Engineering, Faculty of Engineering, Shahid Bahonar University of Kerman, Kerman, Iran, Hessami@uk.ac.ir*

ABSTRACT

Mesh free methods are used in wide range of engineering problems such as astrophysics, continuum solid, fluid mechanics and heat transfer. Mesh free particles based methods are suitable to simulate flows with large deformations and fragmentations due to using discrete particles. Despite of capabilities of these methods, they are rarely used for modeling of open channel flows, specially in the case of ogee spillways. Usually based mesh methods such as finite volume, finite element or finite difference are used to simulate open channel problems which have provided satisfactory results, however their success rely on good meshes. In the Eulerian methods, numerical diffusion due to advection terms may be severe which is eliminated in Lagarangian methods due to the existence of advection terms in the Navier-stokes equations. Furthermore, detecting of the free surface is a complicated procedure in the Eulerian methods which is simply identified in Lagrangian methods using a simple criterion. In this paper, flow over an ogee spillway is modeled using an incompressible Moving Particle Semi-implicit (MPS) method which is a fully Lagrangain meshless method. The governing equations are mass and momentum conservation that are solved in Lagrangian form using a two-step fractional method. In the first step momentum equation is solved without considering the pressure term. In this step velocity field and the position of particles are computed but incompressibility is neglected. In the second step, to satisfy the incompressibility condition, the continuity constraint is considered and the Poisson equation is solved to compute the pressure values of fluid and wall particles. Then the velocity field and positions of particles are corrected. The MPS results are compared with those of experimental ones in free surface comparison. Moreover, velocity and pressure fields are depicted in different times and compared with those of obtained using Flow-3D simulation. The results show the capabilities of used method in open channel flow simulations.

Keywords: Mesh free particle methods, Moving Particle Semi-implicit, ogee spillway, Flow-3D

1. INTRODUCTION

Historically, physical models have been constructed in hydraulic laboratories to study of flow behaviors and measuring of forces on hydraulic structures. But constructing and utilizing of physical models are expensive and time-consuming. Moreover, there are many difficulties associated with scaling effects in physical hydraulic models (Ho et al., 2003). In recent years, due to advances in computer technology and numerical methods, computational fluid dynamics (CFD) have been developed rapidly. The CFD models are more flexible with significant advantages such as need to require less time, expense, and efforts than physical hydraulic models. Therefore, they have been used extensively by engineers to model and to analyse complex issues about design of hydraulic structures and their maintenance, plan studies for future stations and dam safety. Aydin and Ozturk (2009) modeled spillway aerators using FLUENT software in order to study of introduced air to flow and evaluating of aerators performance. Johnson and Savage (2006) modeled ogee-crested spillways using FLOW-3D software and compared the gained numerical results with their physical measurements. The above mentioned studies and many other studies such as Bombardelli et al. (2010) have been done using different CFD softwares which are based on Eulerian mesh dependent method. These methods have provided many useful and satisfactory results. However their success largely rely on good quality meshes. Moreover the connectivity between nodes and elements must be carefully and accurately found and recorded (Ma, 2005). Owning to mesh adaptability and connectivity problems, mesh based methods have difficulties in modeling complicated phenomena with large deformation of boundaries and interfaces (Shakibaeinia and Jin, 2010). Using Lagerangian methods, problem of mesh adaptability and connectivity eliminated because 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). Mesh free methods are used in wide range of 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, 1993). Despite of capabilities of these methods, they are rarely used for modeling of open channel flows, specially in the case of ogee spillways. Shakibaeinia and Jin (2009) modeled flow over an ogee spillway using Weakly Compressible MPS (WC-MPS) and compared the obtained free surface results and experimental ones. However their model overestimates the free surface profile over the straight portion of the spillway. They concluded that this discrepancy can be result of some unphysical pressure fluctuation accompanied by the MPS method or it can be due to some error in prescription of the solid boundary condition in the straight part of the spillway in their model. In this paper, proposed method by Memarzadeh and Hejazi (2012) for SPH simulations is modified to MPS method and is used to simulate the flow over an ogee spillway which consist of several curved lines on its boundary. Validity of the modified MPS method has been shown previously in the case of Scott-Russell wave generator simulation (Arami, 2014). In this study numerical free surface results are in good agreement with physical ones. Moreover, the obtained pressure fields are smooth using the proposed method and are close to FLOW-3D results.

2. Numerical method

The governing equations of the incompressible flow are given as:

$$\frac{1}{\rho}\frac{D\rho}{Dt}+\nabla.\mathbf{u}=0 \quad [1]$$

$$\frac{D\mathbf{u}}{Dt}=-\frac{1}{\rho}\nabla P+\frac{\mu}{\rho}\nabla^2\mathbf{u}+\vec{g} \quad [2]$$

Equation (1) is the continuity and equation (2) is the momentum conservation equation. In the above equations ρ is the density, $\mathbf{u}$ is the velocity vector, t is the time, P is the pressure, μ is the dynamic viscosity and g is the gravitational acceleration. Left side of the Navier-Stokes equation (Eq. 2) denotes the Lagerangian differentiation including convection terms. This is directly calculated by tracing particle motion. Therefore, the problem of numerical diffusion due to the existence of advection terms in the Navier-Stokes equations is completely eliminated. The right side terms expressed by differential operators should be replaced by particle interactions.

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:

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

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 n.

$$\langle n\rangle_i=\sum_{j\ne i}w(|r_j-r_i|) \quad [4]$$

In this equation, the contribution from particle i itself is not considered. Fluid density is proportional to the particle number density and defined as following equation:

$$\langle\rho\rangle_i=\frac{m\langle n\rangle_i}{\int_v w(r)dv} \quad [5]$$

m is the mass of each particles and assumed to be the same for all particles of specified fluid. Thus, the continuity equation is satisfied if the particle number density is constant. This constant value is denoted by n^0. The differential operators of gradient, Laplacian and divergence are formulated as:

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

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

$$\langle\nabla.\Phi\rangle_i=\frac{d}{n^0}\sum_{j\ne i}\left[\frac{\Phi_j-\Phi_i}{|r_j-r_i|^2}\cdot(r_j-r_i)w(|r_j-r_i|)\right] \quad [8]$$

where d is the number of spatial dimensions, ϕ is an arbitrary scalar and Φ is an arbitrary vector. Any constant value is allowed to ϕ_i' (Koshizuka and Oka, 1996). In this paper ϕ_i' is calculated from following equation:

$$\phi_i'=\min(\phi_j)\quad for\quad \{j\,|w(|r_j-r_i|)\ne 0\} \quad [9]$$

This means that the minimum value is selected among the neighboring particles in the support domain. Using Equation (9), forces between particles are always repulsive (Koshizuka et al., 1998). It is possible to use the following equation instead of Eq. 6 that is suggested by Toyoda et al. (2005) in order to reduce the pressure fluctuations (Lee et al., 2011).

$$\langle\nabla\phi\rangle_i=\frac{d}{n^0}\sum_{j\ne i}\left[\frac{\phi_j+\phi_i}{|r_j-r_i|^2}(r_j-r_i)w(|r_j-r_i|)\right] \quad [10]$$

The Laplacian model parameter λ is introduced to keep the same variance increase as that of analytical solution and is defined as:

$$\lambda=\int_v w(r)r^2dv\Big/\int_v w(r)dv \quad [11]$$

The current model of Laplacian is conservative since the quantity lost by particle i is just obtained by particles j (Koshizuka and Oka, 1996). Using the Laplacian model in Poisson equation of pressure, simultaneous equations

expressed by a linear symmetric matrix are solved to obtain the particles pressure. To reach this matrix, following procedure explained by Koshizuka and Oka (1996) must be done.
The continuity equation requires that the fluid density should be constant. This is equivalent to the particle number density being constant, n^0. When the fictitious particle number density n^* is not n^0, it is implicitly corrected to n^0 by:

$$n^* + n' = n^0 \quad [12]$$

Where n' is the correction value. This is related to the velocity correction value $\mathbf{u}'$ through the mass conservation equation:

$$\frac{1}{\Delta t}\frac{n'}{n^0} = -\nabla.\mathbf{u}' \quad [13]$$

The velocity correction value is derived from the implicit pressure gradient term as:

$$\mathbf{u}' = \frac{-\Delta t}{\rho}\nabla P^{n+1} \quad [14]$$

By combining Eqs. (14) and (1), a Poisson equation of pressure is obtained:

$$\left\langle \nabla^2 P^{n+1} \right\rangle_i = -\frac{\rho}{\Delta t^2}\frac{\left\langle n^* \right\rangle_i - n^0}{n^0} \quad [15]$$

Once the pressure of each particle is obtained by solving the above equation, the gradient model (Eq. 6) is used to find pressure gradient and finally using the following equations, velocity and position of fluid particles can be computed.

$$\mathbf{u}^{n+1} = \mathbf{u}^* + \mathbf{u}' \quad [16]$$

$$\mathbf{r}^{n+1} = \mathbf{r}^n + \mathbf{u}^{n+1}\Delta t \quad [17]$$

Where $\mathbf{r}^{n+1}, \mathbf{r}^n$ = particle positions at time $n+1$ and n.

3. Boundary condition

3.1 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 equation is considered as a free surface particle:

$$\left\langle n \right\rangle_i^* < \beta n^0 \quad [18]$$

Where β is the free surface parameter. This parameter is not effective to the calculation results if the calculation proceeds stably (Koshizuka and Oka, 1996). Usually this parameter is chosen from 0.8 to 0.99.

3.2 Wall particles

In the meshfree methods, walls are represented by particles. The Poisson equation of pressure is solved for these particles. This balances the pressure of inner fluid particles and prevents them from accumulating in the vicinity of solid boundaries.

3.3 Inflow boundary

Some layers of particles with prescribed velocity equal to inlet velocity are defined at the inflow 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 l_0. l_0 is the initial distance between neighboring particles in the initial configuration. Inflow particles are added to inflow boundary each K time step, related to inflow velocity, time step size and distance of particles.

3.4 Outflow

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

4. Simulating of the problem

The experimental condition and measurements performed by Chatila and Tabbara (2004) were used to simulate the flow over an ogee spillway. The geometry of the spillway crest is based on the hydraulic design charts of USACE-WES profiles. The upstream face of the spillway is vertical and the radius of the upstream curved surface defined by $0.2H_d$ and $0.5H_d$. The downstream profile of the crest centerline is defined by $x^{1.85} = 2H_d^{0.85}y$ and $H_d = 0.0508$ cm. The slope of the straight portion of the spillway face is 60° (or a slope $1.73:1$).

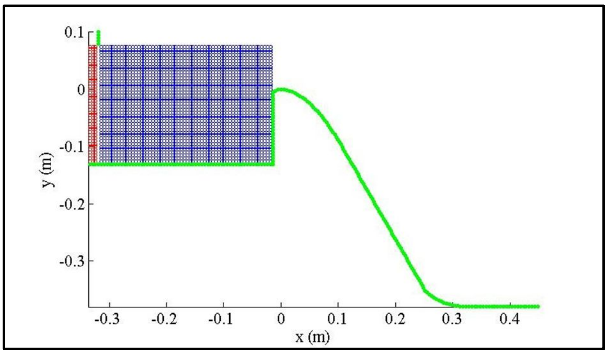

Figure 1. Initial Particle Position

5. Numerical results

Fig. 2 shows the time history of particles position together with velocity fields downstream of the spillway crest at $t = 0.1\,s$ and $t = 0.3\,s$ resulted from the modified MPS simulation and FLOW-3D software. The flow over spillway has a discharge of $0.0467\,(m^3/s)$. The MPS numerical flow over the current spillway has good agreement with those of FLOW-3D ones. This figure shows the ability of the presented incompressible MPS method for simulation of flow over spillways as an open channel problem. Same spillway and flow condition is used to simulate the flow over ogee spillway by Shakibaeinia and Jin (2009) utilizing weakly compressible MPS method. Their model overestimates the surface elevation and they conclude that this discrepancy can be result of some unphysical pressure fluctuation accompanied by the MPS method or it can be due to some error in prescription of the solid boundary condition in the model at spillway portion. In Fig. 2, there is no separation between flow and spillway bottom, which shows the true prescription of solid boundary condition of current problem.

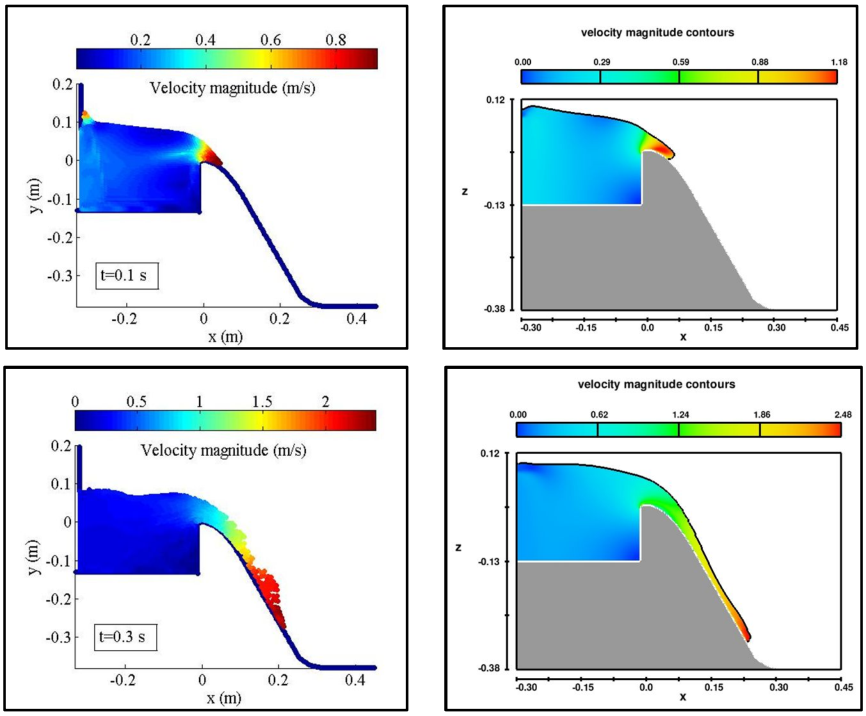

Figure 2. Comparison of velocity Field between MPS (left side) and FLOW-3D (right side) results

Fig. 3 illustrates the computational flow pressure fields at $t = 0.05\,s$ and $t = 0.25\,s$. Using the proposed MPS method (Arami, 2014), smooth pressure fields are obtained.

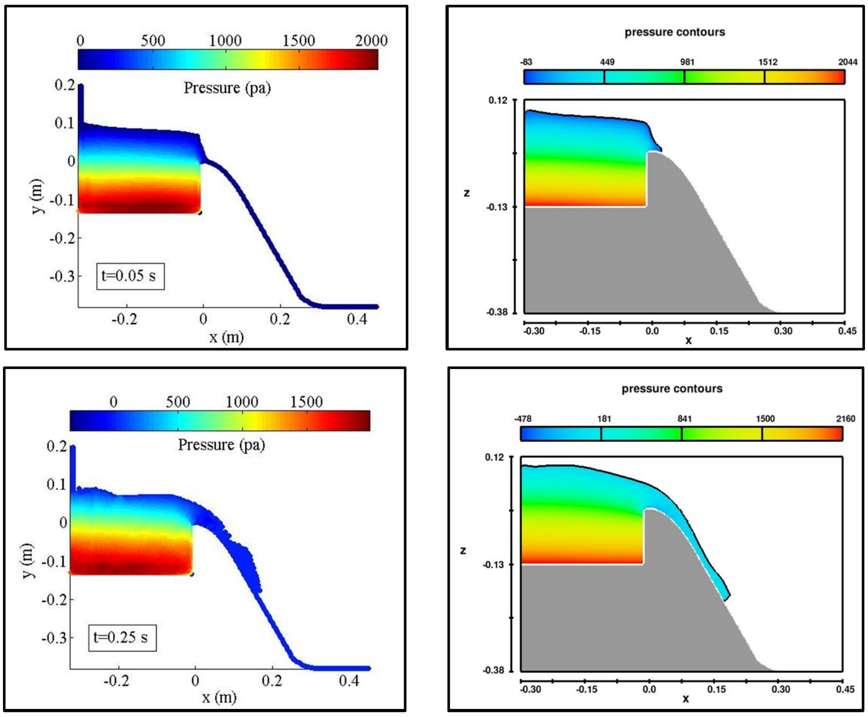

Figure 3. Computational pressure fields, MPS (left side) and FLOW-3D (right side)

Velocity vectors gained by MPS method are shown in Fig.4 at $t = 0.35$ s.

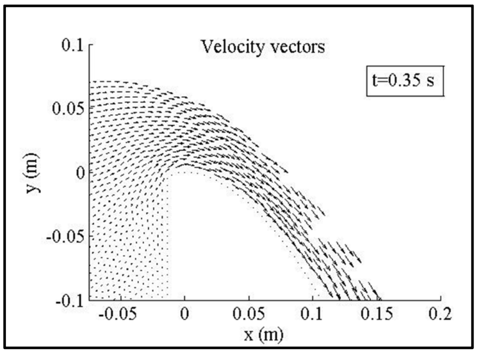

Figure 4. Velocity vectors of flow close to spillway crest

As Fig. 4 shows, flow modeled by MPS method close to spillway crest is smooth. This proves that the configuration of solid particles is true and the interaction between these particles and flow particles are done correctly. Fig. 5 depicts comparison of experimental and the MPS free surface together with velocity field at $t = 0.6$ s. As it shows there is good agreement between the numerical results and experimental measurements.

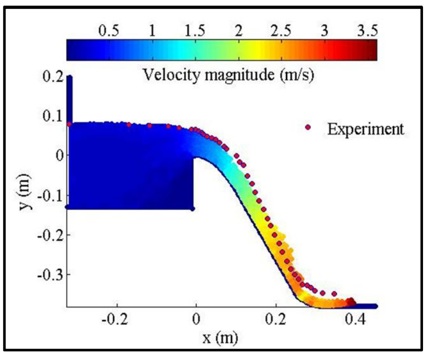

Figure 5. Comparison of experimental and the MPS free surface

6. CONCLUSIONS

Flow over an ogee spillway is modeled using an incompressible Moving Particle Semi-implicit (MPS) method which is a fully Lagrangain meshless method. In this paper, although the reservoir of dam is modeled smaller than experiment in order to reduce the CPU time, but the MPS results are in good agreement with those of experimental ones in free surface comparison. Moreover, velocity and pressure fields are depicted in different times and compared with those of obtained using Flow-3D simulation. The results show the capabilities of used method in open channel flow simulations.

REFERENCES

Arami Fadafan M., (2014). Simulating of Scott Russell Wave Generator Using a Modified Moving Particle Semi-implicit Method, *11th International Conference on Coasts, Ports and Marine Structures (ICOPMASS)*, Tehran, Iran.

Asai M., Aly A.M., Sonoda Y., and Sakai Y. (2012). A Stabilized Incompressible SPH Method by Relaxing the Density Invariance Condition, *Journal of Applied Mathemathics*, Article ID 139583, 24 pages.

Aydin M.C., Ozturk M. (2009). Verification and Validation of a Computational Fluid Dynamics (CFD) Model for Entrainment at spillway Aerators, *Canadian Journal of Civil Engineering*, Vol. 36, pp. 826-836.

Bombardelli F.A., Meireles I., and Matos J. (2011). Laboratory measurments and multi-block numerical simulation of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways, *Journal of Environmental Fluid Mechanics*, Vol. 11, pp. 263-288.

Chatila J. and Tabbara M., (2004). Computational modeling of flow over an ogee spillway, *Journal of Computers and Structures*, Vol. 82, pp. 1805-1812.

Clearly P. (1993). Modelling confined multi-material heat and mass flows using SPH, *Journal of Applied Mathematical Modelling*, Vol. 22, pp. 981-993.

Gingold R.A., and Monaghan J.J. (1977). Smoothed particle hydrodynamics: theory and application to non-spherical stars, *Monthly Notices of the Royal Astronomical Society*, Vol. 181, pp. 375-389.

Hieber S.E., and Koumoutsakos P. (2008). A Lagrangian particle method for the simulation of linear and nonlinear elastic models of soft tissue, *Journal of Computational Physics*, Vol. 21, pp. 9195-9215.

Ho D., Boyes K., Donohoo S., and Cooper B. (2003). Numerical Flow Analysis for Spillways, *43rd ANCOLD Conference*, Hobart, Tasmania.

Johnson M.C. and Savage B.M. (2006). Physical and Numerical Comparison of Flow over Ogee Spillway in the Presence of Tailwater, *Journal of Hydraulic Engineering*, Vol. 132, No. 12, pp. 1353-1357.

Koshizuka S., Nobe A., and Oka Y. (1998). Numerical analysis of breaking waves using the moving particle semi-implicit method, *International Journal for Numerical Methods in Fluids*, Vol. 26, pp. 751-769.

Koshizuka S., and Oka Y. (1996). Moving Particle Semi-implicit method for fragmentation of incompressible fluid, *Journal of Nuclear Science and Engineering*, Vol. 123, pp. 421-434.

Lee B.H., Park J.C., Kim M.H. and Hwang S.C., (2011). Step-by-step improvement of MPS method in simulating violent free-surface motions and impact-loads, *Journal of Computer Methods in Applied Mechanics and Engineering*, Vol. 200, pp. 1113-1125.

Ma Q.W., (2005). Meshless local Petrov-Galerkin method for two-dimensional nonlinear water wave problems, *Journal of Computational Physics*, Vol. 205, pp. 611-625.

Memarzadeh M., and Hejazi K. (2012). ISPH Numerical Modeling of Nonlinear Wave Run-up on Steep Slopes, *Journal of the Persian Gulf*, Vol. 3, No. 10, pp. 17-26.

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

Shakibaeinia A., and Jin Y.C. (2010). A weakly compressible MPS method for modeling of open-boundary free-surface flow, *International Journal for Numerical Methods in Fluids*, Vol. 63, pp. 1208-1232.