【Memoirs of Construction Engineering Research Institute Vol. 58 (report) Nov. 2016】

APPLICATION OF SMOOTHED PARTICLE HYDRODYNAMICS WITH WALL TURBULENCE MODEL TO COMPUTATION OF HIGH SPEED SPILLWAY FLOWS

Akihiko Nakayama Leong Lap Yan Kong Wei Song

ABSTRACT

The density re-initialization technique and the wall model are introduced to the weakly compressible smoothed hydrodynamics (WCSPH) to improve the prediction of high speed turbulent flows by large eddy simulation. The improved method is examined by comparing with the experimental results of flow past ski-jump flip bucket and the corresponding calculation results of existing method. The nappe flow trajectory and the spread of the flow are reproduced much better than the existing FEM method. It is due to the improved representation of the wall shear stress and the near wall near the flip bucket. The density re-initialization contributed to the stable distribution of the density and the pressure.

1. INTRODUCTION

Spillways are the passages of excess water to be discharged from reservoirs safely and effectively [1]. They involve large energy to be dissipated and the flow tends to become very fast and violent. The analysis of the flow for the design of such facilities is not easy and methods have been mostly empirical. In recent years, mesh-free particle-based method of simulating continuous flows has been developed and successfully applied to various hydraulic engineering problems (e.g. Monaghan[2], Violeau and Issa[3]). The smoothed particle hydrodynamics (SPH) formulation was first developed for solving problems of astrophysics[4], but Monaghan[2, 5] showed that it can be extended to represent free surface flows very well. In these and many of the work following it (reviewed in Liu and Liu[6], Monaghan[7]), an incompressible fluid is approximated by a weakly compressible fluid with sufficiently large speed of sound compared with the speed of the bulk flow. The pressure is determined from an assumed state equation as a function of the density. This weakly compressible approximation of the incompressible flow is a subtle compromise between an efficient time advancing and a slight mismatch in the density and pressure distributions. Methods that require strict incompressibility in the same particle interpolation framework for discretization have also been developed. Incompressible Smoothed Particle Hydrodynamics (ISPH) method[8, 9] and Moving Particle Semi-implicit (MPS)[10] are such methods in which the Poisson equation for pressure is solved by an iterative method, but due to their computational inefficiency, they are usually not favored in practical applications. The original Weakly Compressible SPH (WCSPH) which has proved very successful in computation of many open-channel flows, has a few weaknesses which needs to be addressed in specific applications. The selection of the speed of sound for flows in which the velocity changes by large magnitude must be made carefully. The density calculated by the time-advancing manner may accumulate small mismatch between the density and the volume represented by the particles. Density re-initialization is one approach devised to solve this problem.

Another aspect of the original formulation of WCSPH is the treatment of the viscosity and the turbulence and the related modeling problems near solid surfaces. The original method was applied to almost inviscid flow and the viscosity was used only to damp instabilities. So the boundary conditions near the solid surfaces were taken slip condition and the effects of the boundary layer near solid walls are not represented properly. The present authors[11] have implemented the wall similarity boundary conditions on the instantaneous basis so that the method can be run like a Large Eddy Simulation (LES) that calculates the motion of large eddies but model the small-scale motion. This SPH based LES was further extended to simulate the motion of the gas as well as that of the liquid for the flows with mutual effects[12].

In the present work, we show that the application of the density re-initialization technique implemented in the liquid phase calculation improves stability and can be used to compute high-speed spillway flows. We find that this single-phase flow approach is sufficient to simulate flows with nappes and significant mixing of air and water when there is no gas trapped within liquid. The case of ski-jump flow past flip bucket is considered.

2. SPH METHOD WITH DENSTY RE-INITIALIZATION

2.1 Basic governing equations

In SPH the basic equations of motion are written for discrete particles representing continuous fluid medium. The momentum equation for particle *a* located at $\boldsymbol{r}_a$ with velocity $\boldsymbol{v}_a$ is

$$\frac{d\boldsymbol{v}_a}{dt} = -\sum_b \frac{m_b}{\rho_a}\left(\frac{p_a}{\rho_a} + \frac{p_b}{\rho_b} + \Pi_{ab}\right)\nabla_a W_{ab} + \boldsymbol{g} \tag{1}$$

where p_a and ρ_a are the pressure and the density of fluid at $\boldsymbol{r}_a$, or those of particle *a*, $\nabla_a W_{ab}$ is the gradient of kernel function $W(\boldsymbol{r}_a - \boldsymbol{r}_b, h)$ with respect to the position $\boldsymbol{r}_a$, m_b and ρ_b are mass and the density of particle *b* located in the neighborhood of particle *a* so that the ratio m_b/ρ_b is the volume the particle *b* occupies and $\boldsymbol{g}$ is the gravitational acceleration. Π_{ab} is the term to represent the viscous stress given by the coefficients of the dynamic viscosity μ_a and μ_b of particles *a* and *b*, respectively. If the effects of unresolved small-scale turbulent motion are represented by the effective viscosity as in many LES methods, they can be expressed as the additional stress with the viscosity coefficients μ_{ta}, μ_{tb} of particles *a* and *b*. Then the sum of the viscous and the turbulent stresses is written as

$$\Pi_{ab} = -\frac{C_\mu(\mu_a + \mu_{ta})(\mu_b + \mu_{tb})}{\rho_b(\mu_a + \mu_{ta} + \mu_b + \mu_{tb})} \frac{(\boldsymbol{v}_a - \boldsymbol{v}_b)\cdot(\boldsymbol{r}_a - \boldsymbol{r}_b)}{h\sqrt{|\boldsymbol{r}_a - \boldsymbol{r}_b|^2 + 0.01h^2}} \tag{2}$$

h that appears in *W* and in this equation is the smoothing length. This expression was proposed by Kajtar and Monaghan[13]. The summation over all *b* in the neighborhood approximates the integration over the volume with the radius of the order of *h* surrounding point $\boldsymbol{r}_a$.

The mass conservation equation is written as the equation for the density of particle a as

$$\frac{d\rho_a}{dt} = -\sum_b m_b \boldsymbol{v}_b \cdot \nabla_a W_{ab} \tag{3}$$

The above equations together with an assumed state equation for the pressure

$$p = B\left(\left(\frac{\rho}{\rho_0}\right)^\gamma - 1\right) \tag{4}$$

with assumed value of γ (taken to be 1/7 as in most other methods) and *B* and ρ_0 complete the system of ordinary differential equations for $\boldsymbol{v}_a$, p_a and ρ_a. The position $\boldsymbol{r}_a$ of particle *a* needed in these equations is related to the velocity by

$$\frac{d\boldsymbol{r}_a}{dt} = \boldsymbol{v}_b \tag{5}$$

The kernel function $W(\boldsymbol{r} - \boldsymbol{r}_b, h)$ is a function that approximates and approaches the Dirac delta function as length *h* approaches zero. The specific form of the kernel function is needed and we use the following cubic spline function

$$W_{ab} = \begin{cases} \frac{1}{6}\left[(2-q)^3 - 4(1-q)^3\right], & 0 \le q \le 1, \\ \frac{1}{6}(2-q)^3 \quad , & 1 \le q \le 2, \quad q = |\boldsymbol{r}_a - \boldsymbol{r}_b|/h \\ 0 \quad , & q > 2 \end{cases} \tag{6}$$

In the present work, we consider the following density re-initialization given by Shephard[14] (Voronyev[15]) instead of the solution of Eq.(3) once in several tens of time steps.

$$\rho_a = -\sum_b m_b W^*_{ab}, \quad W^*_{ab} = \frac{W_{ab}}{\sum_b \frac{m_b}{\rho_b} W_{ab}}, \tag{7}$$

2.2 Boundary conditions

Although the conditions imposed by the boundary surfaces of the flow region are referred to as the boundary conditions (e.g. 5),6)), the momentum equation Eq.(1) and the evolution equation of the density Eq.(3) are both ordinary differential equations in time and, strictly speaking, the solution of these equations for the velocity and the density do not require spatial boundary conditions. That is one of the reasons why the particle method is suited to compute the free surface flows in which the free surface does not impose any restriction to the motion of the fluid particles. Therefore, here we consider the conditions imposed by other boundaries. Figure 1 shows the typical calculation region. It also shows the boundaries which impose conditions other than the conservation of mass and momentum on the motion of the particles. In order to impose conditions such as the no-penetration and shear stress on the solid surface, we apply relations that are satisfied by the particles in the region close to the boundary and not exactly on the boundary. When a particle comes into this boundary region (taken as the distance of the order of h so that there is one and not more than one particle along the same line normal to the surface), these special relations are applied to the motion of the particles and the momentum and continuity equations are not solved in the way they are solved in the region away from the boundaries. Figure 1 indicates these special boundary regions where the governing equations are replaced by special boundary conditions. These are the region next to the solid surfaces, inflow region at the upstream side and the outflow region at the downstream end of the main calculation region. This approach is different from the common methods using boundary and/or image particles (Colagrossi and Landrini 16), Violeau and Issa3) or Gotoh et al.17)) but may be closer to the method to impose an artificial repulsive force3),5).

In the near wall region, the relation that is imposed is the shear force

$$\frac{\boldsymbol{F}_a}{\rho} = -C_D v_t v_a / h_b \tag{8}$$

exerted on the particle by the solid surface. Here $v_a = |\boldsymbol{v}_a|$ $\boldsymbol{v}_t$ is the tangential velocity and h_b is the thickness of the wall boundary layer and C_D is the resistance coefficient determined by the wall laws for turbulent flows on smooth or rough surfaces depending on the problem. It is noted the opposite force $\boldsymbol{T}_w = -\boldsymbol{F}_a$ is exerted to the solid wall. $\boldsymbol{F}_a$ is added to the right-hand side of Eq.(1) and solved for the tangential components. The velocity component v_n perpendicular to the surface is determined by the incompressible condition and the zero normal velocity on the surface

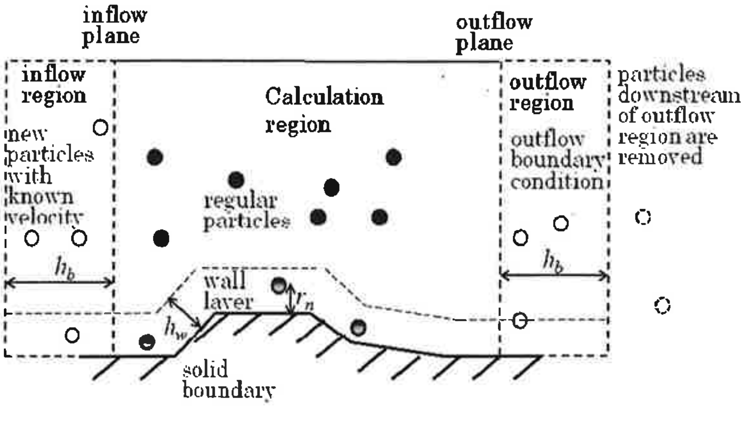


Figure 1. The boundary regions to set boundary conditions.

$$v_n = -\left(\frac{\partial v_{t1}}{\partial t_1} + \frac{\partial v_{t2}}{\partial t_2}\right) r_n \tag{9}$$

where v_{t1} and v_{t2} are the two velocity components tangent to the solid wall in t_1 and t_2 directions and r_n is the distance to the wall. In the strictly incompressible flow, this equation approximates that the velocity on the wall is zero.

In the inflow region set just upstream of the inflow plane, the velocity and the pressure are set to the prescribed inflow velocity and pressure. As these particles move at the prescribed velocity and the distance to the inflow plane becomes large than the initial particle separation, a new particle with the prescribed velocity and the pressure are introduced. Particles in the outflow region just downstream of the outflow plane, the velocities are not calculated but assumed constant until they translate out of this region.

For calculation of fully-developed steady flows in long channels, usually calculation in a short section is done as in many fixed-gird methods applying the ‘periodic boundary condition. In the particle method, with a link-cell method for finding the neighboring particles, we set the periodic condition on the link cells. In other words, the link cells at the most upstream are treated as the link cells neighboring the most downstream cells also and vise versa for the most downstream cells. In the application done in the following section, where the flow is periodic with the vertical shift, we shift the most upstream or most downstream cells.

3. SKI JUMP SPILLWAY WITH FLIP BUCKET

The main objective of the present work is to develop a particle method that can simulate high-speed flow past a ski jump spillway with a good accuracy. For that purpose a basic flow past a flip bucket that deflects a horizontal jet into the air, has been chosen as the test case. The experiment was conducted by Juon and Hager1[18] and the schematic of the flow is shown in Figure 2. The jetbox generates a supercritical high-speed flow without a steep slope spillway in the real spill way. The circular arc-shaped invert is installed at the end of the approach channel to deflect the flow upwards. The flip angle β is either 20deg or 35 deg. Figure 3 is the downstream view taken from Juon and Hager paper[18]. The flow jumps up as is leaves the channel past the flip bucket and spreads out and the water mixes up with air as it falls and lands on the downstream channel.

In the experiment, the geometrical and hydraulic parameters were varied for several test cases. We choose the case of the radius of the circular arc R=0.2m and the elevation difference between the approach channel and the downstream channel h_s of 0.2m. Juon and Hager[18] conducted the flows with and without deflectors on the side walls. The depth of the approach flow h_0 is 5cm or 6cm and the average velocity V_0 is from 2.1m/s to 4.9 m/s and the Froude number Fr based on the flow depth h_0 and the velocity V_0 is varied at 3.0, 5.0 and 7.0.

The results are mostly photographs of the flow taken through the side wall of the flume. In this flow, it is likely that the existence of air does not influence that of water except when air is trapped within water and the side walls.

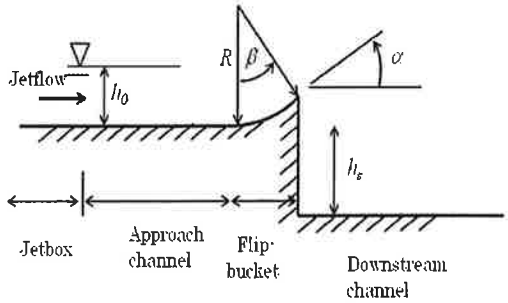


Figure 2. The flip bucket ski jump model with horizontal approach channel.

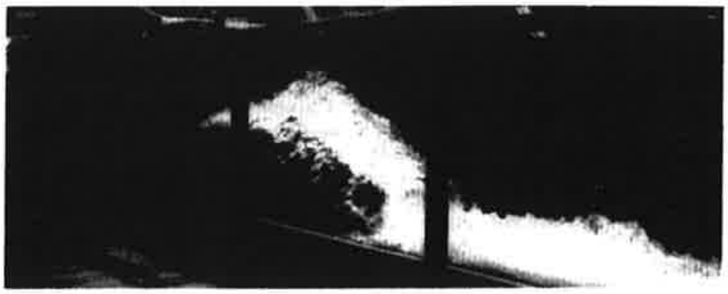


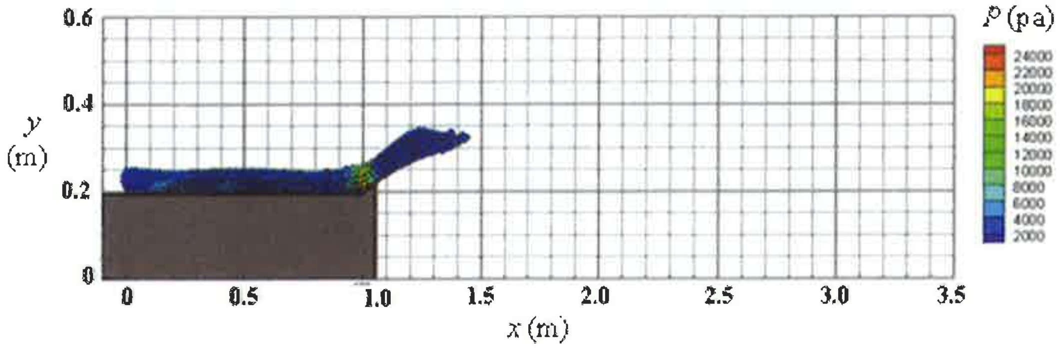
Figure 3. downstream view of the flow with 20 degree flip Bucket deflector, from Juon and Hager[18].

4. CALCULATION RESULTS AND COMPARISON WITH EXPERIMENT

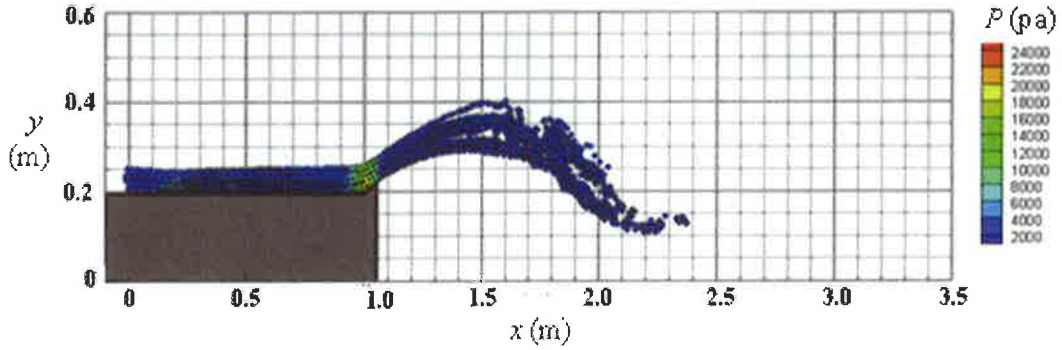
The flow over the flip bucket of the configuration as investigated in the experiment explained in the previous section has been computed by the present SPH based LES method. The calculation was started by placing the rectangular body of water placed in the section corresponding to the jetbox in the experimental setup with assumed initial velocity. Initially the number of particles is 50 x 8 x 5=2000. Then the particles are added in the inflow region with slowly increasing velocity until it reaches the desired velocity and the depth.

Figure 4 shows the sequence of the simulated flow represented by the computed particles in the center span of the channel at 0.25sec, 0.5 sec and 0.75 sec after the start of the flow. The color of the circles representing the particles corresponds to the pressure. It is seen that a smooth initiation of the flow is obtained. The trajectory of the particles after they leave the flip bucket is almost that of the free projectile and parabolic. What is seen is that this parabola is different for the particles with different initial velocity so that the particles with faster initial velocity go higher than those with smaller velocity. This is the reason for the spread of the paths and the nappe is not clearly defined as in more uniform initial flow. After t=0.75sec the first particles have fallen on the floor of the downstream channel the flow start to be steady.

(a) t=0.25sec



(b) t=0.5sec



(c) t=0.75sec

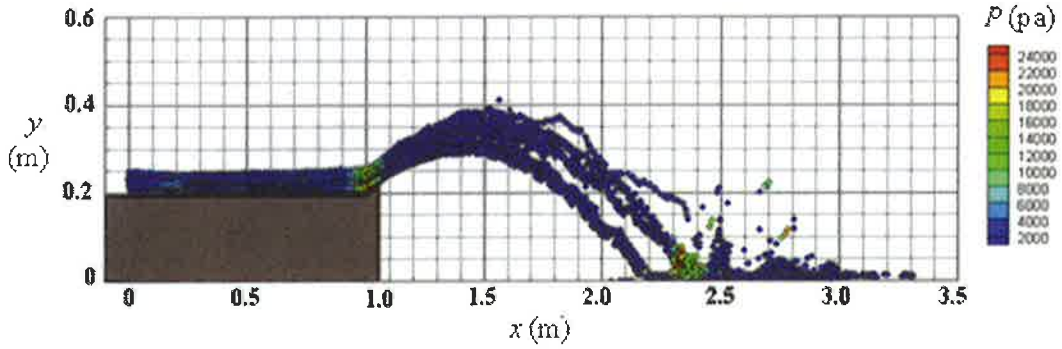


Figure 4. The development of flow for the approach flow of Fr=5.0.

The results of the pressure measurement are not shown but the calculated values at the positions where the water drops on the downstream channel are closely related to the maximum height of the jump and agree well with the measurement.

Next, the results obtained for Froude numbers *Fr*=3.0 and 7.0 are compared with the experimental results and the results of recent calculation by Larese et al[19].using the Particle Finite Element Method (PFEM) method. Figure 5 show the flow trajectories after the steady flow condition is reached. First it is seen from the experimental result that the flow is highly turbulent and the free surface is not clearly defined. Many air bubbles are mixed with the water and the flow path spreads significantly from the initial width of 5cm which is the jet depth.

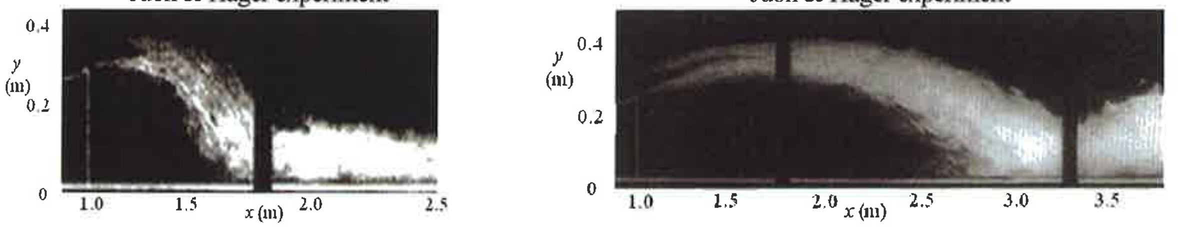
The high resolution (0.5cm) calculation results of PFEM results[19] are shown in the middle row. The flow is indicated in blue color and the average trajectory obtained in the experiments are drawn on the plots. The lines do not indicate the spread but the range of the average trajectory. It is seen that the PFEM results for the low Froude number case shows good agreement with the experiment although the scatter of the flow is very small. For the larger *Fr* case, the discrepancy is even larger the trajectory goes farther downstream than the experiment. The method does not account for the turbulence effects and do not account for the boundary layer in the approach channel, so the velocity at the edge of the flip bucket is very uniform. The free projectile does not spread but rather shrinks due to the lower pressure at the center of the jet.

The present calculation results shown in the bottom row show both the average trajectory and the spread very close to the experiment. The tendency of the spread appears to be two or more, particularly for the higher *Fr* case. Considering the experimental pictures are some kind of time average, the streaks of the jet may not be seen clearly. The agreement with the present calculation is said to be very good.

Some investigation as to the cause of the splitting of the paths was made. The wall stress on the approach channel and the turbulent fluctuation in the approach flow also influence the trajectory. The density re-initialization keeps the uniform pressure but it tends to increase the spread. The effects of the air bubbles will make some influence since some of them are trapped in water and the volume of the bubbles may cause the water jet to spread. More investigation as to the cause of the trajectory spread will have to be made.

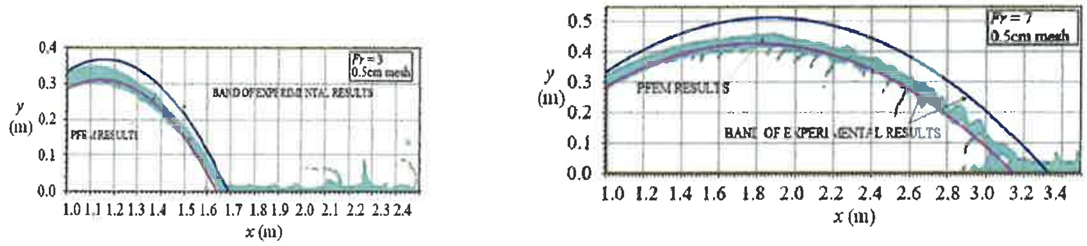
Fr=3.0 Juon & Hager experiment[18]

Fr=7.0 Juon & Hager experiment[18]



PFEM calculation by Larese et al.[19]

PFEM calculation by Larese et al.[19]



Present calculation

Present calculation

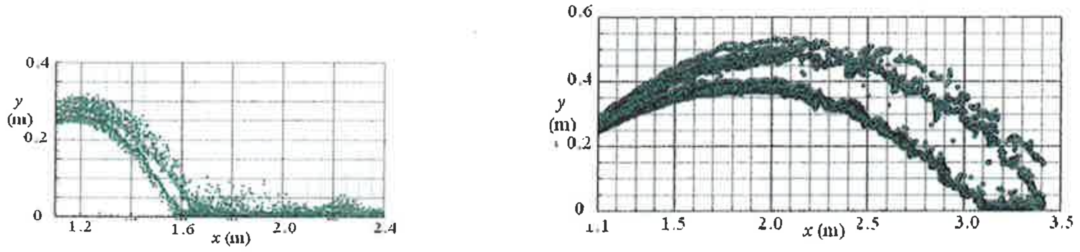


Figure 5. Present simulation results compared with experiment and finite element simulation for *Fr*=3.0 and 7.0.

5. CONCLUSIONS

The basic formulation of the smoothed particle hydrodynamics (SPH) has been improved by implementing the shear stress model based on the wall law and the density re-initialization technique so that high-speed ski jump flows with nappes and air bubbles mixing can be simulated within the framework of large eddy simulation.

The calculation results obtained by the present method have been compared with the experimental results and the recent numerical simulation done by using the Particle Finite Element Method (Larese et al.[19]). The present results are seen to reproduce the average trajectory of the flow jumped up off the flip bucket very well. The degree of the spread of the jet agrees with the experiment as well. These are thought to be the results of the implementation of the wall shear model. The density re-initialization procedure helped maintain the stable pressure distribution. The present calculation using the number of particles with 10,000 or less indicates that one re-initialization every 30 time steps is optimum. The present calculation is done only for the motion of the liquid phase. Whether or not a gas-liquid two-phase approach is needed will have to be investigated, but it is highly likely that the present method equally well in more realistic configuration with the steep spillway or stepped spillway. The number of particles used in the present work is moderate and with higher-power computers or fast GPU processors the number can further be increased for better resolution.

REFERENCES

1) Chanson, H.: Hydraulics of stepped chutes and spillways, Balkema, Rotterdam, (2002).
2) Monaghan, J. J.: Particle methods for hydrodynamics, *Computer Physics Reports*, Vol.3, pp. 71-124, (1985).
3) Violeau, D. and Issa, R. : Numerical modelling of complex turbulent free-surface flows with the SPH method: an overview, *Int. J. Numer. Meth. Fluids*, Vol. 83, pp.277-304, (2007).
4) Lucy, L. B., : Numerical approach to testing the fission hypothesis, *Astronomical Journal*, Vol.82, pp/1013-1024, (1977).
5) Monaghan, J. J. : Simulating free surface flows with SPH, *J. Comput. Phys.* Vol.110. pp.399-406, (1994).
6) Liu, G.R. and Liu, M. B.: *Smoothed Particle Hydrodynamics: A Meshfree Particle Method*, World Scientific Pub. Co. Inc., Singapore, (2003).
7) Monaghan, J.J. : Smoothed particle hydrodynamics and irts diverse applications, Annu. Rev. Fluid MEch. Vol.44,pp.326-346 , (2012).
8) Cummins, S. J. and Rudman, M. : An SPH projection method, *J. Comput. Phys.* Vol.152, pp.584-607, (1999).
9) Shao, S. and Lo, E. Y. M. : Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface, *Advances in Water Resources*, Vol.26, No.7, pp.787-800, (2003).
10) Koshizuka, S. and Oka, Y. : Moving-Particle Semi-Implicit Method for Fragmentation of Incompressible Fluid, *Nuclear Science and Engineering*, Vol.123, pp.421-434, (1996).
11) Nakayama, A. and Hisasue, N. : Applicability of two-phase flow analysis based on particle method for turbulent flows, Memoirs of Construction Engineering Research Institute, Vol.55, pp.23-32, (2014).
12) Nakayama, A., Leong, L.Y. and Kong, W.S.: Simulation of turbulent free surface flow with two-phase SPH method, Journal of Japan Society of Civil Engineers Ser.B1(Hydraulic Engineering) Vol.72, No.4, I_523-I_528, (2016).
13) Kajtar, J. and Monaghan, J. J. : SPH simulation of swimming linked bodies, *J. Comput. Phys.*, Vol. 227, pp.8568-8587, (2008).
14) Shepard, D. A two-dimensional interpolation function for irregularly-spaced data, Proc. 23rd ACM National Conference, pp.517-524, (1968).
15) Vorobyev, A.: A smoothed particle hydrodynamics method for the simulation of centralized sloching experiments, KIT Scientific Publishing Karlsruhe, (2013)
16) Colagrossi, A. and Landrini, M. : Numerical simulation of interfacial flows by smoothed particle hydrodynamics, *J. Comput. Phys.* Vol.191, pp.448-475, (2003).
17) Gotoh, H., Sibahara, T. and Sakai, T. : Sub-particle-scale turbulence model for the MPS method – Lagrangian flow model for hydraulic engineering, *Computational Fluid Dynamics Journal* Vol. 9, pp.339-347, (2001).
18) Juon, R. and Hagger, W. H.: Flip bucket without and with deflectors, J. Hydr. Engrg., ASCE, Vol.126, No.11, pp.837-845, (2000).
19) Larese, A., Rossi, R., Onate, E. and Idelsohn, S.R.: Validation of the particle finite element method (PFEM) for simulation of free surface flows, *International Journal for computer-aided engineering and software,* Vol.25, No.4, pp.385-425, (2008).

AUTHORS

Akihiko Nakayama	Ph.D., Fluid Mechanics, Hydraulics
Leong Lap Yan	Student, Department of Environmental Engineering, Universiti Tunku Abdul Rahman
Kong Wei Song	Student, Department of Environmental Engineering, Universiti Tunku Abdul Rahman