Simulation of Dam Break Using Modified Incompressible Smoothed Particles Hydrodynamics (MPM-I-SPH)

Dr.Mahmoud Reza Mollaie Nia, Dr.Rasool Ghobadian, Satar Ahmadyan, Hamzeh Ebrahimnejadian

Abstract— Generally, hydrodynamic problems can be analyzed using the three different views of Lagrange, Euler and a combination of Lagrange and Euler views. Choosing any of these methods depends on the nature of the problem and its characteristics. Using the existing numerical methods, which are based on network production, for the analysis of such problems can be time consuming and problematic, also numerical propagation error in Euler's methods which may arise through discretization of transfer terms in Navier-Stokes equations, could have a negative impact on the accuracy of results. Considering these problems, it seems that only Lagrangian numerical methods without grid are suitable for the analysis of hydrodynamic problems. This study aims at simulation of dam break using MPM-I-SPH. This approach has been primarily based on integral interpolation theory. In this method, various fluid parameters such as density, velocity and pressure are specified, and the value of a dependent variable is calculated by summing over neighboring particles. Differential equations can be converted to integral equations using an interpolation function. In this paper, the governing equations for free surface flows are described based on the Lagrangian SPH method . Gradient and Laplacian terms modeling, execution of boundary conditions and the predictor - corrector standard method for the solution of dam break equations were presented The good accordance between modeling results and experimental results proved the capability of the resultant numerical model for simulation of complex phenomena.

Index Terms— Lagrange, Euler, Non-grid, Smoothed Particle Hydrodynamics Method (SPH), Dam Break

1 INTRODUCTION

Simulation of free surface flow has always been accompanied with complications such as nonlinearity of equations and difficulty of boundary conditions execution, and on the other hand, the problems with non-geometrical and highly variable computational domains, has intensified the difficulties. In conventional numerical methods such as finite differences 1and finite volume2 methods, is implemented through gridding of flow field which does not cause any problem in 2D simple geometry problems but in 3D problems and problems with complex geometry, gridding is time consuming and increases computation costs and their accuracy is not necessarily high. Obviously, if the problem has a moving boundary, in each step a new gridding is needed, therefore, in recent years, efforts have been made to develop numerical methods in which simulation process is implemented without gridding.
These are referred to as non-grid numerical methods. In this methods, no gridding is needed for discretization of the governing equations and physical interpretation of the flow, which allows for the analysis of a large number of problems. Furthermore, Lagrangian approach can be easily applied in non-grid methods.

2. Smoothed Particle Hydrodynamics Method (SPH)

2.1 Review Stage

SPH is a purely Lagrangian method which does not require any gridding. This method was first used for solving astrophysical problems (Lucy, 1977). Monaghan is among the pioneers of SPH method who used this method for the analysis of free-surface flows, in addition to using it for astrophysical problems, and the obtained results were satisfactory (Monaghan, 2000). More interesting applications of moving particle method have been observed in the field of metal forming (Rajendran and Narasimhan, 2001) and the phenomenon of explosion on underwater structures (Liu and Liu, 2003).
Shao introduced I-SPH which stands for Incompressible Smoothed Particle Hydrodynamics for simulation of different free-surface problems. Using this method, he was able to simulate dam break problems with an acceptable degree of accuracy. In I-SPH method, the fluid is assumed as incompressible and with this provision and a predictive-corrective method, Navier-Stokes equations are resolved (Shao, 2003)
Also, smoothed particle hydrodynamics method has been applied in fluid dynamics problems (Dalrymple and Rogers, 2006), the interaction of liquid and solid phases such as water and rubber control valve (Antoci et all, 2007), and simulations of dolphin kick swimming (a swimming stroke like crawl, etc.) (Cohen et all, 2011).
In fact SPH is a weighted averaging method for estimating a

International Journal of Scientific & Engineering Research Volume 5, Issue 1, january-2014
ISSN 2229-5518

parameter, thus, for estimating a parameter at a certain point it is assumed that neighboring points, each one according to an interpolation function, contribute to the estimation of the value of this parameter at the desired point.

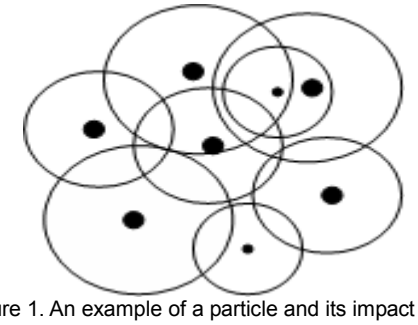

Figure 1. An example of a particle and its impact zone

The integral form of each function is as follows:

$$A(r_a) = \int A(r')W(r - r', h)dr' \quad (1)$$

in which r is position vector of influence point, w is the weighting or kernel function and h is the smoothing length that adjust the support domain (influence domain).

In this method, continuous integration changes to summation in discrete points:

$$A(r_a) \approx \sum_b m_b \frac{A_b}{\rho_b} | (r_a - r_b), h | \quad (2)$$

in which a is the central particle, b representing the neighboring particles, m mass, ρ density and. $W_{ab} = W(|r_a - r_b|, h)$

With derivation of equation (2) and considering that in this method the differential operator acts only on the interpolation function, derivative of this function in point a equals to:

$$\nabla A(r_a) = \sum_b m_b \frac{A_b}{\rho_b} \nabla_a W_{ab} \quad (3)$$

$\nabla_a W$ Gradient is relative to central particle.

3. Governing equations of fluid flow

Flow equations are Navier-Stokes equations which include the 3 equations of conservation of mass, conservation of momentum and conservation of energy. In fluid flow the dominant equations are conservation of mass and conservation of momentum equations.

$$\frac{1}{\rho}\frac{D\rho}{Dt} + \nabla.\vec{u} = 0 \quad (4)$$

$$\frac{D\vec{u}}{Dt} = -\frac{1}{\rho}\Delta p + \vec{g} + \frac{1}{\rho}\Delta.\overrightarrow{\tau} \quad (5)$$

Equations (4) and (5) express the general form of the mass conservation and momentum conservation equations respectively. In this equations ρ is density, $\vec{u}$ velocity vector, p pressure, $\overrightarrow{\tau}$ stress tensor, $\vec{g}$ acceleration of gravity and t time.

4. Smoothing Distance

The smoothing distance (h) in SPH method expresses the impact zone around a central particle which is interacting with other particles in this field, in other words, the impact zone of a particle is a circle to center of the particle and with a 2h radius. According to equation (6), smoothing distance is in related to the initial distance of neighboring particles:

$$h = \kappa d_0 \quad (6)$$

κ is the coefficient of smoothing distance which should be optimized for each problem but its value is normally between 1 and 2.

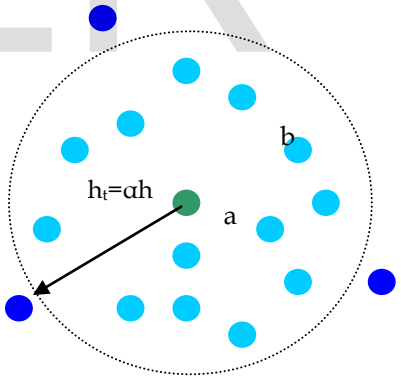

Figure 2. The interaction zone of central particle a with neighboring particles

5. The Interpolation Function

Interpolation functions play a key role in SPH method, since these functions express how to approximate the value of a quantity as well as the size of the impact zone for each particle. Interpolation functions which are used in this method require special conditions (Liu, 2003).

a: If the impact zone of the central particle a be represented by Ω I and the outer area by 0, we have:

$$W(r, h) > 0$$

International Journal of Scientific & Engineering Research Volume 5, Issue 1, january-2014
ISSN 2229-5518

in field ΩI

in field $W(r,h)=0$ Ω0

Under this condition, each point interact with a limited number of neighboring points and based on this, the quantity of a point can be estimated locally.

b:Stability Condition

$$\int_{\Omega_i} W(r,h).d\Omega = 1$$

c: Interpolation functions need to be steadily descending over ΩI area in a way that they are maximum at the center and zero at the boundaries of this region.
The most common form of interpolation functions are polynomial functions of degree 2 and 3. Interpolation function of degree 3 which has been proposed by Monaghan is expressed as equation (7) (Monaghan, 1992).

$$W(r,h)=\frac{10}{7\pi h^2}\left(1-\frac{3}{2}q^2+\frac{3}{4}q^3\right) \quad q<1$$

$$W(r,h)=\frac{10}{28\pi h^2}(2-q)^3 \quad 1<q<2 \qquad (7)$$

$$W(r,h)=0$$

in which $q=\frac{r}{h}$ and r is the distance between particles.

The smoothing distance (h) determines the impact radius around particle a. It should be noted that, in this study, this function has been used as interpolation function.

6. Boundary Conditions

6.1 Solid Boundaries

Accurate simulation of solid boundaries has an important role in SPH method. In recent years, different method have been proposed. In one method, the wall surface is covered with fixed particles so that applying a virtual return force on fluid particles, prevents them from leaving the wall. This method is well illustrated in Figure 3.

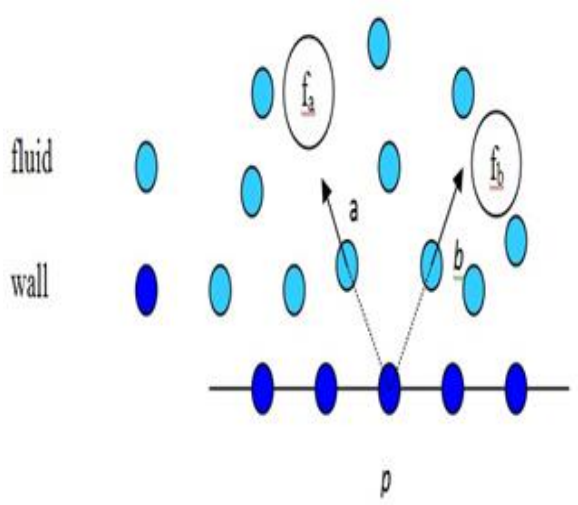

Figure 3. The impact of boundary particles on internal particles

6.2 Free Surface

Particles satisfying the following requirements are known as free surface particle and zero-pressure condition is applied for them.

$$\rho_* < \beta\rho_0 \qquad (8)$$

In this equation, ρ∗ is the particle density in prediction step, ρ0 the fluid's constant density and β the free surface parameter which is within the range of 0.8 to 1. This value is effective in the optimization of results.
Given that there is no particle across the free boundary, equation (9) cannot be used directly to calculate the pressure gradient for free-surface particle and a number of virtual particles are needed to be considered. In Figure 4, i is a particle within the fluid, s a free surface particle and m, a virtual particle. The virtual particle m and particle i are symmetric with respect to particle s (Grilli, 1999).

$$\nabla.(\frac{1}{\rho}\nabla P)_a = \sum_b m_b \frac{8}{(\rho_a+\rho_b)^2}\frac{P_{ab}\,\vec{r}_{ab}.\nabla_a W_{ab}}{|\vec{r}_{ab}|^2+\eta 2} \qquad (9)$$

International Journal of Scientific & Engineering Research Volume 5, Issue 1, january-2014
ISSN 2229-5518

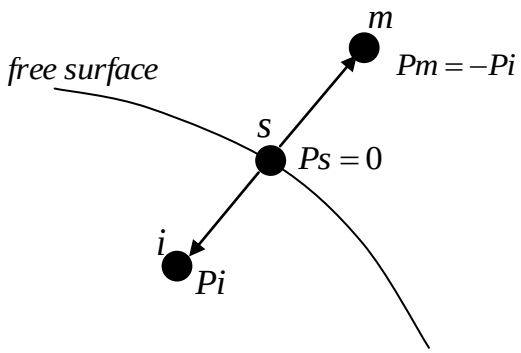

Figure 4. The position of a fluid virtual particle across the free-surface border

Thus, particles i and m are neighboring particles to particle s and their pressure gradient can be calculated as follows:

$$(\frac{1}{\rho}\nabla P)_{si} = m\ (\frac{P_s}{\rho_s^{\ 2}} + \frac{P_i}{\rho_i^{\ 2}})\nabla_a W + m(\frac{P_s}{\rho_s^{\ 2}} + \frac{P_m}{\rho_m^{\ 2}})\nabla_a W \quad \textbf{(10)}$$

As is shown in Figure 4, the pressure of virtual particle m is symmetric to internal particle pressure i, in which case, with the interpolation between the pressure of particles m and i, the pressure of free-surface particle will be zero.
So we can write:

$$P_m = -P_i$$
$$P_s = 0$$
$$\nabla_s Wsm = -\nabla_s Wsi \quad \textbf{(11)}$$

By applying the conditions of equation (11) in equation (10) it can be easily realized that the actual value of the pressure gradient of a free-surface particle is twice the value calculated for internal particles and free-surface particles can be moved correctly by applying these conditions.

7. Solving Navier-Stokes equations for incompressible fluid using M-I-SPH

Shobeiri and Ataei Ashtiani by providing a new form for the source term of the pressure Poisson equation and also modifying part of I-SPH method formulation by which the incompressibility condition of free-surface particles is satisfied more efficiently as well as proposing a new type of source term for the pressure Poisson equation, increased the accuracy and stability of I-SPH method. They used this method for simulation several problems, including the expanding an incompressible water drop under the influence of initial velocity, the movement of a solitary wave on constant depth, wave breaking on the ramp, dam break and shock wave and obtained results were constant and satisfactory, thus they introduced MI-SPH method as the modified version of I-SPH for solving various free-surface problems.

7.1 Applying the initial conditions, velocity prediction and the position of particles in the next time-step

The SPH method involves the prediction and correction stages for incompressible fluids. After applying the initial conditions, in the prediction stage, regardless of the pressure term in the momentum equation and considering the following equations, the position and velocity of particles in the next time step is predicted.

$$\Delta\vec{u}_* = (\vec{g} + \frac{\mu}{\rho}\nabla^2\vec{u})\Delta t \quad \textbf{(12)}$$

$$\vec{u}_* = \vec{u}_t + \Delta\vec{u}_* \quad \textbf{(13)}$$

$$\vec{r}_* = \vec{r}_t + \vec{u}_*\Delta t \quad \textbf{(14)}$$

$\vec{u}_t$ and $\vec{r}_t$ are respectively velocity and the position of the fluid particles at the current time step, $\Delta\vec{u}_*$ the variation of the fluid velocity in the prediction step, $\vec{u}_*$ and $\vec{r}_*$ are respectively the predicted velocity and th position of fluid particles in the next time step and Δt is the value of time step.

7.2 Calculation of particle density in the prediction stage

The density of particles is calculated according to the predicted position of particles (r_*) which is indicated by ρ_*. The calculated density of particles varies with constant density ρ_0 which is due to not considering the pressure term in the momentum equation.

7.3 Particles Pressure Estimation

The equation, by which the pressure of fluid particles is calculated and can satisfy the incompressiblity condition of fluid, is the mass conservation equation which is expressed as follows.

$$\frac{1}{\rho_0}\frac{\rho_0 - \rho_*}{\Delta t} + \nabla.(\Delta\vec{u}_{**}) = 0 \quad \textbf{(15)}$$

In this equation, ρ_* is particles density in the prediction step, ρ_0 the constant density of particles and $\Delta\vec{u}_{**}$ the corrected velocity in the correction step which is derived from momentum equation.

International Journal of Scientific & Engineering Research Volume 5, Issue 1, january-2014
ISSN 2229-5518

$$\Delta\vec{u}_{**} = \frac{-1}{\rho_*}\nabla P_{t+1}\Delta t \quad (16)$$

The right hand of equation (16) is solved using equation (15). The pressure Poisson equation is obtained by combining equations (15) and (16), that is, the combination of the continuity and momentum equations, and by solving this equation, the pressure of the fluid particles can be calculated.

$$\nabla.(\frac{1}{\rho_*}\nabla P_{t+1}) = \frac{\rho_0 - \rho_*}{\rho_0 \Delta t^2} \quad (17)$$

Considering the error arising from the discretization of governing equations and also the error arising from solving the system of linear equations arising and other errors, satisfying the exact incompressibility condition is virtually impossible. To satisfy the incompressibility conditions, in this section the source term is calculated differently which increases the accuracy and stability of the I-SPH method. Shao using a finite difference approximation presented a Laplacian equation, which prevents the instability (Shao, 2004):

$$\nabla.(\frac{1}{\rho}\nabla P)_a = \sum_b m_b \frac{8}{(\rho_a + \rho_b)^2}\frac{P_{ab}\vec{r}_{ab}.\nabla_a W_{ab}}{|\vec{r}_{ab}|^2 + \eta 2} \quad (18)$$

in which $P_{ab} = P_a - P_b$, $r_{ab} = r_a - r_b$ and η are a small and equal to 0.1h.

7.4 Correcting the predicted velocity

In this step, the particles calculated considering the calculated pressure of particles in the next time step, $\Delta\vec{u}_{**}$ which is in fact the correction value of the predicted velocity, can be added to predicted velocity and the particles velocity at the next time step is obtained.

$$\vec{u}_{t+1} = \vec{u}_* + \Delta\vec{u}_{**} \quad (19)$$

7.5 Particles movement

In this is, which is in fact the last step in the algorithm, given the velocity of particles in the next time step and their current current velocity, the correct position of particles in the next time step is calculated in equation (20), as follows:

$$\vec{r}_{t+1} = \vec{r}_t + \frac{\vec{u}_{t+1} + \vec{u}_t}{2}\Delta t \quad (20)$$

in which $\vec{r}_{t+1}$ is the position of particles in the t+1 time step. In this stage, one time step is finished. Considering the obtained data, the subsequent time steps will be passed according to the same algorithm until the desired time.

8. Simulation of dam break as the initial test for MPM-I-SPH method

One modification which seems essential for M-I-SPH program is converting the program into a multi-phase state. Because this program, with a change in fluid particle properties, such as changing the mass of only one particle, exhibit unstable solutions. The linearity of MI-SPH numerical model was the main cause of this problem and to solve this problem, in all the governing equations, the changing parameter was inserted into equations as a matrix. Thus, in all the governing equations, mass and density parameters have been rewritten as matrices and the stability of this method increased considerably.

Also, this problem has been simulated using other numerical methods. As previously noted, in MPM-I-SPH method, the changing parameters need to be inserted into equations as matrices. Therefore, in the computer program which is written for the consideration of particles with different mass and density in the fluid, as is illustrated in equations (21) and (22), a c parameter has been defined, and first the value of this parameter is determined in the program, and then the mass and density of the desired particles is defined as follows:

$$m_s = c * m_f \quad (21)$$

$$\rho_s = c * \rho_f \quad (22)$$

in which m_s and ρ_s are the mass and density of sediment particles and m_f and ρ_f mass and density of fluid particles. As it is evident in the equations (21) and (22), if we set the value of parameter c as 1, it will remain in form of a matrix and at the same time it can be used to simulate one-phase free-surface problems. Therefore, simulation of dam break, which is one of the most common problems for the validation of numerical methods, have been used for the validation of MPM-I-SPH the method.

8.1 Validation of Dam Break Simulation

The geometrical conditions of a typical dam problem is shown in Figure 5, in which a water column with the dimensions of 6.14cm width and 2.29cm height is kept using a barrier at the time of t = 0s. This barrier is removed quickly and by removing the barrier, a wave is formed which moves toward the downstream wall and after collision with this wall it climbs the wall and then moves back to the left wall and this continues until the water reaches equilibrium.

International Journal of Scientific & Engineering Research Volume 5, Issue 1, january-2014
ISSN 2229-5518

Computational parameters used in this section are shown in Table 1.

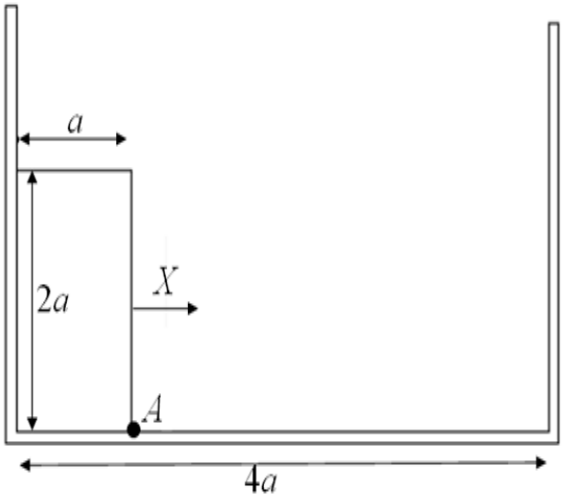

Figure 5. The initial requirements for dam break simulation

Table 1. Computational parameters used in the simulation of dam break

1	The time step is controlled by the Courant number 0.1= $\frac{\Delta t V_{max}}{l_0}$
2	extreme time step: 0.001 s>??
3	particles diameter and thus the initial distance between particles: l_0=0.008m
4	The number of solid particles is 648 and the number of fluid particles 452.
5	free-surface parameter β=0.95
6	water density: ρ_0=1000 $\frac{kg}{m^3}$
7	C= $\frac{\rho_{max}}{\rho_0}$ =1
8	smoothing distance coefficient: k=1.2 $\frac{h}{l_0}$ = or in other owrds h=1.2l_0

In Figure 6 and 7 the simulation results of water surface profile using MPM-I-SPH method at different times are compared with experimental results. As it can be seen in these figures, the results of this method are in good agreement with experimental results that suggest the initial validity of the desired computer code.

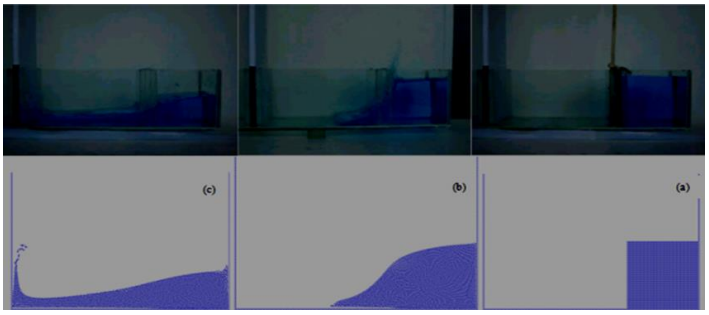

Figure 6. Comparison of the results of dam break simulation using MPM-I-SPH method with experimental results at 0, 0.2, 0.4 seconds

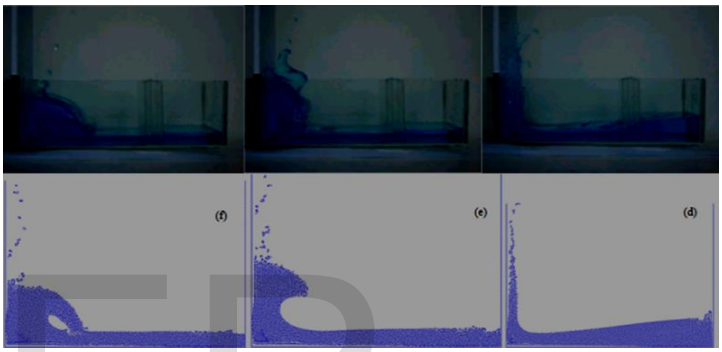

Figure 7. Comparison of the results of dam break simulation using MPM-I-SPH method with experimental results at 0.6, 0.8, 1 seconds

Now that the validity of MPM-I-SPH method for solving one-phase problems has been proved, it is suitable to show the ability of this method in simulation of multi-phase problems, because the main difference between this method and M-I-SPH is in solving multi-phase problems. So in the first step, as shown in Figure 8, in the dam break problem, the value of parameter c is set to be greater than one and the results of c = 1 and c = 2.65 are compared and the results are analyzed.

As is shown in Figure 8 in case of c = 1, a number of particles which are separated from the rest of particles, are moving in with other particles harmoniously, but in case of c = 2.65, these particles create a slight disharmony in the movement of other particles. Also, the amount of progress and climbing of these particles in the latter case is less than in the first case which is logical and acceptable as the mass and density of particles in the latter case is more than the former case.

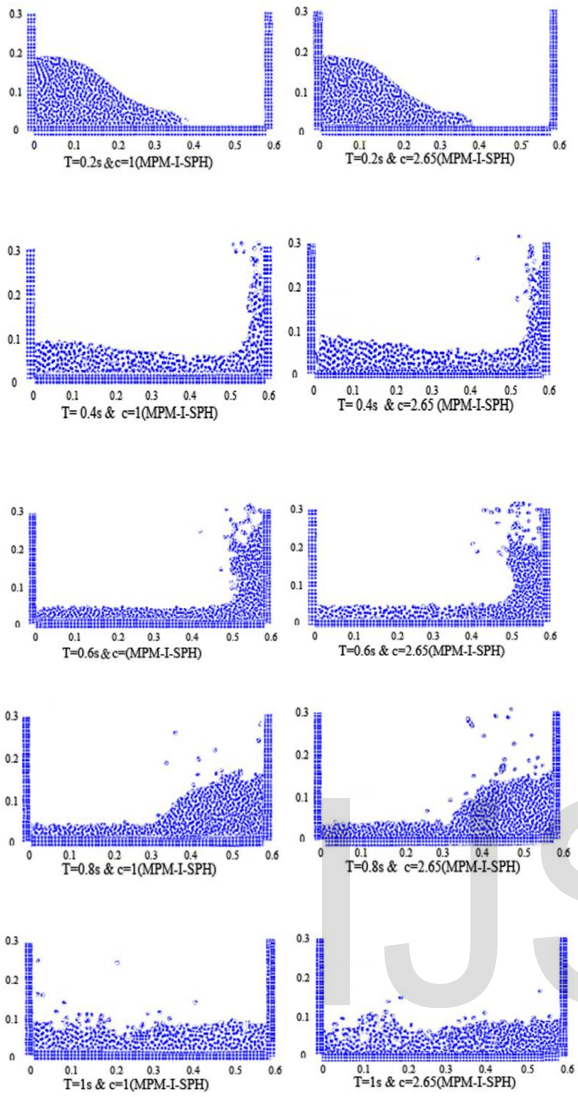

Figure 8. The results of dam break simulation using MPM-I-SPH method in different time steps

This problem is similar to the situation in which the simulation of dam break is performed with water containing sediment particles. Now, considering the initial validation and the obtained stable and reasonable results, MPM-I-SPH method that is multi-phase M-I-SPH method, is introduced for solving various multi-phase problems.

CONCLUSION

In this method, in addition to fluid particles, sediment particles have been brought into simulation. Initially dam break simulation was conducted with MPM-I-SPH in which c = 1 and the results were compared to experimental results and the good agreement of simulation results with experimental results revealed that it has a high capacity in simulation of dam break and similar problems. Then, the simulation of dam break was conducted with c = 2.62 and sediment particles in addition to fluid particles and the results of c = 1 and c = 2.65 were compared. The analysis of the obtained results indicated that MPM-I-SPH method has a high capacity in the analysis and simulation of complex free-surface flows and similar problems.

REFERENCES

[1] Lucy, L.B. 1977 "A numerical approach to the testing of the fission hypothesis." Astron J. Vol 82(12), pp:1013-1024.

[2] Monaghan, J.J. 2000." SPH without tensile instability." J Comput. Phys, Vol 159, pp: 290- 311.

[3] Rajendran, R. and K. Narasimhan.2001. Performance evaluation of HSLA steel subjected to underwater explosion. Journal of Materials Engineering and Performance, 10: p: 66-74.

[4] Liu, G.R. and Liu .M.B. 2003 Smoothed particle hydrodynamics a meshfree particle method.: World Scientific Publishing.

[5] Shao, S.D. and Lo, E.Y.M. 2003 "Incompressible SPH Method for Simulating Newtonian and Non-Newtonian Flows with a Free Surface", Adv. Water Res., Vol 26(7), PP: 787–800.

[6] Dalrymple, R.A. and Rogers. B.D. 2006 Numerical modeling of water waves with the SPH method. Coastal Engineering, 53: p:141–147.

[7] Antoci, C., Gallati. M. and Sibilla.S. 2007 Numerical simulation of fluid-structure interaction by sph. Computers & Structures, 85, p: 879-890.

[8] Cohen, R.C.Z., Cleary, P.W. and Mason, B.R. 2011.Simulations of dolphin kick swimming using smoothed particle hydrodynamics, Human Movement Science.

[9] Monaghan, J.J. 1992 "Smoothed particle hydrodynamics", Annu RevAstron Astrophys, Vol 30, PP: 543-574.

[10] Grilli, S.T., Watts, P. 1999 "Modeling of waves generated by a moving submerged body.Applications to underwater landslides". J.Engineering Analysis with Boundary Elements, Vol(23), PP: 645–656.

[11] Shao, S. and Gotoh, H. 2004 Simulating coupled motion of progressive wave and floating curtain wall by SPH-LES model. Coastal Engineering Journal, Vol. 46:2 171-202.