A self-developed algorithm based on the Moving Particle Semi-implicit method for investigating dam break

Zhen Zhang, Hohai University, zhenzhang515@hhu.edu.cn
Aifeng Tao, Hohai University, aftao@hhu.edu.cn

INTRODUCTION

Cases of dam failures are seen almost every year globally, which is a topic of great importance for hydraulic researchers because in reality this catastrophic phenomenon can result in immense economic damage and loss of lives (Esmaeeli Mohsenabadi et al., 2023; Hooshyaripor and Tahershamsi, 2015; Larocque et al., 2013). However, it is difficult to explore the dam-break flow characteristics because of their non-linearity, quickly changing unstable flow, and even breaking waves(Hooshyaripor et al., 2014). The meshless methodology demonstrates its superiority to the conventional mesh method in simulating the splashing and rolling (Ye et al., 2020). As a mesh-free Lagrangian particle method, the Moving Particle Semi-implicit (MPS) method can easily handle with simulating problems with complex flow. Instead of calculating the state equation in a weakly compressible state as in the weakly-compressible Smoothed Particle Hydrodynamics (SPH) method, MPS methods solve the pressure Poisson equation (PPE) implicitly. Therefore, the MPS method can simulate the incompressible flow more accurately (Wu et al., 2023).

ALGOTITHM BASED ON MPS

The theory of the self-developed algorithm is based on the MPS method by Koshizuka (1995). The continuity and Navier-Stokes equations are applied for the incompressible viscous flow as follows:

$$\frac{d\rho}{dt} + \rho\nabla \cdot v = 0 \tag{1}$$

$$\frac{dv}{dt} = f - \frac{1}{\rho}\nabla P + \upsilon\nabla^2 v \tag{2}$$

where, ρ is the density of the fluid, v is the velocity vector of the flow field, f is the external force, P is the pressure and υ is the kinematic viscosity coefficient.

In the MPS method, the interaction between particles is achieved by means of a kernel function.Every particle interacts with each other in the domain of control radius r_e. In this paper, the widely used kernel function shown in equation (3) is used.

$$W(r) = \begin{cases} \frac{r_e}{r} - 1 & 0 \le r < r_e \\ 0 & r \ge r_e \end{cases} \tag{3}$$

where, r_e is the radius of action of the particle and r is the distance between two adjacent particles.

The Navier-Stokes equation is solved in two sections using the procedure, which introduces the intermediate velocity. Additionally, the computation process of the numerical method is shown as follows, and Figure 1 displays the relevant flow chart.

(1) The particle velocity is explicitly changed to obtain the intermediate velocity using the forces of viscosity and mass as source terms, and the particle is then moved to the intermediate position in accordance with the intermediate velocity.

(2) Calculating the density of the particle at its intermediate location. The particle search method is used to search for the neighbour particle once more, and the definition of particle density is used to calculate the intermediate particle density.

(3) Determining the free surface. The pressure of the free-surface particles will be replaced with the value of 0 in the following Poisson equation solution.

(4) Calculating the pressure at the following instant by solving the Poisson equation.

(5) The intermediate velocity v^* is adjusted based on the pressure gradient to determine the velocity at the subsequent instant $n+1$, and the intermediate displacement r^* is then updated based on the velocity and intermediate position.

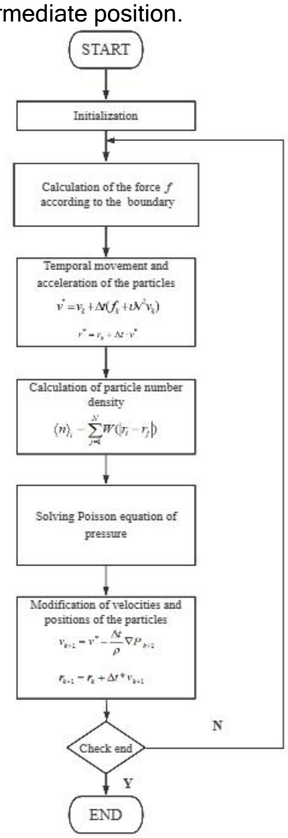

Figure 1- Flow chart

VALIDATION AND INVESTIGATION

There are walls at the bottom and on the left and right of the dam break model shown in Figure 2. The model has a height of one meter and a length of 1.61 meters. The first water block is 0.6 meters high and long. P1 was positioned at a height of 0.003 meters on the right. The experimental results conducted by Lobovský et al.(2014) are compared with the numerical results.

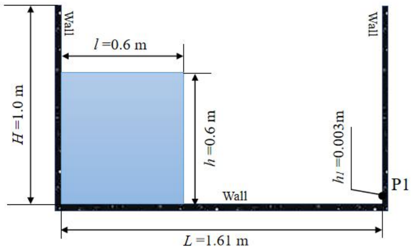

Figure 2 - Model of the dam break

The pressure at position P1 is compared to the experiment and the numerical technique in Figure 3. The results are dimensionless processing. The number starts at 0 because the fluid flows to the point P1 where time is needed. The curve produced by the numerical approach matches the curve of the experiment.

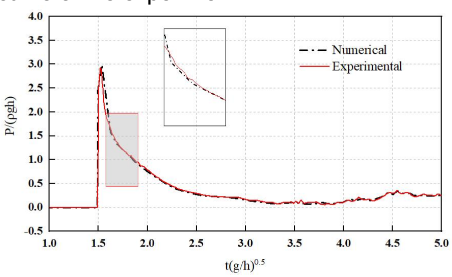

Figure 3 - Comparison of the results in numerical method and experiment

The comparison between the experiment's results and those obtained by the self-developed solver is displayed in Figure 4. This image illustrates the dam break occurrence for time intervals of 0.455 s and is in good agreement with the experiment. After that, the dam fluid starts to ascend the wall. The liquid breaking and splashing event is evident in this picture.

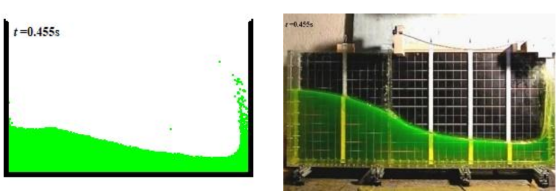

Figure 4 - Comparison of the Phenomenon

The model of the dam break with barrier is shown in Fig. 5 with the size of $L \times 2L$ and the length of the tank is $4L$. There is a fixed barrier of size $h \times 2h$ at the left $2L$ of the container. It defines that L=0.144m and h=0.024m.

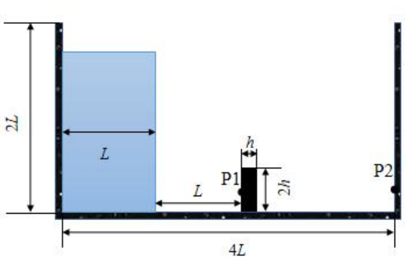

Figure 5 - Model of the dam break with barrier

CONCLUSIONS AND PROSPECT

In this paper, an autonomous algorithm based on the Moving Particle Semi-implicit method (MPS) is developed to investigate the issues about large deformation of free surfaces. Initially, the dam-breaks were modeled using the self-developed algorithm and the experimental findings have been adopted to validate the solver and demonstrate satisfactory agreement. Take the purpose of comprehending the dynamic process of the dam break and how to lessen the risks caused by it, the issues about variations in flow depth with time, impact on the wall, and the formation and propagation of negative and positive waves in the presence of obstacles during dam-break events are then looked into and analyzed. In addition, the phenomena of wave breaking and liquid splashing were recorded and observed during the simulation of dam break to provide reference in the issues about beach erosion etc.In order to more thoroughly examine the challenges brought on by natural disasters, the algorithm will be applied to the other domains addressing large deformation issues.

REFERENCES

Kheirkhah Gildeh H（2023): CFD modelling of initial stages of dam-break flow. Canadian Journal of Civil Engineering. vol. 50, pp. 838-852.

Hooshyaripor F, Tahershamsi A (2015): Effect of reservoir side slopes on dam-break flood waves. Engineering Applications of Computational Fluid Mechanics, vol.9, pp.458-468.

Hooshyaripor F, Tahershamsi A, Golian S (2014): Application of copula method and neural networks for predicting peak outflow from breached embankments. Journal of Hydro-environment Research, vol. 8, pp. 292-303.

Koshizuka S (1995): A particle method for incompressible viscous flow with fluid fragmentation.

Larocque L, Imran J, Chaudhry M (2013): Experimental and Numerical Investigations of Two-Dimensional Dam-Break Flows. Journal of Hydraulic Engineering, vol. 139, pp. 569-579.

Lobovský L, Botia-Vera E, Castellana F, Mas-Soler J, Souto-Iglesias A (2014): Experimental investigation of dynamic pressure loads during dam break. Journal of Fluids and Structures, vlo. 48, pp. 407-434.

Wu J, Zhang G, Sun Z, Yan H, Zhou B (2023): An improved MPS method for simulating multiphase flows characterized by high-density ratios and violent deformation of interface. Computer Methods in Applied Mechanics and Engineering, vol. 412, pp.116103.

Ye Y, Xu T, Zhu D Z (2020): Numerical analysis of dam-break waves propagating over dry and wet beds by the mesh-free method, Ocean Engineering, vol. 217, pp. 107969.