Evolution of Free Surface of 3-D shematic Dam Break using Smoothed Particle Hydromechanics Method

Mohamed Roudane, Sid Ahmed Gouri, Sofiane Khelladi, Abdelkader Krimi and Abderrahmane Mahallem

***Abstract*—**In this paper, a modeling of free surface fluid flow is performed using Smoothed Particle Hydro¬dynamics method. A simulation of water dambreak benchmark with obstacle is carried out. To highlight and validate our developed numerical model, a comparison between two different methods for free surface fluid flow simulation as well as with experiment results is presented. The first method is based on finite volume method (FVM) which is a mesh-dependent method. In their study, W. C. Moon and al attempt to validate a multiphase model with a third-order accurate scheme for two-dimensional flows with free surface based on the Volume/Surface Integrated Average Multi-Moment Method (VSIAM3). The second one is based on smoothed particle hydrodynamics method which is a meshless method. In this method, we propose a consistent weakly compressible SPH model (WCSPH) to simulate Newtonian fluids.

***Keywords*—** Free surface flow, Smoothed particle hydrodynamics method, Incompressible fluid.

I. Introduction

The numerical simulation of free surface fluid flows is a fascinating topic for researchers regarding its importance to simulate several natural phenomena like tsunami flow. Many works in the literature studied the behavior of free surface flow using different methods for numerical simulation. the most classical problem is the dam-break which we consider in this work. Our simulation results are compared with a previously re¬alized simulations published by W. C. Moon and al(1).

In their work, they tested The performance of their numerical model by simulating a violent free-surface flow (dam-break problem with an obstacle). They used the Tangent of Hyperbola for Interface Capturing (THINC) scheme in the numerical model to advect the volume of fluid (VOF) function. To increase accuracy of the numerical model, they solved The advection term in the momentum equations using the third-order constrained interpolation profile-conservative semi-Lagrangian (CIP-CSL3) scheme. their Results showed Satisfactory agreements between the numerical and the previously published experimental results in term of the development of free surface profile. They employed a Single phase model as a tool for flow analysis due to its simplicity, However, they still had a challenging issue to implement a higher order scheme to solve the advection term in single phase flow model due to the boundary conditions at the free surface. Herefore, we propose suitable method to produce accurate results with optimum computational cost.

Smoothed Particle Hydrodynamics (SPH) is one of the most efficient and robust meshfree particle methods. It is a Lagrangian gridless method developed initially to simulate astrophysical phenomena, originally formulated by Lucy (2) and Gingold and Monaghan (3) and since then, it has been known for a large number of applications, especially for fluid flow simulations. Contrary to the grid-based method, the SPH method can handle free surface and interfacial fluid flow simulation including large deformations naturally and without the need for any specific treatment. The basic idea of the meshfree methods is to discretize the continuum into a set of nodes without presence of any connectivity between these nodes. This property makes treatment of large deformation problems, and representation of the free and moving interfaces an easier task while keeping a reasonable computational effort. When the nodes represent a massive element (particle) of the material domain and carrying its physical properties, the methods are so-called "meshfree particle methods (MPMs)" (4). This kind of methods follows in general a Lagrangian approach. The highlight of the SPH method is related to its ability to represent numerical approximations for functions and their spatial derivatives without need to any topological connectivity between nodes (mesh).

II. Smothed Particle Hydrodynamics Method

A. Governing Equation

A weakly compressible viscous fluid flow in isother¬mal conditions is considered in this paper. Thus, the Lagrangian form of the Navier-Stokes and displacement equations are expressed as

$$\begin{cases} \frac{d\rho}{dt} = -\rho \nabla . v \\ \frac{dv}{dt} = \frac{1}{\rho}\left(-\nabla p + \mu \nabla^2 v\right) + g \\ \frac{dr}{dt} = v \end{cases} \quad [1]$$

Mohamed Roudane, Department of Mechanical Engineering, University of Blida1, Algeria.

Sid Ahmed Gouri, Sofiane Khelladi, Abdelkade Krimi and Abdarrahmane Mahalam, Department of Mechanical Engineering, University of Blida1, Algeria

where $\frac{d(.)}{dt}$ represents the material derivative following an infinitesimal fluid element. is the nabla operator (gradient), ρ,μ,p, v, r and g represent density, dynamic viscosity, pressure, velocity vector, position vector, and the gravitational acceleration vector, respectively. FVis denotes the viscous force.

The weakly compressible smoothed particle hydrodynamics approach (WCSPH) was used in this work (5). In order to close the system (1),it is required the use of an equation of state (EOS) which explicitly defines the pressure from the density instead to solve the Poisson equation. In this work the isothermal equation of state (6) is used which is expressed as

$$p = \rho_0 c_0^2 \left\{ \left(\frac{\rho}{\rho_0} \right) - 1 \right\} \qquad [2]$$

Where ρ0 and c0 denote the reference density, the reference pressure and reference speed of sound. Here the value of c0 is chosen equal to 10 times of the reference velocity related to the problem to keep the Mach number close to 0.1 in order to ensure the incompressibility of the fluid (6).

B. Discrete Form Of Governing Equations

The smoothed particle hydrodynamics is a meshless method. It discretizes the physical space into many discrete elements, usually called particles, without any connectivity among them. This method is based on the approximation of any physical scalar (or vector) field using the convolution formulation. Numerically, it is performed by replacing the Dirac delta function with a regular smooth function, which is called kernel. This function must satisfy some conditions such as symmetry (even function), normalization, and compactness of it support, among others. We refer the interested reader to (7) for more details.

The kernel function used in this work is the quintic spline (8). This kernel was selected since it prevents the high disorder in the particle distribution (9). The kernel function depends on a parameter h, called the smoothing length h, which defines the domain of influence of the kernel function. In this work, the smoothing length h is a constant which is chosen relative to the initial inter-particle distance δx0 (h = 1.33 δx0). The initial particle volume is taken as V0 = δx0d, with d is the space dimension number. The mass of each particle i of different fluid phases is chosen to be constant and equal to m = ρ0PhaseV0 during all the simulation time.

In the context of SPH method, the discretized version of the system (2) can be read as

$$\begin{cases} \frac{d\rho_i}{dt} = \rho_i \sum_j^{n_b} \frac{m_j}{\rho_j} \boldsymbol{v_{ij}} \boldsymbol{\nabla} W_{ij} \\ \qquad + 2\delta h_i c_0 \sum_j^{n_b} (\rho_i - \rho_j) \frac{r_{ij}}{r_{ij}^2} \boldsymbol{\nabla} W_{ij} V_j \\ \frac{d\boldsymbol{v_i}}{dt} = -\frac{1}{m_i} \sum_j^{n_b} (V_i^2 + V_j^2) \frac{\rho_j p_i + \rho_i p_j}{\rho_i + \rho_j} \boldsymbol{\nabla} W_{ij} \\ + \frac{1}{m_i} \sum_j^{n_b} (V_i^2 + V_j^2) \frac{2\mu_i \mu_j}{\mu_i + \mu_j} \frac{\boldsymbol{v_{ij}}}{r_{ij}} \frac{\partial W}{\partial r_{ij}} + \boldsymbol{g_i} \\ \frac{d\boldsymbol{r_i}}{dt} = \boldsymbol{v_i} \\ p_i = \rho_{0_i} c_{0_i}^2 \left\{ \left(\frac{\rho_i}{\rho_{0_i}} \right) - 1 \right\} \end{cases} \qquad [3]$$

where p, and m, are the density and the mass of the particle i, respectively. $W_{ij} = W(r_{ij}, h)$ is the Kernel function, $r_{ij} = r_i - r_j$ is the distance between the parti¬cle i and its neighbours j. The number of particles in the neighborhood of particle i is denoted as $n_b . V_i = \frac{m_i}{\rho_i}$ is thevolume of particle i. The term $\left(\nabla W_{ij} = \frac{\partial W}{\partial r_{ij}} e_{ij} \right)$ is the gradient of the kernel function, and $e_{ij} = \frac{r_{ij}}{r_{ij}} = \frac{r_i - r_j}{r_{ij}}$ is the unit inter-particle vector. $\upsilon_{ij} = \upsilon_i - \upsilon_j$ is the relative velocity between the particle i and j. The value of the dimensionless parameter δ is chosen as (δ = 0.1) as in (10)

For the generalized wall boundary condition, we used the methods proposed in (11) and (12). In this method, three layers of dummy particles must be added in the normal direction to the wall interface. The system is integrated by usnig the Predictor Corrector scheme proposed by krimi et al (12).

III. Application And Validation

A. Comparison Of SPH Method With Both VOF Based Method And Experimental

In this work, for verifying the numerical developed model results, a previously realized work published by **W.C. Moon** and al(1) is considered. This later has been set and checked based on an experimental study of the collapse of liquid performed by (13). We keep the same geometry an parameters of water column with an obstacle for comparing our results. Fig. 1 shows the diagram of the experimental configuration used, with (h0 = 0.3m) is the initial height of the fluid's column, (L0 = 0.15m) is the initial length of the fluid's column.

The purpose of this first comparison study is to highlight the advantage of using meshless methods over mesh dependent ones at reproducing the considered experimental test in term of free surface profile. the fig. 2 shows the illustrations of the experimental dam-break with obstacle from (1, 13) in the left, the numerical VOF method simulation results performed by **W. C. Moon and al**(1) in the middle and our SPH based model simulation results in the right-side of fig. 2.

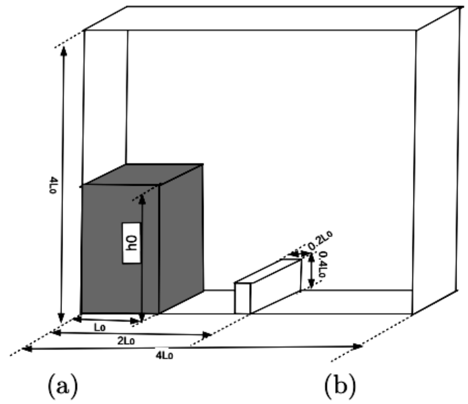


Fig. 1. Collapse of rectangular fluid column: Schematics of the experimental configuration of a dam-break with obstacle (13)

At time t = 0, The fluid column was constrained by

a diaphragm, after removing it from it’s position, the flow motion begins. Simulation results of fig. 2 illustrate the most critical steps of the experience as follow: after the first impact of the fluid with the obstacle at time t = 0.2s, the front wave of the fluid overtops the obstacle and splashes up in the air reaching the highest speed (red color). At time t = 0.3s, an important amount of the fluid is advancing in the air towards the wall. At time t = 0.4s, the fluid hit the downstream wall and then the rising water starts to go down at t = 0.5s. In the last sequence at t = 1.0s, all the fluid comes down but still presents displacing small waves on the top surface and turbulence in the middle.

As shown in fig. 2, both VOF and SPH based methods gave accepted results comparing to the experimental ones. But as we can easily figure it, our numerical model shows closer free surface profile to experimental than the VOF based one, especially in term of smoothness. The advantage of SPH method over the VOF one is more clear after the first impact of the fluid with the downstream wall. While the VOF simulation results show deformations on the top side of the free surface and discontinuities on the flow, our model gives high performances at reproducing the simulated phenomena even in the presence of turbulence.

A comparison study is shown in fig. 3. The dimensionless free surfaces profiles of the adopted dam-break with obstacle for the experimental, the VOF based model and our SPH based model simulation results are plotted this time for the tow first preset time steps (for t = 0.2s and t = 0.3s). It is easy to see that our numerical model gives closer profile of the free surface than the other one especially at t = 0.3s (fig. 3 (b)). When the fluid flow becomes more complicated after hitting the downstream wall, it is hard to represent graphically the free surface profile because of the high amount of discontinuities and splashes in the fluid.

B. Convergence Study Of SPH Method

In this section, we compare the simulation results of the considered dam-break with obstacle for different particle resolution in order to verify the convergence of our SPH based method. It is observed that the difference in results becomes less significant as the number of particles increases indicating the spatial convergence of our SPH model. Thus, the developed numerical model allows to perform simulations for various phenomena with a high number of particles and lower time step *dt*, to get higher resolution and also for the same phenomena with lower resolution and greater time step *dt* but still gives very satisfying results, this allows to make an important gain in time when the problem to be solved has large dimensions. This criteria is really important in the assessment of any developed simulation benchmark.

IV. Conclusion

In this paper, an incompressible single phase Smoothed Particle Hydrodynamics (SPH) model was developed in order to simulate a two-dimensional free-surface flow. The accuracy and robustness of our model has been demon¬strated via several comparison studies. First, the obtained results have been projected to experimental ones and VOF based method ones. The mesheless method shows really satisfying capabilities at simulating the adopted experi¬ence even in case of advanced severe turbulence in the fluid’s flow. Another performed comparison study, to highlight the robustness of our model, consists in testing the convergence of the program in term of particles reso¬lution. The efficiency of the developed model considering this important assessment criteria has been validated through three different simulations from lower to higher resolution. Finally, we can say that the use of the SPH as a meshless method guaranties an important amount of benefits over grid-dependent ones, especially in term of accuracy and load and time computation gain.

References

[1] Moon W, Puay H, Lau T [2018] Numerical simulation of free surface flow using a multiphase model with higher order scheme in AIP Conference Proceedings. [AIP Publishing], Vol. 2020, p. 020078. https://doi.org/10.1063/1.5062704

[2] Lucy LB [1977] A numerical approach to the testing of the fission hypothesis. pp. 1013-1024. https://doi.org/10.1086/112164

[3] Gingold RA, Monaghan JJ [1977] Smoothed particle hydro¬dynamics: theory and application to non-spherical stars. pp. 375-389. https://doi.org/10.1093/mnras/181.3.375

[4] Liu GR, Liu MB [2003] Smoothed particle hydrodynamics: a meshfree particle method. [World Scientific]. https://doi.org/10.1142/9789812564405

[5] Monaghan JJ [1994] Simulating free surface flows with sph. pp. 399-406. https://doi.org/10.1006/jcph.1994.1034

[6] Morris JP [2000] Simulating surface tension with smoothed particle hydrodynamics. pp. 333-353. https://doi.org/10.1002/1097-0363(20000615)33:3<333::AID-FLD11>3.0.CO;2-7

[7] Liu M, Liu G, Lam K [2003] Constructing smoothing functions in smoothed particle hydrodynamics with applications. pp. 263-284. https://doi.org/10.1016/S0377-0427(02)00869-5

[8] Morris JP [1996] Analysis of smoothed particle hydrodynam¬ics with applications. [Monash University Australia].
[9] Price DJ [2004] Ph.D. thesis [University of Cambridge Cambridge, UK].
[10] Molteni D, Colagrossi A [2009] A simple procedure to improve the pressure evaluation in hydrodynamic context using the sph. pp. 861–872.
https://doi.org/10.1016/j.cpc.2008.12.004
[11] Adami S, Hu X, Adams N [2012] A generalized wall boundary condition for smoothed particle hydrodynamics. pp. 7057–7075.
https://doi.org/10.1016/j.jcp.2012.05.005
[12] Krimi A, et al. [2018] Smoothed particle hydrodynamics: A consistent model for interfacial multiphase fluid flow simulations. pp. 53–87.
https://doi.org/10.1016/j.jcp.2017.12.006
[13] Koshizuka S, Oka Y [1996] Moving-particle semi-implicit method for fragmentation of incompressible fluid. pp. 421–434.
https://doi.org/10.13182/NSE96-A24205

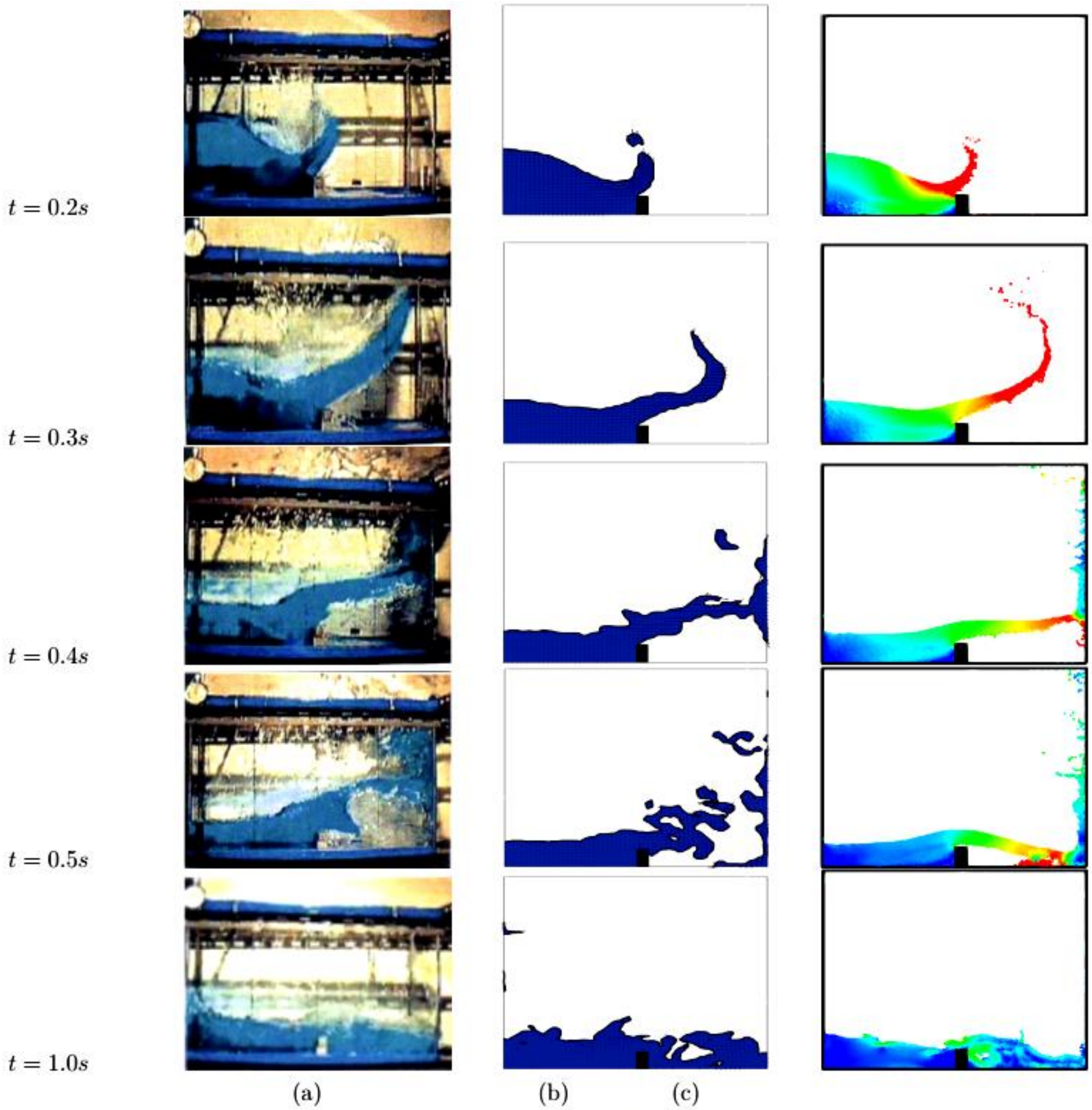
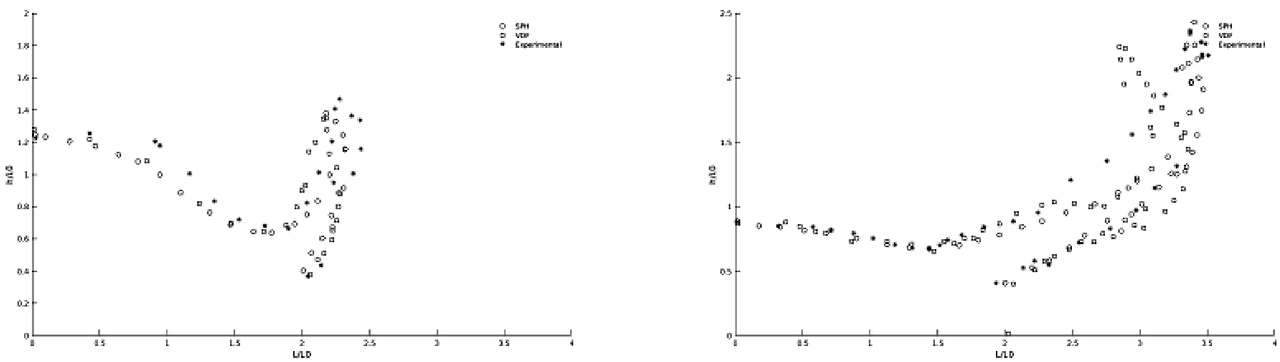


Fig. 2. Free surface profile of dam-break with obstacle for comparison between (a) experiment (13), (b) VOF based method numerical simulation (1) and (c) Our SPH based method numerical model



(a) (b)

Fig. 3. Free surface dimensionless profile of Dam-break with obstacle : Comparison between experimental (Small asterix) (13), VOF based method (Small squares) (1) and Our SPH based method numerical model (Small circles)

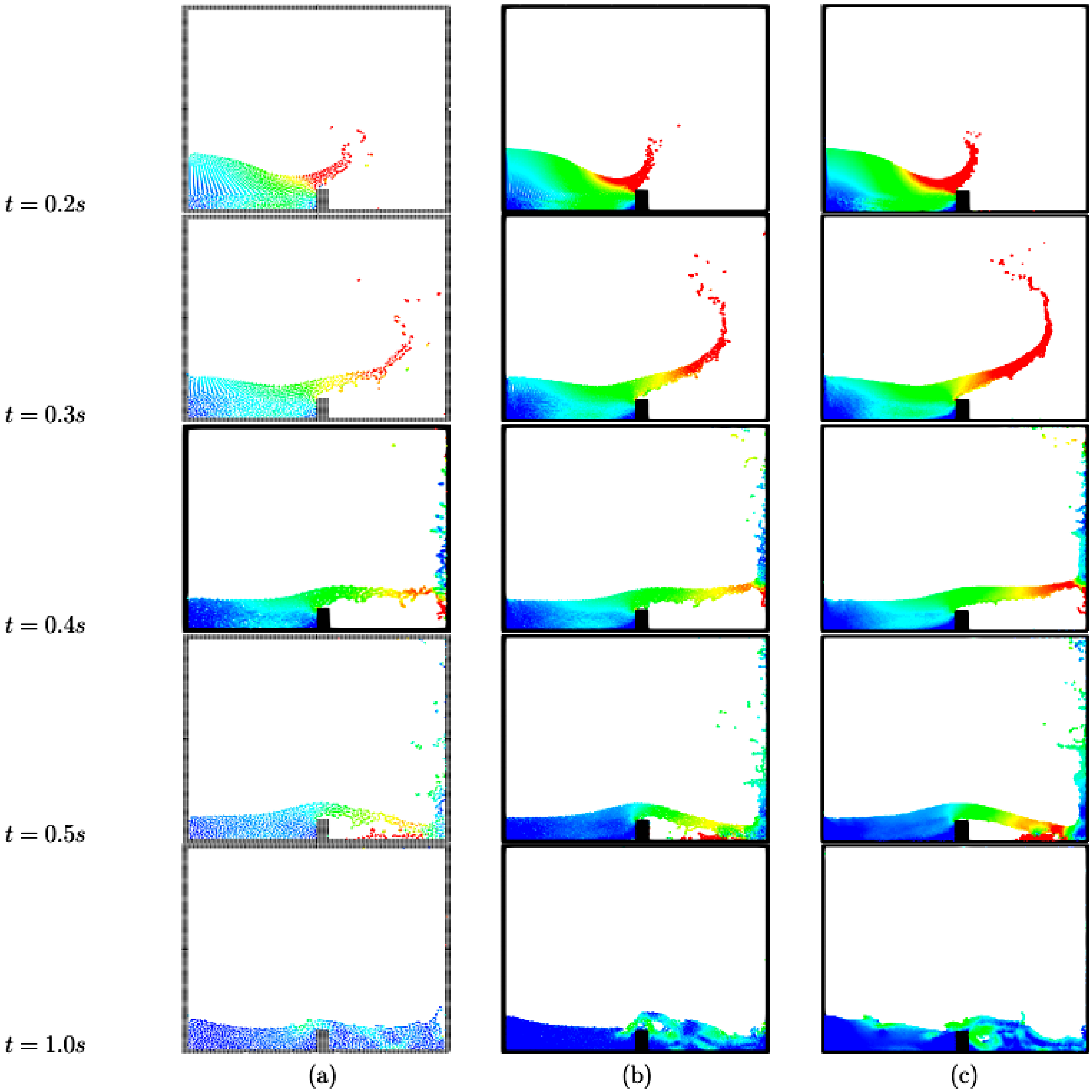


Fig. 4. Five sequences of Dam-break with obstacle of three different particle resolutions: (a) 3348 particle, (b) 7636 particle and (c) 15336 particle