

Numerical simulation of 2D hydraulic jumps using SPH method

Jinbo Lin*, Sheng Jin†, Congfang Ai† and Weiye Ding†

* State Key Laboratory of Coastal and Offshore Engineering
Dalian University of Technology
Dalian 116024, China
e-mail: kingpo@mail.dlut.edu.cn

† State Key Laboratory of Coastal and Offshore Engineering
Dalian University of Technology
Dalian 116024, China
e-mail: tings@mail.dlut.edu.cn

ABSTRACT

A hydraulic jump that generally occurs in river or spillway is a rapid transition from supercritical to subcritical flow characterized by the development of large-scale turbulence, surface waves, energy dissipation and considerable air entrainment. The hydraulic jump is widely used as energy dissipaters in hydraulic engineering due to the high energy dissipation rate. In this study, a weakly compressible smoothed particle hydrodynamics model (WCSPH) is established to simulate the 2D hydraulic jump in open channel. To test the model, two hydraulic jump cases with different inflow Froude number are simulated. The comparison between numerical conjugate depths in the subcritical section with theoretical results show generally good agreement with theory. In addition, an aeration at the jump toe can be clearly observed in numerical results with only Single-phase flow. It is proved that SPH method has unique advantages dealing with the hydraulic jumps.

Keywords: Hydraulic jumps; SPH; Aeration.

1. INTRODUCTION

The hydraulic jumps are a common way to dissipate energy in hydraulic engineering. A hydraulic jump will occur when the supercritical flow discharged from sluice or overflow dam is lifted by the subcritical flow in downstream channel. The turbulence in a jump zone is intense and the energy loss is great. The energy dissipation rate can generally reach 60%-70%. In addition, the stilling basins installed downstream of discharge structures are often adopted to form hydraulic jumps behind gates in the design and construction of hydraulic projects. The stilling basins have the advantages of simple structure, convenient design and construction, and large energy dissipation rate. Therefore, the hydraulic jumps are widely used in large, medium and small hydraulic projects.

The hydraulic jumps have been widely investigated by researchers. López et al. [1] used a similar tank to obtain several jump shapes with different upstream Froude numbers. Then, the experimental data was adopted to check the SPH outcomes. The SPH model provided good average pressures values at the boundaries, but large dispersion was observed for instantaneous water depth. Federico et al. [2] developed a 2D SPH model with a new scheme to enforce different inlet and outlet flow conditions. The proposed treatment could correctly represent the boundary conditions without the generation of spurious pressure shock waves caused by a direct creation or deletion of fluid particles. The model has been successfully validated through several test cases of free-surface channel flows and hydraulic jumps. Babaali et al. [3] studied the hydraulic jump in a convergence stilling basin by a commercially software Flow-3D. The Navier-Stokes equations with standard k- ϵ and RNG model were solved by finite volume model. The comparison of the pressure, velocity, flow rate, kinetics energy, kinetics energy dissipation, and Froude number between numerical results and experimental data shown that this finite volume model could predict the hydraulic jump in a convergence stilling basin, accurately. Jonsson et al. [4] focused on the general behavior of the hydraulic jumps using the SPH methods. Four hydraulic jump cases with different particles resolution were set up and investigated by comparing the conjugate depth with the theoretical results. All of the numerical results shown good agreement with the analytical solution. Their work has shown the possibility to reproduce the internal velocity field

and its impact on the free surface in the hydraulic jumps by a relative simple and coarse SPH model. Azimi et al. [5] used a finite volume model with the volume of fluid scheme to study a hydraulic jump in U-shaped channel. A comparison between the numerical and experimental results shown that the numerical model simulated the flow field characteristics with good accuracy.

The present study uses the discretized governing equations proposed by Federico et al. [2] to simulate two test cases of undular and full hydraulic jumps. The integration of the discretized SPH equations in time is achieved by a two-stage Symplectic method [6]. The time step is a variable value updated in each step. The accuracy of the model is validated by comparing conjugate water depth with the analytical solution. Meanwhile, the evolution of the flow field for the two types of the hydraulic jumps is analyzed and compared.

2. NUMERICAL METHOD

2.1 Governing equations

The governing equations are viscous, weakly compressible Navier-Stokes equations. It is discretized by the SPH method, following Lagrangian form Navier–Stokes equations are obtained [2]

$$\begin{cases} \rho_a \frac{D\mathbf{u}_a}{Dt} = -\sum_b (p_a + p_b) \nabla_a W_b(\mathbf{r}_a) V_b + \rho_a \mathbf{g} + \mu \sum_b \frac{8\mathbf{u}_{ba} \cdot \mathbf{r}_{ba}}{\|\mathbf{r}_{ab}\|^2} \nabla_a W_b(\mathbf{r}_a) V_b \\ \frac{D\rho_a}{Dt} = -\rho_a \sum_b \mathbf{u}_{ba} \cdot \nabla_a W_b(\mathbf{r}_a) V_b \\ p_a = c_0^2 (\rho_a - \rho_0) \\ \frac{D\mathbf{r}_a}{Dt} = \mathbf{u}_a \end{cases} \quad (1)$$

where ρ represents the density; \mathbf{u} is the velocity vector; p represents the pressure; \mathbf{r} represents the position of a generic material point; $\mathbf{g} = (0, 0, -9.81) \text{ m/s}^2$; ρ_0 is the reference density (1000 kg/m³ for water); c_0 represents the reference speed of sound which usually adopts ten times of the maximum wave speed. $\mathbf{\Gamma}$ is the viscous stress tensor; and t is the time.

The sub-index is the a-th and b-th particles. More specifically, $\mathbf{u}_{ba} = \mathbf{u}_b - \mathbf{u}_a$; μ is the dynamic viscosity ($1.0 \times 10^{-3} \text{ N} \cdot \text{s/m}^2$ for water. For ideal fluid, there is no viscosity in fluid. An artificial viscosity is adopted to maintain computational stability. Here a formula $\mu = \rho_0 \alpha h c_0 / 8$ is adopted. Following Federico et al. [2], $\alpha = 0.02$ is taken.); V is the particle volume, $V = m / \rho$, where m represents the particle mass; $W_b(\mathbf{r}_a)$ refers to the kernel function at b-th particle induced by a-th particle. In this paper, a renormalized Gaussian kernel [7] is adopted with the smoothing length $h = 4\Delta x / 3$. A two-stage Symplectic method [6] is selected to integrate the discretized SPH equations in time. Meanwhile, the time step is a variable value updated in each step. The symplectic integration scheme is time reversible when there is no friction or viscous effects. The governing equations of N-S and motion can be rewritten as:

$$\frac{d\mathbf{v}_a}{dt} = \mathbf{F}_a; \quad \frac{d\rho_a}{dt}; \quad \frac{d\mathbf{r}_a}{dt} = \mathbf{v}_a \quad (2)$$

At the predictor stage, the acceleration and density are updated:

$$\mathbf{r}_a^{n+1/2} = \mathbf{r}_a^n + \mathbf{v}_a^n \frac{\Delta t}{2}; \quad \rho_a^{n+1/2} = \rho_a^n + D_a^n \frac{\Delta t}{2} \quad (3)$$

where the superscript n represents the time step.

At the corrector stage, half-time step values are used to calculate the next time step values of velocity and position.

$$\mathbf{v}_a^{n+1} = \mathbf{v}_a^{n+1/2} + \mathbf{F}_a^{n+1/2} \frac{\Delta t}{2}; \quad \mathbf{r}_a^{n+1} = \mathbf{r}_a^{n+1/2} + \mathbf{v}_a^{n+1} \frac{\Delta t}{2} \quad (4)$$

2.2 Boundary conditions

For SPH model, the free surface can be captured naturally without additional special treatment. As for the wall boundaries, a fixed ghost particle technique is adopted to construct the wall boundary with four-layer fixed wall particles. The wall particles are fixed at its position while the density, velocity, and pressure of wall particles are determined by the mirror particles in the fluid domain. In this work, a slip boundary condition is selected to reproduce the inviscid condition. The inflow and outflow condition are treated by two buffer zone. Four-layer inflow and outflow particles are initially contained in the inflow and outflow buffer zone. Particles in buffer zone carry specific values of density, velocity, and pressure. The detail of boundary conditions can be found in [2].

3. NUMERICAL TEST CASES

An undular and full hydraulic jump test cases [2] are simulated to validate this model by comparing the numerical conjugate water depth with the analytical solution.

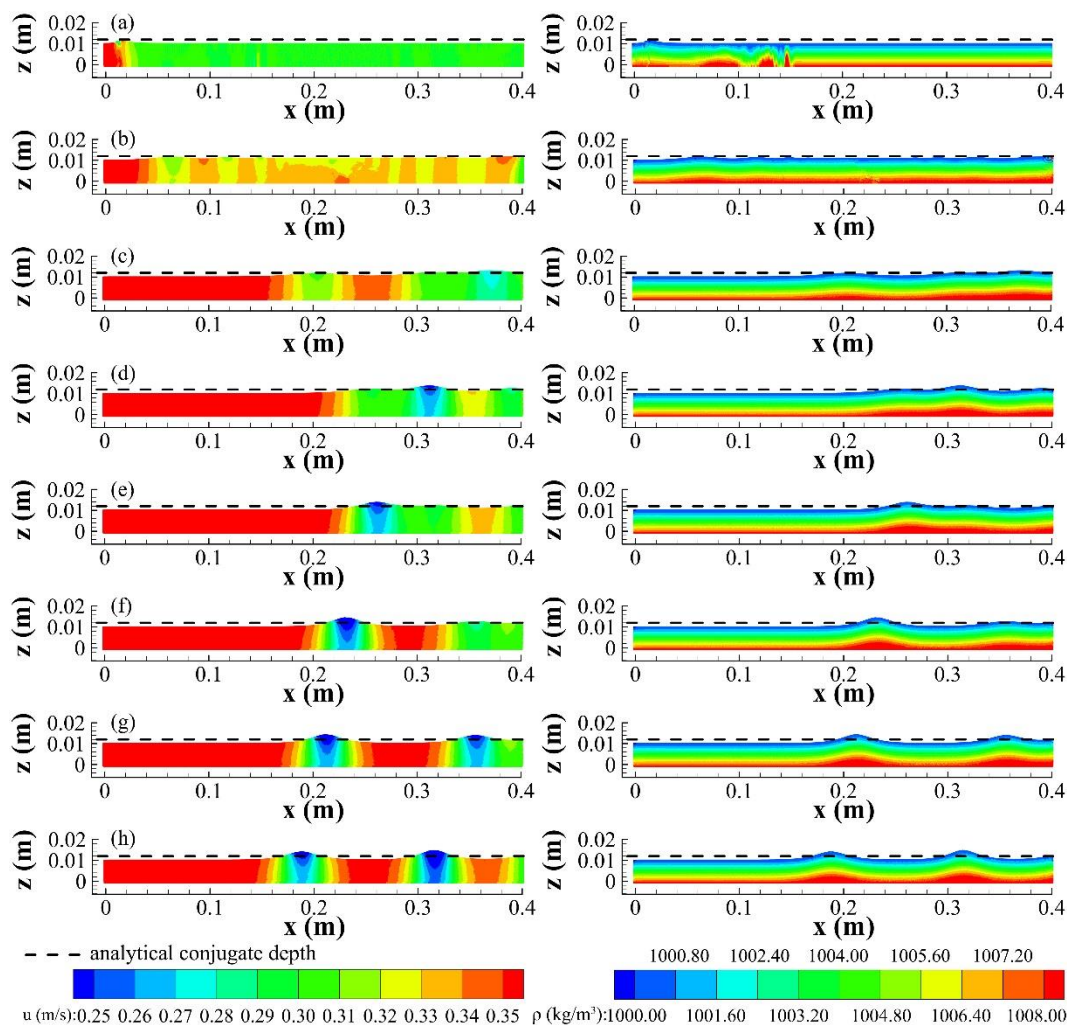


Fig. 1. Velocity (left) and density (right) magnitude field of case 1: (a) $t = 0.04$ s; (b) $t = 0.66$ s; (c) $t = 6.14$ s; (d) $t = 8.52$ s; (e) $t = 9.60$ s; (f) $t = 11.80$ s; (g) $t = 13.42$ s; (h) $t = 15.96$ s.

For ideal fluid, the conjugate water depth can be calculated as follow:

$$h_2 = \frac{h_1}{2} (\sqrt{1 + 8Fr_1^2} - 1) \quad (5)$$

where h_1 is the upstream water depth; h_2 is the downstream water depth; Fr_1 is the upstream Froude number $Fr_1 = U_1 / \sqrt{gh_1}$, U_1 is the upstream velocity. The Fr_1 of case 1 and case 2 are 1.15 and 1.88, respectively. The upstream boundary condition sets to $h_1 = 0.01$ m and $U_1 = 0.36$ m/s for case 1. The corresponding downstream boundary condition is $U_2 = 0.3$ m/s. For case 2, $h_1 = 0.01$ m and

$U_1 = 0.589$ m/s are the upstream boundary condition. $U_2 = 0.268$ m/s is the downstream boundary condition. The length of numerical flume is $L = 40h_1$. The initial density and pressure are set according to the distribution of hydrostatic pressure. The space between particles is $h_1 / \Delta l = 50$.

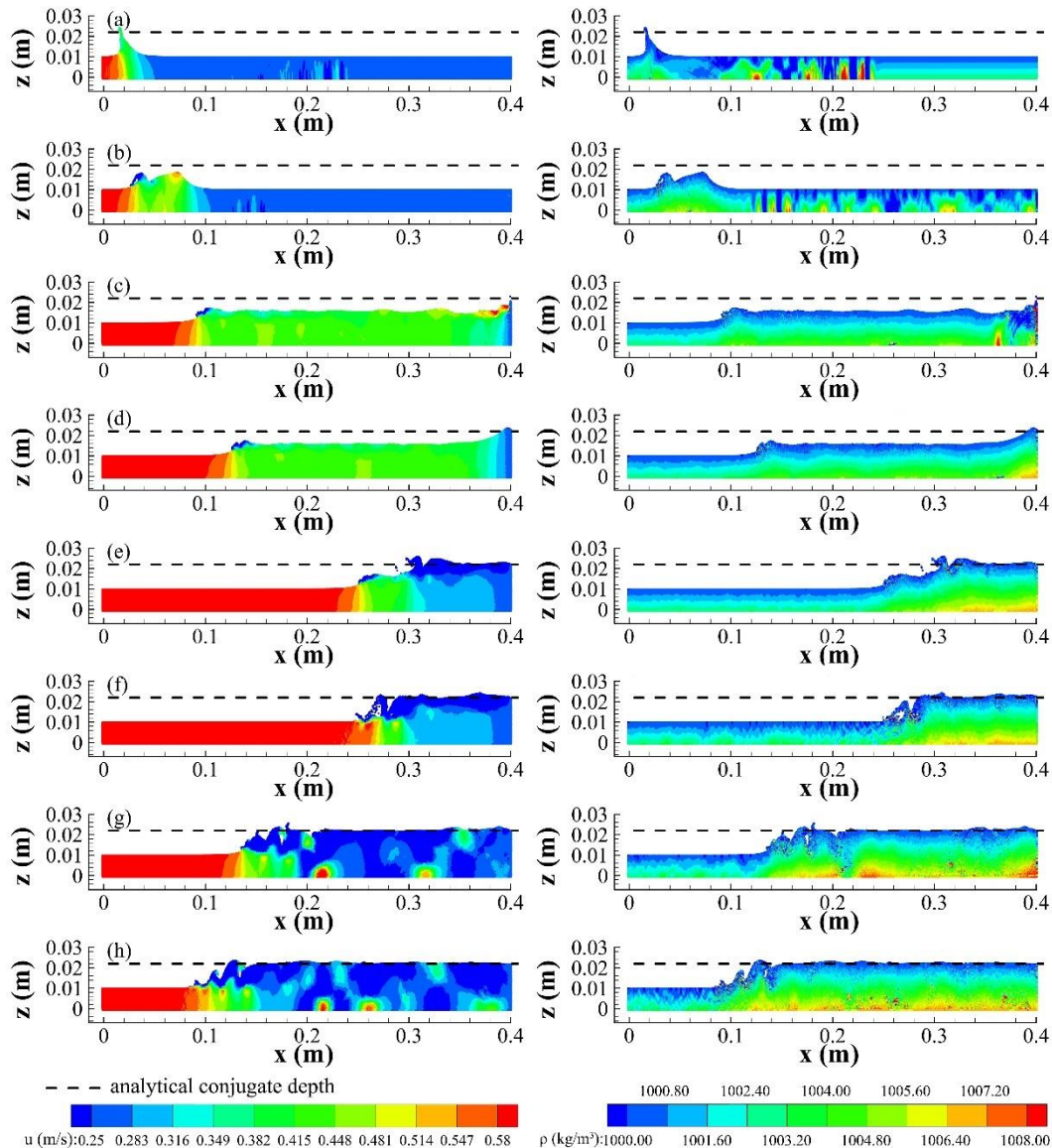


Fig. 2. Velocity (left) and density (right) magnitude field of case 1: (a) $t = 0.04$ s; (b) $t = 0.12$ s; (c) $t = 0.56$ s; (d) $t = 0.80$ s; (e) $t = 1.68$ s; (f) $t = 1.96$ s; (g) $t = 11.90$ s; (h) $t = 15.96$ s.

The velocity and density magnitude field of case 1 are shown in **Fig. 1**. Inflow particles interact with in-domain fluid particles and form a undular jump propagating to downstream at $t = 0.04$ s. Until $t = 0.66$ s, the jump reaches outflow boundary. Meanwhile, a series of weak wave are generated. Then, these waves move downstream until propagate upstream, firstly, at $t = 8.52$ s. Now, there is only one crest in the computational domain. The crest continually propagates upstream at $t = 9.60$ s and $t = 11.80$ s. At $t = 13.42$ s, a new crest downstream of the first crest appears. Finally, the two crests move upstream for a little distance and basically reach quasi-constant state at $t = 15.96$ s. It can be seen that both of the velocity and density field are quite smooth in **Fig. 1**. In addition, the downstream water depth shows a good agreement with the analytical conjugate while the two crests are relatively higher than the analytical data. The maximum errors of the two crests are 0.002 and 0.0025, respectively.

Fig. 2 shows the velocity and density magnitude field of case 2. The evolution of the flow field can be described as follows. At $t = 0.04$ s, the inflow particles with large velocity interact with the slowly in-domain particles and generate a high jump. Two shock waves appear and move downstream at $t = 0.12$ s. Until $t = 0.56$ s, the shock wave arrives the outflow boundary. At $t = 0.80$ s, the shock wave reflects to upstream until the upward shock wave merges with the slower shock wave

at $t = 1.96$ s. An aeration at the jump toe can be clearly observed at this time. Then, the merged jump continually moves upstream and basically reaches quasi-constant state at $t = 15.96$ s. Similar to **Fig. 1**, the velocity field in **Fig. 2** is very smooth. However, the density magnitude field is a little noisy in **Fig. 2**. The numerical conjugate water depth agrees very well with the analytical conjugate water depth.

4. CONCLUSIONS

Two types of 2D hydraulic jump, undular hydraulic jump and full hydraulic jump, are simulated by a WCSPH model. In this model, an artificial viscosity is adopted to stabilize the calculation due to the ideal fluid condition. Comparing the numerical conjugate depth with the analytical solution, the model can accurately reproduce the undular and full hydraulic jump. In addition, the numerical flow field shows that the aeration in the hydraulic jumps can be captured with this Single-phase model. In one word, the WCSPH model is a very power tool to investigate the hydraulic jumps.

5. FUTURE WORK

Though this model calculates the conjugate water depth with a good accuracy, the density field with large inflow Froude number is a little noisy. To eliminate the noise in density field will be our next step work. Besides, the time consumption of SPH model is very large. Therefore, a parallel version of the model is necessary to reduce the time consumption.

REFERENCE

- [1] D. López, R. Marivela, L. Garrote, Smoothed particle hydrodynamics model applied to hydraulic structures: A hydraulic jump test case, *J. Hydraul. Res.* 48, 142–158 (2010).
- [2] I. Federico, S. Marrone, A. Colagrossi, F. Aristodemo, M. Antuono, Simulating 2D open-channel flows through an SPH model, *Eur. J. Mech. B/Fluids.* 34, 35–46 (2012).
- [3] H. Babaali, A. Shamsai, H. Vosoughifar, Computational Modeling of the Hydraulic Jump in the Stilling Basin with Convergence Walls Using CFD Codes, *Arab. J. Sci. Eng.* 40, 381–395 (2015).
- [4] P. Jonsson, P. Jonsén, P. Andreasson, T.S. Lundström, J.G.I. Hellström, Smoothed Particle Hydrodynamic Modelling of Hydraulic Jumps: Bulk Parameters and Free Surface Fluctuations, *Engineering.* 08, 386–402 (2016).
- [5] H. Azimi, S. Shabanlou, S. Kardar, Characteristics of Hydraulic Jump in U-Shaped Channels, *Arab. J. Sci. Eng.* 42, 3751–3760 (2017).
- [6] C. Altomare, A.J.C. Crespo, J.M. Domínguez, M. Gómez-Gesteira, T. Suzuki, T. Verwaest, Applicability of Smoothed Particle Hydrodynamics for estimation of sea wave impact on coastal structures, *Coast. Eng.* 96, 1–12 (2015).
- [7] D. Molteni, A. Colagrossi, A simple procedure to improve the pressure evaluation in hydrodynamic context using the SPH, *Comput. Phys. Commun.* 180, 861–872 (2009).