Open boundary condition for a numerical SPH method to characterize the flow in open channels

Dr. Sarhang Husain
Dams & Water Resources dept.
College of Eng., Salahaddin University
Erbil, Iraq
Sarhang.husain@su.edu.krd

Abstract- In this research an open boundary technique is developed for the numerical SPH method to simulate and characterize the flow field variables in open channels. For this purpose the open source code 2D SPHysics is applied after being modified to satisfy the flow conditions in open channels. The modified code is then calibrated against the experimental results gathered from experiments conducted previously on the flow over embankment dam stepped spillways. The numerical results of this study in terms of the position of free surface flow and velocity distribution at different sections along the chute slope are close to the corresponding experimental results. However, the new technique developed in this study still needs to be calibrated against various open channels problems to verify its applicability.

Keywords; SPH; open boundary condition; stepped spillways.

I- Introduction

SPH is a particle based method initially developed by [18} and [12] in the field of astrophysics. In 1994 [22] modified the classical equations of the SPH method to deal with free surface flow. This method can be considered as the second generation of the computational fluid dynamics CFD [17]. It is a Lagrangian and adaptive particle method [19] in which a set of smooth particles is used to simulate the computational domain. These particles are divided into two kinds; fluid particles to represent the flow and boundary particles to generate the physical geometry. The fluid particles are movable as they can move in space according to some forces acting on them [14], while the boundary particles could be either fixed in their position or movable according to some external forces [8]. Each fluid particle carries the physical fluid properties, including density, pressure and velocity, which might change with time, and moves according to the governing equations [14]. Thus, each fluid particle in the computational domain is considered as an individual interpolation point which may be involved in a summation procedure to estimate a flow field function at a desired particle [20].

Numerous efforts have been made to apply this method in various engineering disciplines, particularly in coastal, fluid-structure interactions and free surface flow problems. To get more familiar and gather more details about the SPH method

the reader is referred to ([21]; [17]; [23], [27] and [16]). The numerical SPH method has gained its popularity in the field of CFD through its capability in dealing with various flow problems. However, the open boundary condition to simulate open channel flow problems is still an open issue of this method. In this study a technique is developed and introduced into the 2D SPHysics code, as an implementation of the numerical SPH method, to simulate and characterize the flow field variables in open channels. The modified code is then calibrated against the experimental results obtained from the experiments conducted by [19] over embankment dam stepped spillways.

II- Literature review

A number of numerical studies have been performed to simulate the skimming flow condition over stepped spillwyas, namely; [6], [1], [27], [5] and [9]. In these studies the authors applied gird-based method such as finite volume and finite element approaches in which mesh generation is needed to reproduce the computational domain. However, [16] applied SPH method to characterize the skimming flow condition over stepped spillways using the large tank approach, which was proposed by [11] to simulate the flow behavior over sharp crested weirs, at the upstream of the stepped spillway.

To verify the capability of particle-based methods in simulating the flow properties recorded from the experimental works in open channels in which experiments are usually conducted under the steady state condition, different techniques have been adopted. [10] applied SPH method to simulate the flow properties for different Froude numbers. They defined two thresholds one at the inlet and the other at the outlet. These thresholds are placed in the computational domain at a distance equal to the kernel radius from the inlet and outlet of the computational domain. They also introduced two new sets of particles, namely the inlet and outlet boundary particles. [26] introduced the inlet and out let boundary condition into a numerical particle-based method known as the moving particle semi-implicit method to simulate the flow over smooth spillways and flow under sluice gates. The study was performed by recycling the fluid particles leaving the outlet into the inlet

after re-setting the value of all flow properties such as position, velocity and pressure with respect to the upstream/downstream flow condition. In the present study the computational 2D SPHysics, as an implementation of the numerical SPH method, source code is modified using the numerical technique developed by [26]. It is worth mentioning here that the numerical SPHysics code is an open source code which has been developed, and is still under development, via the collaboration of four international institutions: The Johns Hopkins University (USA), Universidade de Vigo (Spain), University of Manchester (UK) and University of Rome, La Sapienza (Italy) [14].

III-   Hydraulics of stepped spillways

Depending on the flow rate and step geometry the flow over stepped spillways is classified to three different categories, namely; nappe, transition and skimming flow [4]. Nappe flow deals with relatively low discharges and high steps, while skimming flow occurs when the discharge over the structure is high and the step height is relatively low. Transition flow condition is found for some discharges between the nappe and skimming flow. Researchers in their investigations on stepped spillways observed different characteristics for each flow condition. Further, numerous experimental works have been carried out to define the upper and lower limit for the nappe and skimming flow condition respectively using different chute slopes and step geometry. The present work concentrates on the skimming flow condition on stepped spillways having downstream slopes typical of embankment dams. [4] stated that with the skimming flow condition two flow regions could be clearly observed on stepped spillways, namely; the non-aerated flow region which starts at the crest and extends to some distance to the downstream, and the aerated flow region where air enters the flow and provides white color to the flow. The point separating both regions is known as inception point of air entrainment. [4] defined this point as a point on the free surface where the thickness of developing turbulent boundary layer is equal to the flow depth.

IV-   Governing Equations

Analogue to the grid-based methods, the conservation laws of mass and momentum, Navier Stocks equations, are applied in the SPH method to express the fluid motions. The 2D SPHysics code applied in the current research solves the Navier Stocks equation considering the fluid as weakly compressible [14]. The mass conservation is given as:
In the Lagrangian form, the mass conservation law can be expressed as:

$$\frac{1}{\rho}\frac{D\rho}{Dt} = -\nabla.u \tag{1}$$

where ρ and u are respectively the density and velocity of the particle, t is the time and $\frac{D}{Dt}$ is the total material derivative. The discretized SPH form of the above equation would be:

$$\left(\frac{d\rho}{dt}\right)_a = \sum_b m_b \left(u_a - u_b\right)\nabla_a W_{ab} \tag{2}$$

where $\nabla_a W_{ab}$ is the gradient of the kernel function calculated at a particle a located in the computational domain under the influence of particles b surrounding it inside an influence area, u_a and u_b are the velocity of particles, a and b respectively.
However, the following expression can be used to describe momentum equation on the basis of the Lagrangian approach.

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla P + \Theta + \mathbf{g} \tag{3}$$

where u is the spatial velocity of the particle, P is the pressure and g accounts for the body force per unit mass due to the effect of gravitational acceleration. In this equation, all diffusion terms, caused mainly by the effects of the fluid viscosity, are parameterized by Θ.

Different approaches have been proposed to model the diffusivity term, Θ in the SPH momentum equation, which is highly affected by the fluid viscosity, especially in free surface flows. Following [16] in this investigation the SPS turbulence model, which is initially developed by [15], is applied. This is because the current study is carried out to simulate the flow over stepped spillways, which is characterized by fully turbulent flow.
The discretized SPH Lagrangian form of the momentum equation in the SPH notations:

$$\begin{aligned}
\left(\frac{du}{dt}\right)_a = &-\sum_b m_b \left(\frac{P_b}{\rho_b^2} + \frac{P_a}{\rho_a^2}\right)\nabla_a W_{ab}\\
&+ \sum_b m_b \left\{\frac{4\nu_o r_{ab}\nabla_a W_{ab}}{(\rho_a+\rho_b)|r_{ab}|^2}\right\}\mathbf{u}_{ab}\\
&+ \sum_b m_b \left(\frac{\tau_b}{\rho_b^2} + \frac{\tau_a}{\rho_a^2}\right)\nabla_a W_{ab}
\end{aligned} \tag{4}$$

The above equation conserves both the linear and angular momentums of all of the particles involving the computational domain [14].
As mentioned earlier the numerical SPHysics code applied in this study considers the fluid as weekly compressible which allows the explicit determination of pressure on fluid particles using the state equation [2] expressed as:

$$P = B\left[\left(\frac{\rho}{\rho_o}\right)^{\gamma} - 1\right] \tag{5}$$

where P is the pressure, and ρ_o is the reference fluid density. The density of the fluid at the free surface, which is subject to atmospheric pressure, is usually taken as the reference fluid density; for water ρ_o= 1000 kg/m³, γ is the polytrophic constant, with values ranging from 1 to 7 which fits the oceanic data, and B is the bulk modulus elasticity of the fluid [14].

Particles used in the simulation with an SPH method are moved according to the following expression, known as the XSPH variant [20]:

$$\frac{dr_a}{dt} = \mathbf{u}_a + \epsilon \sum_b \frac{m_b}{\rho_{ab}} \mathbf{u}_{ba} W_{ab} \qquad (6)$$

where ϵ is a constant the value of which varies from zero to one with a frequently used value as 0.5 and $\rho_{ab} = (\rho_a + \rho_b)/2$.

V-   Boundary Condition

Only two boundary conditions are provided by the 2D SPHysics code, namely; close and periodic boundary conditions. This motivates the author to modify the source code by introducing a numerical SPH open boundary condition technique such that the numerical code can simulate the flow in open channels. This is carried out by removing the rigid walls fixed on the left and right hand sides of the computational domain. Following [26] particle storage is allocated which contains a number of fluid particles having zero velocity and pressure values. Then, in order to keep the flow at the steady state condition, a column of fluid particles is continuously added to a zone located inside the computational domain at about the kernel radius from the inlet at a regular time interval. The sub-critical flow condition is applied in this study to meet the condition of flow at the upstream of stepped spillways [4]. The velocity of each fluid particle of this column is dependent on the discharge and upstream flow depth measured in the experimental work. The time required to add the column of fluid particles at the inlet depends on the size of fluid particles and their velocities. Moreover, fluid particles leaving the domain at the outlet are returned to the particle storage and their properties are set to zero. Then, these particles are added to the inlet after assigning new velocity/ pressure values depending on the flow condition (supercritical/ subcritical) at the inlet and outlet of the computational domain.

VI-   Code Calibration

To see how the open boundary condition technique, which is developed in this study and introduced into the 2D SPHysics, works, it is calibrated against the experimental results of [19] measured in the non-aerated flow region for a range of flow rate typical of the skimming flow over embankment dam stepped spillways. To do so, the experimental results are compared with the modified 2D SPHysics computational results in terms of the position of free surface, length to and depth at the inception point and velocity distribution at different sections along the chute slope in the non-aerated flow region for various unit discharges tested in the experimental work. [19] conducted a series of experiments on the skimming flow condition in the non-aerated flow region on moderate slope stepped spillways. The downstream slope was 1V:2H on which ten identical steps

of height 5cm were used to fit the slope from the spillway crest to its end. The spillway crest was of broad crested weir type of 0.5m length.

In order to set up the numerical model, in the present work the repulsive boundary particles developed by [20] is used to create a physical geometry identical to that constructed in the laboratory by [19]. However, the Cartesian gird method is used to fix the position of fluid particles at the reservoir upstream of the spillway. The length of the reservoir is fixed to 1.5m, while the depth of water is variable according to the unit discharge tested in the experiments. The distance between fluid particles, boundary particles and fluid and boundary particles is constant and set to 0.005m. Fig. 1 presents the computational domain of the physical laboratory model described above that is generated by the 2D SPHysics code.

Also, it is worth mentioning that cubic spline kernel function is used to approximate the flow filed variable and the kernel correction technique is incorporated in this study to fix the numerical instability which may be produced due to the kernel truncation for particles near the free surface and boundaries. Moreover, the non- conservative Riemann solver method is applied in the current study to smooth out fluctuations which may be occurred in the pressure and velocity flow fields as a result of considering the fluid as weakly compressible. Furthermore, The Verlet time integrator scheme used by [30] is adopted in the current study. Moreover, in this study the whole simulation time is set to 8 sec and following [16] the time step is initially set to 10^{-4}sec to fulfil the CFL condition.

VII-   Results and Discussion

The unit discharge value of the numerical model is obtained from the flow depth and velocity profile at the critical section, which occurs at the point on the weir crest where the Froude number equals one. This allows using the following equation to estimate the numerical unit flow rate at the critical section on the spillway crest [7]:

$$q = \sqrt{h_c^3 g} \qquad (7)$$

where q is the discharge per unit channel width, h_c is the flow depth at the critical section on the weir crest known as the critical flow depth and g is the gravitational acceleration. In the current work the critical section for all discharges is found at about one third of the crest length downstream of the upstream crest corner which well agrees to the range reported in [7] and [1]. Furthermore, in this study the highest particle existing at any section perpendicular to the flow direction is defined as a free surface particle. Moreover, velocity profile in this study is taken at various sections perpendicular to the flow direction. Each section includes a number of fluid particles, a, and the velocity of each particle, V_a, is estimated from the equation given by [13]:

$$V_a = \frac{\sum_b V_b W_{ab}}{W_{ab}} \qquad (8)$$

where, b denotes all fluid particles located inside the influence area of the particle being considered, V_b is the velocity of surrounding particles and W_{ab} is the kernel function.

To qualitatively compare the experimental and the current numerical results a number of snapshots taken from the modified 2D SPHysics code during the numerical simulation are presented in Fig. 2. These snapshots depict what does really happen for the flow on stepped spillways under the skimming condition. This mainly includes: occurring a jet as water initially flows down over the steps, circulation of part of fluid inside the steps while the jet striking the step treads [1] and finally the appearance of the flow as a coherent stream over the steps [4]. This is an indication that the modified numerical code could detect the skimming flow condition over stepped spillways.

In the current work quantitative comparison is also conducted. For this purpose the flow depth (d) perpendicular to the flow direction along the chute slope (L) measured from the end of the spillway crest for the range of critical flow depth h_c normalized by step height h_s $1.27 \leq h_c/h_s \leq 1.7$ are plotted in Fig. 3 and compared with the corresponding values obtained from the experimental results gathered by [19]. As can be seen the comparisons demonstrate very good agreement between the experimental and the computational results predicted by the modified 2D SPHysics code with a maximum relative error of less than 7%.

To find the coordinate of the inception point of air entrainment in the current study the definition of [4] is considered who states that it is the intersection point of flow depth and thickness of boundary layer. The thickness of boundary layer at any section in the non-aerated flow region on stepped spillway can be estimated from the velocity profile as that depth above which the velocity is uniform. In the present investigation the velocity flow field along the chute slope estimated by the modified 2D SPHysics for $h_c/h_s = 1.74$ and 1.59 are presented in Figs 4a and 4b respectively. Apparently, different velocity distributions on the steps can be observed along the flow direction. The flow velocity at steps close to the weir crest is relatively low and is almost constant over the entire flow depth. This indicates that the thickness of boundary layer is small. Then the velocity increases towards the downstream and reaches the peak value at a section called the inception point of air entrainment. At this section the flow velocity varies across the whole flow depth revealing that the boundary layer has fully developed. Furthermore, [19] measured the streamwise flow velocity, V, parallel to the chute slope at different sections for various unit discharges. Fig. 5 and Fig. 6 respectively show the velocity profiles computed by the numerical code and those recorded by [19] at two sections within the non-aerated flow region. In these profiles, y represents the distance of the point, where the flow velocity was measured, from the step outer edge taken at any section perpendicular to the pseudo bottom formed by the step outer edges and V is the flow velocity perpendicular to the chute

slope and to the flow direction. As can be seen from these figures the shape of the velocity profiles for both the computational and experimental works are almost the same. However, there is a slight difference between the numerical and experimental velocity values with a maximum relative error of 6%.

The velocity profiles at different section along the chute slope allow the author to detect the growth of boundary layer and estimate its thickness. Fig. 7 shows the determination of the inception point of air entrainment for the unit discharge $q = 0.05$ m^2/s and $2.2 \leq L/Ks \leq 5$. In this figure δ is the thickness of turbulent boundary layer, L is the distance measured from the crest down to the location on the chute slope where the flow depth d is considered and K_s is the roughness height perpendicular to the pseudo bottom which can be determined from $K_s = h_s \cos\theta$, where θ is the slope of the stepped spillway. Table 1 compares the experimental results of [19] and the modified 2D SPHysics numerical predictions in terms of the coordinates of the inception point for the range of discharge tested in the experimental work. This table clearly shows that both results are close to each other with a maximum relative error% of about 6.54%.

VIII-   Conclusions

The open boundary in the numerical SPH method is addressed in this study. An open source code known as SPHysics code is used after being modified to deal with fundamentals of steady state flow condition. Then the modified code is calibrated against an experimental work carried out previously on flow over stepped spillways. Qualitative and quantitative comparisons are made. The numerical results are promising as they are in close agreement with the experimental results in terms of different flow parameters including free surface profiles and velocity profiles for different unit discharges typical of skimming flow condition on embankment dam stepped spillways. The results of this study help the researchers and engineers to study the flow and design this kind of hydraulic structure. However, the author recommends applying the modification that is performed in this study and introduced into the 2D SPHysics code on different steady state flow cases, especially flow in open channels to verify its validity.

REFERENCES

[1] André, S. "High velocity aerated flows on stepped-chutes with macro-roughness elements" Communication 20, (2004), Laboratorie de Constructions Hydrauliques Ecole Polytechnique Federale de Lausanne: Lausanne, Switzerland. ISSN 1661-1179.
[2] Batchelor, G., K. "Introduction to fluid mechanics" (1974), Cambridge University press, UK.

[3] Chanson, H. "Hydraulics of skimming flows over stepped channels and spillways" Journal of Hydraulic Research, (1994), 32(3): pp.445-460.

[4] Chanson, H. The hydraulics of stepped chutes and spillways. (2002),Lisse, A.A. Balkema publishers.

[5] Carvalho, R. F., & Martins, R. "Stepped spillway with hydraulic jumps: application of a numerical model to a scale model of a conceptual prototype", 2009, .Journal of Hydraulic Engineering, 135(7), 615-619.

[6] Chen, Q., Dai, G., and Liu, H. "Volume of fluid model for turbulence numerical simulation of stepped spillway overflow.", Journal of Hydraulic Engineering, 2002, 128(Compendex): 683-688.

[7] Chow, V., T. Open channel hydraulics , McGraw-Hill, 2002, New York, USA.

[8] Dalrymple, R. A. and Rogers, B. D. "Numerical modelling of water waves with SPH method" Journal of Coastal Engineering , 2006, 53(2-3): pp.141-147.

[9] Fabian, A., Ines, M., Matos, J. "Laboratory measurements and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways." Journal of Environmental Fluid Mechanics: in press 26/8/2010: DOI 10.1007/s10652-010-9188-6.

[10] Federico, I., Marrone, S., Colagrossi, A., Aristodemo, F. and Veltri, P. "Simulating free-surface channel flows through SPH". In Proc. 5th International SPHERIC Workshop, Manchester, 2010.

[11] Ferrari, A. "SPH simulation of free surface flow over a sharp crested weir " Journal of Advances in Water Resources, 2010, 33 (3): pp.270-276.

[12] Gingold, R. A., and Monaghan, J. J. "Smoothed particle hydrodynamics, theory and application to non-spherical stars" Mon. Not. Roy. Astr. Soc., 1977, 181, pp.375-389.

[13] Gómez-Gesteira, M., Cerqueiro. D., Crespo, C., and Dalrymple, R. A. "Green water overtopping analysed with a SPH model." Journal of Ocean Engineering, (2005), 32, pp.223-238.

[14] Gómez-Gesteira M., Rogers B.D., Crespo A.J.C., Dalrymple R.A, Narayanaswamy M. and Dominguez J.M., "SPHysics - development of a free-surface fluid solver- Part 1: Theory and Formulations". Computers & Geosciences, DOI:10.1016/j. cageo.2012.02.029, 2012.

[15] Gotoh, H., Shao, S. And Memita, T.. "SPH-LES model for numerical investigation of wave interaction with partially immersed breakwater" Journal of Coastal Engineering, 2004, 46(1):p39-63.

[16] Husain, S.M., Muhammed, J.R., Karunarathna, H.U. and Reeve, D.E. "Investigation of pressure variations over stepped spillways using smooth particle hydrodynamics". Advances in Water Resources, 2014, 66, pp.52-69.

[17] Liu, M. B. and Liu, G. R. "Smoothed Particle Hydrodynamics-A Meshfree Particle Method. 2003, Singapore", World scientific.

[18] Lucy, L. B. "A numerical approach to testing the fission hypothesis" The Astronomical Journal, 1977, 82(12): pp.1013–1924.

[19] Meireles, I. and Matos, J. "Skimming flow in the non-aerated region of stepped spillways over embankment dams." Journal of Hydraulic Engineering, 2009, 135(8): pp.685-689.

[20] Monaghan, J. J. "On the problem of penetration in particle methods" Journal of Computational Physics, 1989, 82(1): pp.1-15.

[21] Monaghan, J. J. "Smoothed Particle Hydrodynamics" Annual Review of Astronomy and Astrophysics, 1992, 30(1): pp.543-574.

[22] Monaghan, J. J. "Simulating free surface flows with SPH." Journal of Computational Physics, 1994, 110: pp.399-406.

[23] Monaghan, J. J. "Smoothed particle hydrodynamics." Reports on progress in physics, 2005, 68.8: 1703.

[24] Rogers, B. D. and Dalrymple, R. A., SPH Modeling of tsunami waves, Advances in Coastal and Ocean Engineering, 2008, Vol. 10 Advanced Numerical Models for tsunami waves and runup, World Scientific.

[25] Rogers, B. D., Dalrymple, R. A. and Stansby, P. K. "Simulation of caisson breakwater movement using 2-D SPH" Journal of Hydraulic Research, 2008, 48 Extra issue: pp.135-141.

[26] Shakibaeinia A, Jin YC. A weakly compressible MPS method for modeling of open- boundary free- surface flow. International journal for numerical methods in fluids. 2010 Aug 10;63(10):1208-32.

[27] Shao, S., Ji, C., Graham, D. I., Reeve, D. E., James, P. W., & Chadwick, A. J.. Simulation of wave overtopping by an incompressible SPH model.Coastal Engineering, (2006), 53(9), 723-735.

[28] Tabbara, M., Chatila, J. and Awwad, R. "Computational simulation of flow over stepped spillways." Computers and Structures, 2005, 83: 2215-2224.

[29] Violeau, D., Issa, R. "Numerical modelling of complex turbulent free surface flows with the SPH method: An overview". Int. J. Numer. Meth. Fluids, 2007, 53(2), pp.277–304.

[30] Verlet, L. "Computer "experiments" on classical fluids. I. Thermodynamical properties of Lennard-Jones molecules." Journal of physical review, 1967,159(1), 98.

Figure 1. 2D schematic representation of the numerical computational domain for simulating the geometry of the experimental work of [19] for flow over stepped spillway.
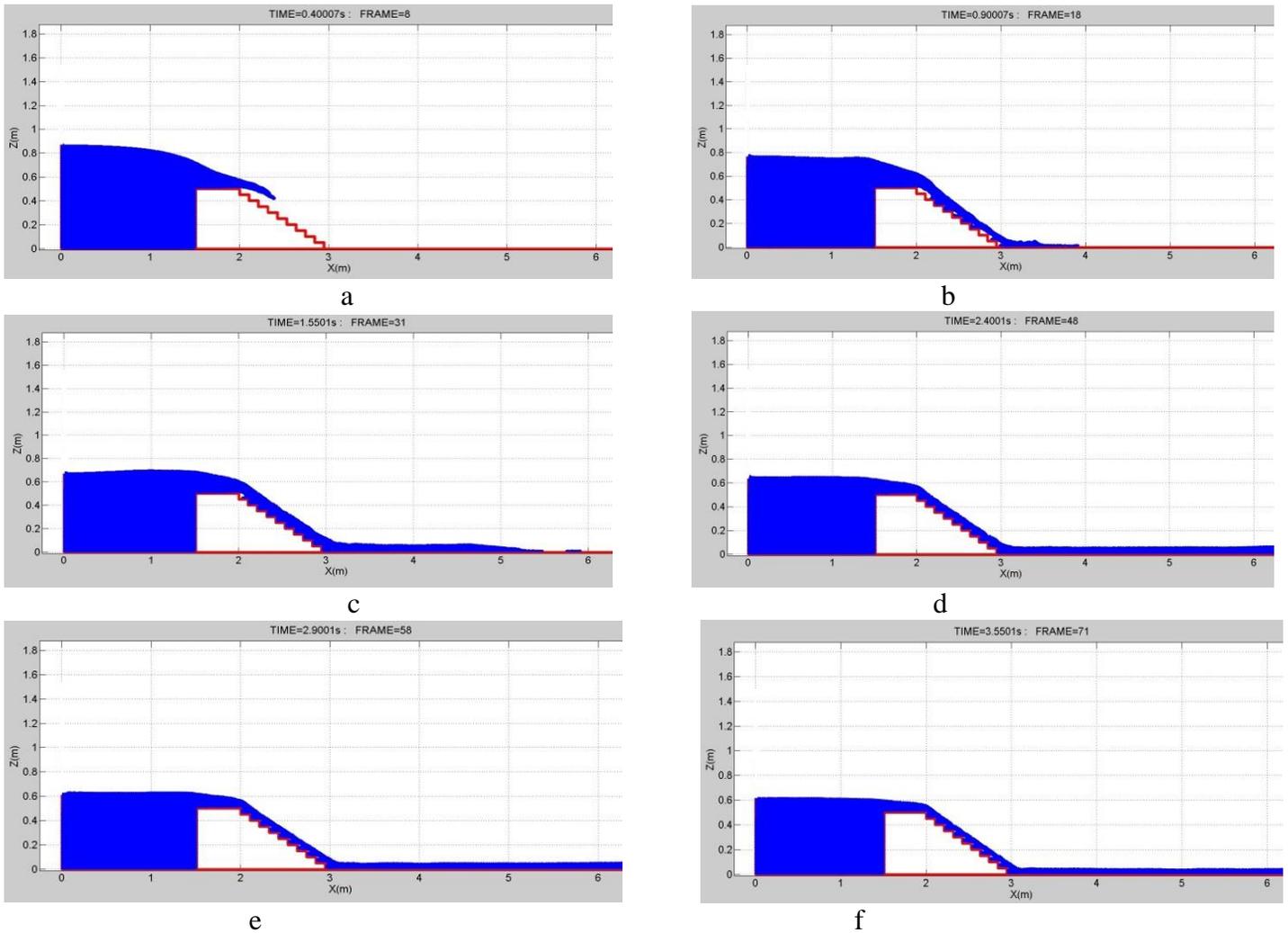

Figure 2. Snapshots of flow over stepped spillways predicted by the SPHysics numerical code applied in this study at (a) 0.40sec; (b) 0.90sec; (c) 1.55sec, (d) 2.40sec, e) 2.90sec and f) 3.55sec.
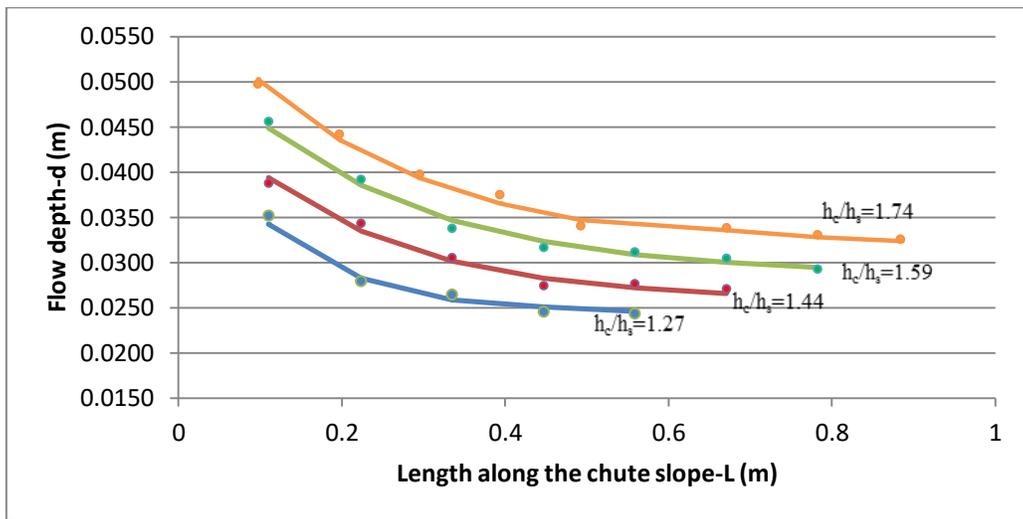
Figure 3. Comparisons between the experimental (curves) and numerical (circles) free surface profiles along the chute slope for different discharges represented by h_c/h_s.
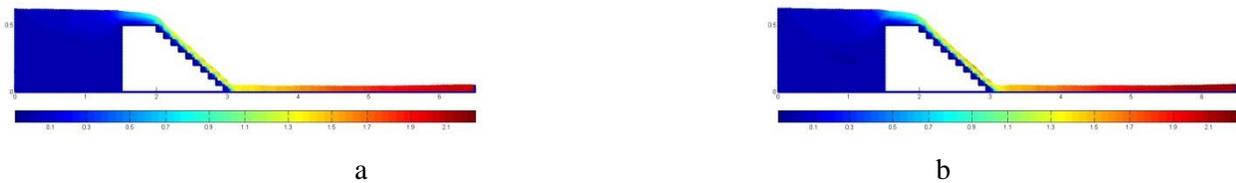

Figure 4. Variation of flow velocity over a stepped spillway predicted by the SPHysics code applied in this study with color coded velocities: (a) $h_c/h_s= 1.74$ and (b) $h_c/h_s= 1.59$.
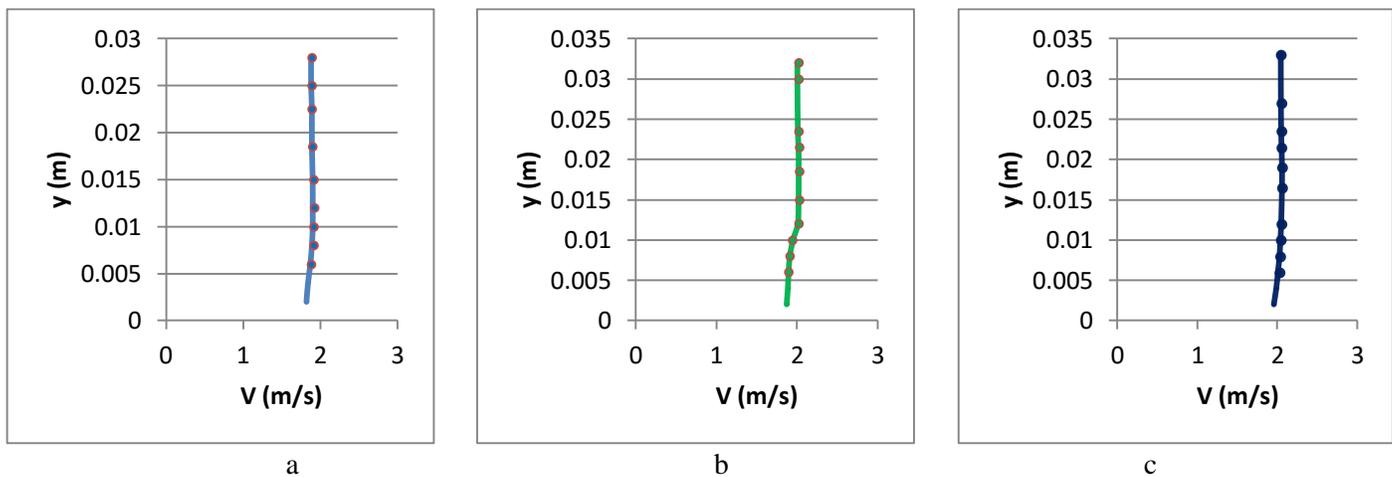

Figure 5. Comparison between the experimental (continuous lines) and SPHysics numerical (circles) velocity profiles at L= 0.11m.   (a) h_c/h_s=1.27, (b) h_c/h_s=1.44 and (c) h_c/h_s=1.59.

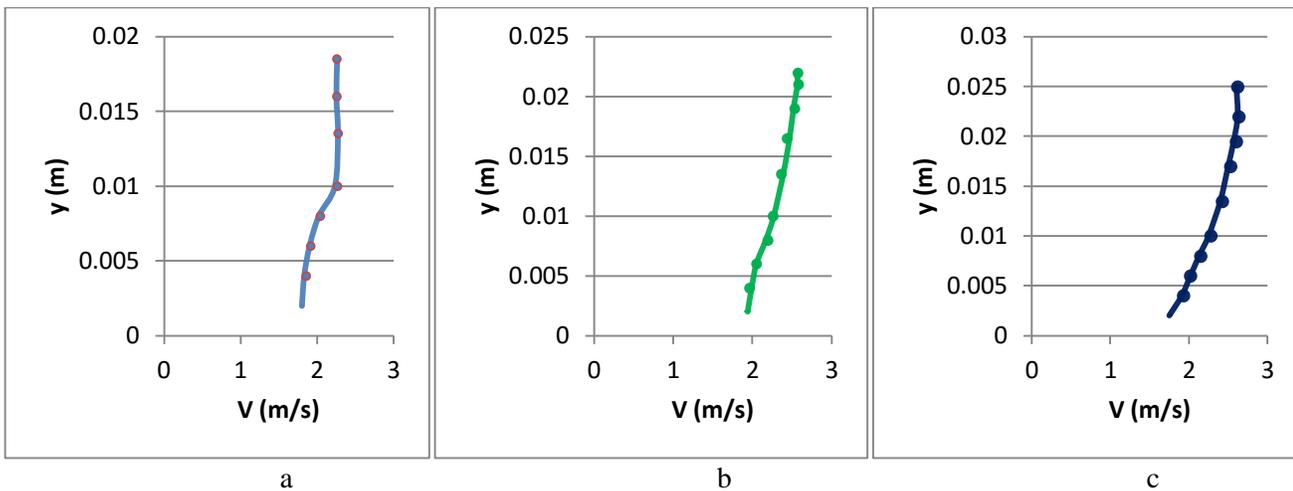a   b   c

Figure 6. Comparison between the experimental (continuous lines) and SPHysics numerical (circles) velocity profiles at the inception point. (a) h_c/h_s=1.27, (b) h_c/h_s=1.44 and (c) h_c/h_s=1.59.
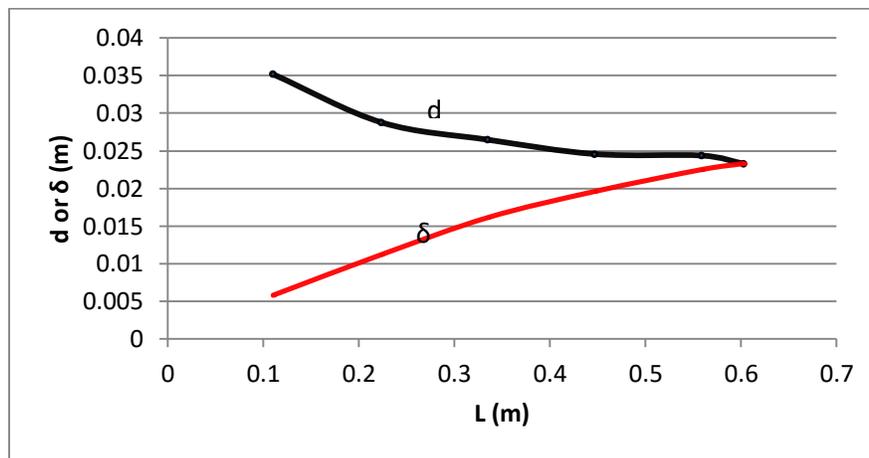

Figure 7. The inception point of air entrainment determined by the SPHysics code from the intersection point of numerical free surface with the developed turbulent boundary layer for $q=0.05m^2/s$ and $2.2 \leq L/K_s \leq 5$.

Table 1. Comparison between the experimental and SPHysics numerical results in terms of the coordinates of the inception point of air entrainment for different normalized discharge, h_c/h, values.

hc/hs	diexp. (m)	diSPHysics (m)	%Error= 100(diexper.-diSPHysics)/diSPHysics	Liexp. (m)	LiSPHysics (m)	%Error= 100(Liexper.-LiSPHysics)/LiSPHysics
1.27	0.0235	0.0242	2.98	0.566	0.603	6.54
1.43	0.0266	0.0274	3.01	0.673	0.698	3.71
1.59	0.0295	0.0303	2.71	0.779	0.821	5.39
1.74	0.0325	0.0335	3.08	0.884	0.914	3.39