Validation of a numerical particle based method for free surface flow downstream of sluice gates

Sarhang M. Husain¹, Abdulla, A. Abo² and Basil Y. Mustafa³

1, 2 Department of Dams and water resources, College of Engineering, Salahaddin University, Erbil, Kurdistan Region, Iraq
3 Department of Civil Engineering, Technical Engineering College, Polytechnic University, Erbil, Kurdistan Region, Iraq

 ARTICLE INFO

Article History:
Received: 14/05/2017
Accepted: 12/04/2018
Published:01/06/2018

Keywords:
Numerical simulation Particle-based method
SPH
Sluice gates

*Corresponding Author Contact
Email:
Sarhang.husain@su.edu.krd

ABSTRACT

This study is a numerical investigation in which a meshfree computational method known as the Smoothed Particle Hydrodynamics (SPH) is applied to examine its efficiency and accuracy in predicting the flow field variables of free surface flow passing a sluice gate. For this purpose, the numerical 2D SPHysics model, as an implementation of the computational SPH method, is adopted. The numerical code is validated against the theoretical and experimental results of previous works. The validation is performed taking into consideration the conjugate flow depths and velocities for different total upstream heads passing under a fixed gate opening height. The quantitative agreement between the results computed by the numerical 2D SPHysics code and the theoretical and experimental results is fairly good confirming that the numerical code is robust in predicting the flow properties in sluice gates. Then, the validated code is used to find the energy dissipation rate for various total upstream heads and Froude numbers. The results obtained in this study are promising, indicating that the numerical model can be considered as an efficient alternative tool for hydraulic engineers to predict and understand the flow behavior in hydraulic structures.

1. INTRODUCTION

Sluice gates have been widely used for flow control and discharge measurement in irrigation and drainage channels. The formation of hydraulic jumps is a natural occurring phenomenon in flowing fluids such as water. The main characteristic is the sudden transition of rapid shallow flow to slow moving flow with rise of the fluid surface better known as a transition from supercritical to subcritical flow. The subcritical flow condition occurs when the value of Froude number is smaller than one, whereas the supercritical flow condition takes place when the value of Froude number is greater than one. However, the critical flow condition occurs when the value of Froude number is exactly equal to one (Chow, 1959). Froude number is commonly defined as the ratio of the inertia and gravitational forces.
Transition between supercritical and subcritical flow are characterized by strong dissipative mechanism, which is favorable when high kinetic energy levels is unwanted such as in spillway flows. Further characteristics of hydraulic jumps are the development of the large-scale highly turbulent zone known as the “roller” where surface waves and spray, energy dissipation and air entrapment is present (Chanson, 2004).

Figure (1) presents the most common parameters needed in describing the flow in sluice gates.

![Figure (1): Schematic diagram showing the most important flow parameters passing sluice gates](image)

Following a streamline for frictionless, incompressible and steady flow the Bernoulli equation states that:

\[ \frac{v^2}{2g} + \frac{P}{\rho g} + Z = \text{constant} \]  

where, \( v \), \( P \), \( g \), \( Z \) and \( \rho \) are the mean flow velocity, pressure, gravitational acceleration and water density respectively.

For the horizontal and frictionless case depicted in the above figure, the Bernoulli equation can be written as:

\[ y_1 + \frac{v_1^2}{2g} = y_2 + \frac{v_2^2}{2g} \]  

(2)

Applying the principles of continuity, the flow rate per unit width of the channel \( q \) must be the same at positions 1 and 2, yielding:

\[ q = y_1v_1 = y_2v_2 \]  

(3)

As stated above the hydraulic jump is characterized by a supercritical and a subcritical region where the depths are significantly different. These depths and are referred to as conjugate depths and can be seen in the schematic diagram of Figure 1.

Substituting eq. (2) into eq. (3) and after some manipulations a relationship between the unit flow rate \( q \) and the depths \( y_1 \) and \( y_2 \) can be obtained as:

\[ q = C\sqrt{2gy_1} \]  

(4)

where,

\[ C = \frac{y_2}{\sqrt{1 + \frac{y_2^2}{y_1^2}}} \]  

(5)

The flow depth at position 2, \( y_2 \), can be approximated by:

\[ y_2 = C_c a_o \]  

(6)

where, \( C_c \) is the vena contraction coefficient and \( a_o \) is the gate opening.

that the most important factors affecting the value of \( C_c \) are the total upstream head, sharpness of the gate and the gate opening. However, the same author recommended 0.57 as the average value for the design of hydraulic structures. On the other hand, Henderson (1966) pointed out that the value of \( C_c \) is 0.61 for this kind of hydraulic structure. Based on these facts and since that the shape and dimension of the gate used in the current work are kept unchanged for all values of the total upstream head, hence its effect on the value of \( C_c \) can be neglected. Following Chanson (2004) the value of \( C_c \) is taken as 0.57 in the current work to determine the theoretical value of \( y_2 \) for the sake of comparison with the corresponding numerical value for various total upstream heads.

A dimensionless relation between the conjugate depths can easily be derived from continuity, momentum and energy equations for a rectangular channel. Here is assumed hydrostatic pressure distribution and uniform velocity distribution at the up- and downstream end of the control volume. Further, the friction between the bottom and the fluid is assumed to be zero. With these assumptions the conservation equations yield the dimensionless relation between the conjugate depths for a rectangular channel as,

\[ \frac{y_3}{y_2} = \frac{1}{2} \left( \sqrt{1 + 8F_r^2} - 1 \right) \]  

(7)
where, $F_{r1}$ is the Froude number upstream of the jump, $F_{r1} = \frac{v_1}{\sqrt{gy_1}}$, which by definition must be greater than one. The upstream Froude number is also used as an indicator of the general characteristics of the jump in a rectangular horizontal channel as different upstream Froude numbers produce different hydraulic jumps (Henderson, 1966).

The importance of the practical implications of sluice gates has drawn the attention of hydraulic researchers to investigate the hydraulics of flow passing under this kind of hydraulic structure. Numerous experimental studies have been conducted to study the hydraulics of flow under sluice gates. While, only few studies are available in the literature concerning the role of numerical approach in identifying the flow characteristics in this type of hydraulic structures. These numerical simulations were conducted using the grid based methods like finite volume and finite difference which may suffer from different numerical difficulties most importantly are the mesh deformation and mesh generation (Liu and Liu, 2003). Khanpour M. et al. (2015) applied the numerical SPH method to characterize the flow field variables downstream of a sluice gate. They developed an innovative method and introduced it to the SPH method to impose the open boundary condition to get the constant inlet discharge. However, in the present work the concept of upstream tank is adopted to simulate the flow behaviour downstream of sluice gates. This technique, which is well-known as semi boundary condition, was initially used by Ferrari (2010) and Husain et al. (2014) to simulate the flow over sharp crested weirs and stepped spillways respectively. This technique allows us to detect the flow properties passing the sluice gate for different total upstream heights at different time by fixing the water depth at the tank which decreases with time. The main objective of this research is to find the efficiency of SPH method as a numerical particle based approach in simulating and predicting the flow properties under sluice gates. Also, it looks at the flow characteristics of the hydraulic jump that would be taken place downstream of the gate. The characteristics include the conjugate depths for various Froude numbers and total heads upstream of the gate. To achieve these, the numerical 2D SPHysics code, as an implementation of the SPH method will be applied.

2. MATERIALS AND METHODS

This section is concentrated on the theory of the SPH method applied in this study. Also, it demonstrates the main features and presents the numerical tools which have been provided by the computational 2D SPHysics code adopted in this study.

2.1 Numerical Smoothed Particle Hydrodynamics Method

SPH is a particle based method initially developed by Lucy (1977) and Gingold and Monaghan (1977) in the field of astrophysics. Since its first application to the simulation of free surface flows by Monaghan (1992), numerous efforts have been made to increase the stability and accuracy of SPH computational results. Over the past two decades this method has been applied with promising outcomes in various free surface flow problems, such as flow in open channels (Lopez and Marivela, 2009; Lopez et al., 2009; Lopez et al., 2010; Federico et al. 2010; Husain, 2013; Husain et al.; 2014, Taban, 2016) and flow over hydraulic structures (Gatti et al., 2007; Lopez et al. 2009b; Ferrari, 2010). In addition, this method has gained considerable interest from industry, for instance Violeau et al. (2007); Violeau (2008) and Lee et al. (2010) for solving a range of difficult problems where the grid-based methods have great difficulty, such as mesh generation for complex geometries and non-linear large deformations (Liu & Liu, 2003).
SPH method is a purely mesh free, Lagrangian and adaptive particle method, in which a set of particles is used to represent the state of a system. Conservation equations can be applied on these particles, which carry fluid properties, and they move accordingly (Liu & Liu, 2003). The approximate expression of the integral interpolant of any function \( A(r) \) at any position \( r \) located in the space \( \Omega \) may be written as:

\[
A(r) \approx \int_{\Omega} A(r')W(r - r', h)dr'
\]  

(8)

where \( W(r - r', h) \) is the kernel function and \( h \) is the smoothing length defining the influence area of the kernel function. The value of the smoothing length is commonly taken as 1.3 of the initial distance between the particles. Equation (8) can be written in the discretized form to estimate a flow field at a desired point, \( a \), using the following summation equation that is carried out over all of the neighboring particles located within its influence area:

\[
A(r_a) = \sum_b A_b V_b W(r_a - r_b, h)
\]  

(9)

where the subscript, \( b \), refers to all of the particles residing the desired particle within an area of a circular shape, in 2D fluid flow simulations, of radius characterized by the smoothing length times a scaling factor, \( kh \), known as the compact support of the kernel function, \( V_b \) is the volume of that particle and \( A_b \) is the value of the function at position \( r_b \). Figure (2) illustrates the influence area of a given particle, \( a \), and the neighboring particles represented by, \( b \), located inside this area.

![Figure (2). Support domain of particle, a, and its neighboring particles, b, inside a circular area in 2D](image)

The volume \( V_b \) can be expressed as the relationship between the mass \( m_b \) and mass density \( \rho_b \) as follows:

\[
V_b = \frac{m_b}{\rho_b}
\]  

(10)

Substituting equation (9) into equation (8) yields:

\[
A(r_a) = \sum_b A_b \frac{m_b}{\rho_b} W(r_a - r_b, h)
\]  

(11)

The above equation can be considered as the basic expression of the SPH method which approximates a continuous vectorial or scalar field function.

### 2.2 Computational 2D SPHysics Code

Since the invention of the smoothed particle hydrodynamics method, many codes have been developed to simulate fluid flow problems. The SPHysics code as an implementation of the SPH method is applied in the current work. It 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). This code solves the Naiver-Stokes equations based on the formulations presented in Monaghan (1992) for simulating free surface flows of weakly compressible fluids (Gómez-Gesteira et al. 2012). In addition, it implements various techniques which have recently been proposed to improve some numerical issues such as consistency, accuracy and stability. The governing equations and the techniques employed by the SPHysics code are presented below.

The mass and momentum conservation equations in SPH discrete notation are respectively:

\[
\frac{dp_a}{dt} = \sum_b m_b v_{ab} \nabla_a W_{ab}
\]  

(12)

\[
\frac{dv_a}{dt} = -\sum_b m_b \left( \frac{\rho_a + \rho_b}{\rho_a^2} + \frac{\tau_a + \tau_b}{\rho_b^2} \right) \nabla_a W_{ab} + \\
\sum_b m_b \frac{4v_{a}r_{ab}w_{ab}}{(\rho_a + \rho_b)|r_{ab}|^2} v_{ab} + g
\]  

(13)

where \( \rho, m, v, W, P, \tau, v_o, r \) and \( g \) are the density, velocity, mass, kernel smoothing function, pressure, shear stress, kinetic viscosity of laminar flow, distance between two interactive particles and gravitational acceleration respectively.

Since the flow through most of hydraulic structures is extremely turbulent, the laminar viscosity and sub-particle scale turbulence method SPS is
therefore used in the current work to consider the effect of the turbulent flow field on the numerical results. This method was developed by Dalrymple and Rogers (2006) to introduce the effect of viscosity into the momentum equation. Also, the XSPH variant method (Monaghan, 1989) is adopted to prevent particle penetration through the rigid boundaries as follows:

$$\frac{d\rho_a}{dt} = v_a + \varepsilon \sum_b \frac{m_b}{\rho_{ab}} v_{ab} W_{ab}$$ \hspace{1cm} (14)

where \( \varepsilon \) is a constant \((0 \leq \varepsilon \leq 1)\) with a common value of 0.5 and \( \rho_{ab} = \frac{1}{2} (\rho_a + \rho_b) \).

Equation (14) updates the value of the velocity at the particle under consideration based on the surrounding particles located inside its influence area (Gómez-Gesteira et al. 2012). As mentioned earlier the SPH model applied in this study considers the fluid to behave as weakly compressible. This significantly reduces the computation time as the pressure term appearing in Equation (13) can be solved explicitly using an equation of state instead of solving a differential equation such as Poisson’s equation (Gómez-Gesteira et al. 2012). Batchelor (1974) and Monaghan and Kos (1999) modified the equation of state by assuming the pressure-density relationship as:

$$P = B \left[ \left( \frac{\rho}{\rho_o} \right)^\gamma - 1 \right]$$ \hspace{1cm} (15)

where \( P \) is the pressure, \( \rho_o \) is the reference fluid density (for water \( \rho_o = 1000 \text{ kg/m}^3 \)), \( \gamma \) is the polytropic constant and \( B \) is the bulk modulus elasticity of the fluid defined by:

$$B = \frac{c_o^2 \rho_o}{\gamma}$$ \hspace{1cm} (16)

where \( c_o \) is the speed of sound defined in terms of the reference density which may be found from:

$$c_o^2 = c^2(\rho) = \frac{\partial P}{\partial \rho} \bigg|_{\rho_o} = \frac{B \gamma}{\rho_o} \left( \frac{\rho}{\rho_o} \right)^{\gamma-1} \bigg|_{\rho_o}$$ \hspace{1cm} (17)

$$c_o^2 = \frac{B \gamma}{\rho_o}$$ \hspace{1cm} (18)

It should be noted here that using the actual physical value of sound speed yields exceedingly small time steps and consequently high computational time. To overcome this problem the SPHysics code utilizes an artificial sound speed based on the maximum velocity of the fluid particles. Monaghan (1994) documented that the Mach number should not exceed 0.1, which requires the minimum value for the speed of sound be 10 times greater than the largest velocity scales in the flow. This may keep the variation of the relative density, \( \rho/\rho_o \) appearing inside the state equation to within nearly 1% and consequently the compressibility effects may reasonably be neglected. In the present study the value of the artificial sound speed for each test case is used based on the maximum expected fluid velocity. Different kernel functions are available in the literature to carry out the numerical computations. The Spline function is commonly used to simulate free surface problems which can be defined from the following equation:

$$W(r, h) = \begin{cases} 1 - \frac{3}{2} q^2 + \frac{3}{4} q^3, & 0 \leq q \leq 1 \\ \frac{1}{4} (2 - q)^3, & 1 \leq q \leq 2 \\ 0, & q \geq 2 \end{cases}$$ \hspace{1cm} (19)

where,

$$\alpha_D = \begin{cases} \frac{10}{(7\pi h^2)}, & \text{in } 2D \\ \frac{1}{(5\pi h^3)}, & \text{in } 3D \end{cases}$$ \hspace{1cm} (20)

In which \( h \) is the smoothing length and \( q \) is the dimensionless distance between the particles defined by:

$$q = \frac{r}{h}$$ \hspace{1cm} (21)

where \( r \) is the distance between the particle under consideration and others located within its compact support. Differential equations appearing in the governing equations of any numerical modelling method need to be integrated numerically with respect to time. Since these equations in a particle based method are ordinary, they can be integrated in a more straightforward way than grid based methods. However, Gómez-Gesteira et al. (2010) recommended that the accuracy of the time scheme used in particle methods should be no less than the second order. The Verlet scheme is a second order time integrator, which has been commonly used in many particle based numerical simulations, is implemented in the SPHysics code. The Verlet time integrator scheme is therefore adopted in the current study. Moreover, the variable time step technique is activated in this study which periodically updates the value of the time step based on the minimum time step value provided by the forcing terms and the combined effects of the viscous diffusion term and the CFL number, (Monaghan and Kos, 1999).
In the present work the linked list technique is utilized to search for the fluid particles residing within each particle’s influence area. This procedure is performed at each time step to update particle positions and the flow field functions they carry. The reader is referred to (Monaghan and Lattanzio, 1985) for further details.

It should be noted that since the fluid is treated as weakly compressible by the SPHysics code, high pressure fluctuations may be observed with the standard SPH formulations even with a slight variation of the fluid density. Numerous efforts have been made to remove these fluctuations in the pressure field in the SPH scheme and different methods have been proposed such as Shepard, moving least squares and Riemann solver. In the present work the non-conservative Riemann technique employed by the SPHysics code is applied. This approach was initially proposed by Parshikov (1999) and later by Cha and Whitworth (2003) as the Godunov Particle Hydrodynamics method, or GPH.

Particles close to the solid boundaries and interface may produce instabilities in the numerical solution as the compactness condition may not be satisfied. Two techniques are implemented in the SPHysics code to overcome this issue; namely: kernel correction and kernel gradient correction. In the present study the kernel correction method is used. This method is used by Bonet and Lok (1999) to periodically correct the kernel function in such a way that the exact interpolation up to a given degree has to be achieved.

SPHysics code provides two techniques to model solid boundaries: dynamic (Dalrymple and Kino, 2000, Crespo et al., 2007); and repulsive boundary conditions (Monaghan, 1994). The authors observed a number of fluid particles penetrating solid walls when the dynamic boundary condition is applied. Therefore, the repulsive boundary condition is used in the present study. In the case of immobile walls this approach generates one layer of equally spaced boundary particles with zero pressure and velocity. The repulsion mechanism of this technique is based on prescribed boundary forces exerted by boundary particles in the normal direction. The reader is referred to Monaghan and Kos (1999) and Rogers and Dalrymple (2008) for more details.

It is also worth mentioning that the Lagrangian property of the SPH numerical method applied in the present study allows us to estimate the mean velocity and free surface elevations at desired points from a number of successive output files, for instance 3-5 output files.

3. RESULTS & DISCUSSION

Researchers have compared their numerical results with either the experimental observations and/or analytical results to examine the performance of the applied numerical code in predicting the physical quantities of their studies. In the present investigation the 2D SPhysics code is applied to simulate the characteristics of flow under sluice gates. In order to have confidence in the numerical SPhysics model it is important to validate it against the existing published data. To accomplish this, the results of a previously published experimental model for the flow in sluice gates are considered to evaluate the computational results obtained in this study, namely the experimental work of Lopez et al. (2010) for flow under a sluice gate.

Lopez et al. (2010) conducted their experiments to characterize the flow properties down of a sluice gate. The experimental work was performed in a horizontal and rectangular laboratory flume of 12m length, 0.6m width and 1.0m height. The gate was placed at a distance of 1.2m from the flume inlet. The gate had a sharp upstream edge, thickness = 0.03m, height = 0.30m and width matching the entire flume width. To accelerate the jump formation Lopez et al. (2010) placed a broad crested weir at the end of the flume of length 0.2m and height as the same as the gate opening. Figure (3) demonstrates the initial condition for the flow under a sluice gate physical model tested experimentally by Lopez et al. (2010). It is worth mentioning here that in the current work the concept upstream tank is used to represent the fluid particles stored at the upstream and to detect the flow field variables in the flow domain. Although this approach significantly increases the computational time it is used with successful outcomes in a number of free- surface fluid flow cases, namely; Ferrari (2010) for simulating the flow over sharp crested weirs; Husain (2013) Husain et al. (2014) for the flow over broad crested weirs and stepped spillways and Taban (2016) for the flow over stepped spillways.
Normally, the flow condition upstream of sluice gates is subcritical as the flow depth is relatively high and the flow velocity is low, whereas it passes beneath the gate with high velocity with high level of turbulence and hydraulic jump may form at a distance somehow downstream of the gate, depending on the gate opening, due to the change of flow condition to the super critical condition (Chow, 1959; Henderson, 1966; Chanson, 2004).

Figure (3): 2D schematic view of the experimental laboratory model used by Lopez et al. (2010) for flow under sluice gates

Figures (4.a) to (4.d) illustrate snapshots depicted by the numerical 2D SPHysics code of flow conditions for the flow under the sluice gate at different time instants. These snapshots demonstrate that the computational results provided by the code are qualitatively in good agreement with the description of flow condition while passing the gate. This indicates that the numerical SPHysics code well captures the physical behavior of the flow under such hydraulic configuration.

Now the quantities of the flow field variables predicted by the numerical SPHysics code will be presented and compared with the experimental and theoretical data. This includes the position of the conjugate depths of the hydraulic jump for various Froude numbers.

Table (1) shows the comparison between the theoretical values of the of the flow depths and flow velocities before $y_2theo, V_2theo$ and after the jump $y_3theo, V_3theo$ and the corresponding ones of the computational results of this work obtained from numerical SPHysics code for different total upstream heads $y_1$. The theoretical values of the flow depths are estimated from using equations 6 and 7 respectively, while the continuity equation, equation 8, is used to calculate the theoretical values of the flow velocities before and after the jump.

However, as mentioned earlier in this investigation the position of the highest particle in a given section is taken as the computational flow depth at that section. Also, the mean velocity of the velocity profile in a given section is considered as the flow velocity at that section. As can be observed the agreement between the theoretical and computational flow depths and velocities is fairly good for all total upstream head values tested in this work as the differences between them are relatively small.
Table (1): Comparison between theoretical and numerical values of flow depths and velocities before and after the jump for various total upstream heads

| $y_1$ | $y_2$ | $y_2^{\text{SPHysics}}$ | $V_2$ | $V_2^{\text{SPHysics}}$ | $y_3$ | $y_3^{\text{SPHysics}}$ | $V_3$ | $V_3^{\text{SPHPhysics}}$ |
|-------|-------|-------------------------|-------|-------------------------|-------|-------------------------|-------|-------------------------|
| 0.174 | 0.0114 | 0.0118                  | 1.870 | 1.595                   | 0.0846 | 0.0562                  | 0.2520 | 0.3266                  |
| 0.165 | 0.0114 | 0.0115                  | 1.815 | 1.483                   | 0.0820 | 0.0546                  | 0.2523 | 0.3057                  |
| 0.155 | 0.0114 | 0.0112                  | 1.760 | 1.474                   | 0.0793 | 0.0653                  | 0.2530 | 0.2453                  |
| 0.146 | 0.0114 | 0.0110                  | 1.698 | 1.352                   | 0.0764 | 0.0637                  | 0.2534 | 0.2288                  |

Figure (5) presents the velocity flow field predicted by the SPHysics code at the time instants measured experimentally by Lopez et al. (2010).

Table (2) presents the comparison between the experimental values recorded by Lopez et al. (2010) and the computational results obtained in this study in terms of the flow depths and velocities for various total upstream heads tested in the current work. The flow depths were experimentally measured by Lopez et al. (2010) using digital point gauges and the flow velocities were measured by an advanced pitot tube. It is clear from this Table that the computational results obtained in this study are close to the corresponding experimental values revealing the capability of the SPHysics code as an implementation of the SPH method to efficiently simulate and estimate the flow under sluice gates.

The following section presents and discusses the effect of various flow parameters on the energy dissipation for the flow under sluice gates. These include the relative ratio of gate opening with respect to the total upstream head and Froude number. Also, focuses on the variation of the position of free surface with the Froude number values tested in this study.

Figure (6) is the plot of the variation of the relative gate opening with respect to the total upstream head $y_1/a_0$ with the percentage of the
amount of energy dissipation through the jump \(100(E_2 - E_3)/E_2\) caused by the sluice gate. In this figure \(y_1\) is the total upstream head, \(a_o\) is the gate opening, which is fixed to 0.02m through the whole work of this project, \(E_2\) and \(E_3\) are the energy upstream and downstream of the jump. As it can be seen from this figure the energy dissipation increases as the relative ratio of the gate opening is increased. This indicates that the energy dissipation rate is high when the total head of water stored upstream of the sluice gate is high. This can be attributed to the effect of the flow turbulence, which is considered to have significant role on the energy dissipation in hydraulic structures (Chanson, 2004), which may be increased as the upstream head is high.

### Table (2): Comparison between the experimental and numerical values of flow depths and velocities before and after the jump for various total upstream heads

| \(y_1\) | \(y_2\) \(exp.\) | \(y_2\) \(SPHysics\) | \(V_2\) \(exp.\) | \(V_2\) \(SPHysics\) | \(y_3\) \(exp.\) | \(y_3\) \(SPHysics\) | \(V_3\) \(exp.\) | \(V_3\) \(SPHysics\) |
|--------|----------------|----------------|----------------|----------------|----------------|----------------|----------------|----------------|
| 0.174  | 0.0120         | 0.0118         | 1.614          | 1.595          | 0.0573         | 0.0562         | 0.3371         | 0.3266         |
| 0.165  | 0.0117         | 0.0115         | 1.506          | 1.483          | 0.0552         | 0.0546         | 0.3165         | 0.3057         |
| 0.155  | 0.0113         | 0.0112         | 1.494          | 1.474          | 0.0661         | 0.0653         | 0.2609         | 0.2453         |
| 0.146  | 0.0111         | 0.0110         | 1.375          | 1.352          | 0.0642         | 0.0637         | 0.2451         | 0.2288         |

### Figure (6): Variation of energy dissipation rate with the relative gate opening with respect to the total upstream head

Variation of the percentage of the energy dissipation \(100(E_2 - E_3)/E_2\) with Froude number due to the pre-jump flow depth \(Fr_2\) is plotted in Figure (7). One can notice that the energy dissipation increases as the Froude number is increased. The increase of the Froude number is associated with the increase of the total head of water stored upstream of the sluice gate.

### Figure (7): Variation of energy dissipation rate with the pre-jump Froude number

In Figure (8) the variation of pre-jump Froude number \(Fr_2\) is plotted versus the percentage of the height of the hydraulic jump \(H_j\), which can be defined as relative of the difference between the conjugate depths \(H_j = 100(y_3 - y_2)/y_3\) to the flow depth after the jump. This figure shows that the height of the hydraulic jump increases with the decrease of the Froude number.
4. CONCLUSION

The upstream tank approach is adopted in this study to examine the capability of the numerical 2D SPHysics code, as an implementation of the Smoothed Particle Hydrodynamic method, in detecting the flow field variables in sluice gates qualitatively and quantitatively. Also, the effect of flow parameters such as upstream flow depth, gate opening and Froude number on the amount of energy dissipation downstream of the gate is investigated. For the sake of validating the numerical code, the results obtained in this study are compared with the theoretical as well as the experimental results gathered in this type of hydraulic structure. The results produced by the numerical code are in close agreement with the corresponding theoretical and experimental results. Also, it is observed that the differences between them are insignificant for different total heads upstream of the gate. This is an indication that the computational code is efficient in predicting the flow properties in free surface flow problems. It is also found that the percentage of the total amount of energy dissipation may increase linearly with the increase of the ratio of the flow depth to the gate opening and Froude number. Conversely, the jump length could decrease linearly with the increase of the Froude number. Despite the fact that the numerical code applied in this study is performed well in simulating the flow in sluice gates, it is still needed to be applied in other free-surface flow problems to prove its validity.

REFERENCES

Batchelor, G., K. (1974). Introduction to fluid mechanics, UK, Cambridge University press.

Bonet, J. and Lok, T. S. (1999). Variational and momentum preservation aspects of Smooth Particle Hydrodynamics. Journal of Computer Methods in Applied Mechanics and Engineering, 180 (1), 97-115.

Cha, S.-H., and Whitworth, A. P. (2003). Implementations and tests of Godunov-particle hydrodynamics, Mon. Not. R. Astro. Soc., 340, 73-90.

Chanson, H. (2004). The hydraulics of open channel flow: an introduction, 2nd edition, Oxford, UK, Butterworth-Heinemann.

Chow, V., T. (1959). Open channel hydraulics, New York, USA, McGraw-Hill.

Crespo, A. J. C., Gómez-Gesteira, M and Dalrymple, R. A. (2007). Boundary conditions generated by dynamic particles in SPH methods. Journal of Computers, Materials and Continua, 5(3), 173-184.

Dalrymple, R. A. and Kino, O. (2000). SPH modelling of water waves. Proc. Coastal Dynamics, Lund.

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

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

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

Gatti, D., Maffio, A., Zuccala, D. and Di Monaco, A., (2007). SPH Simulation of Hydrodynamics Problems Related To Dam Safety. 2007. 32nd Congress of IAHR, CD-ROM. Paper 2, SS09. Venice. Italy.

Gingold, R. A., and Monaghan, J. J. (1977). Smoothed particle hydrodynamics, theory and application to non-spherical stars, Mon. Not. Roy. Astr. Soc., 181, 375-389.

Gómez-Gesteira, M., Rogers, B., Dalrymple, R. A. and Crespo, A. J. C. (2010). State-of-the-art of classical SPH for free-surface flow. Journal of Hydraulic Research, 48, Extra issue, 6-27.
Husain .S. et al / ZJPAS: 2018, 30(s1): s94-s104

Gómez-Gesteira, M., Rogers, B., Dalrymple, R. A., Crespo, A. J. C., and Narayanaswamy, M., (2012). User Guide for the SPHysics Code v2.0, http://www.sphysics.org.

Husain, M., Sarhang, (2013). Computational investigation of skimming flow over stepped spillways using the smoothed particle hydrodynamics method. PhD thesis, university of Swansea, Swansea, UK.

Husain, S. M., Muhammed, J. R., Karunarithna, H. U., & Reeve, D. E. (2014). Investigation of pressure variations over stepped spillways using smooth particle hydrodynamics, Advances in Water Resources, 66, 52-69.

Henderson, F. M., (1966). Open Channel Flow. New York, Macmillan Publishing Co.

Khanpour, M., Zarrati, A.R. and Kolahd, (2014). Numerical simulation of the flow under sluice gates by SPH model. Scientia Iranica. Transaction A, Civil Engineering, 21(5), 1503.

Lee, E. S., Violeau, D., Issa, R. and Ploix, S. (2010). Application of weakly compressible and truly compressible SPH to 3-D water collapse in waterworks, Journal of Hydraulic Research, 48(extra issue), 50-60.

Liu, M. B. and Liu, G. R. (2003). Smoothed Particle Hydrodynamics-A Mesh free Particle Method, Singapore, World scientific.

Lopez, D., Marivela, R. (2009). Applications of the SPH model to the design of fishways. 33rd Congress of IAHR, Water Engineering for a Sustainable Environment (Vancouver 9-14 de agosto de 2009). ISBN: 978- 90-78046-08-0.

Lopez, D., Marivela, R. and Garrote, L. (2009b). Calibration of SPH using prototype pressure data from the stilling basin of the Villar del Rey dam, Spain, 33rd Congress of IAHR, Water Engineering for a Sustainable Environment (Vancouver 9-14 de agosto de 2009). ISBN: 978- 90-78046-08-0.

Lopez, D., Marivela, R. and Garrote, L. (2010). Smooth particle hydrodynamics model applied to hydraulic structures: A hydraulic jump test case, Journal of Hydraulic Research, 48, Extra Issue (2010), 142-158. ISSN: 0022-1686.

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

Monaghan, J. J. (1989). On the problem of penetration in particle methods. Journal of computational physics, 82(1), 1-15. Monaghan, J. J. 1992. Smoothed Particle Hydrodynamics, Annual Review of Astronomy and Astrophysics, 30(1), 543-574.

Monaghan, J. J. (1994). Simulating free surface flows with SPH. Journal of computational physics, 110, 399-406.

Monaghan, J. J. and Kos, A. (1999). Solitary waves on a Cretan beach. Journal of waterway, port, coastal, and ocean Engineering, 125(3), 145-154.

Monaghan, J. J., and Lattanzio, J. C. (1985). A refined particle method for astrophysical problems. Journal of Astronomy and Astrophysics, 149 (1), 135-143.

Parshikov, A. N. (1999). Application of a solution of the Riemann problem to the SPH method. Computational Mathematics and Mathematical Physics, 39, 1173.

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

Taban K., Hamad (2016). Hydraulic performance evaluation of Bastora dam spillway. MSc thesis, Salahaddin university, Erbil, Iraq.

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

Violeau, D., Buvat, C., Abed-Meraim, K. and de Nanteuil, E. (2007). Numerical modelling of boom and oil spill with SPH, Journal of Coastal Engineering, 54(12), 895-913.

Violeau, D., Issa, R., Benhamadouche, S., Saleh, K., Chorda, J. and Maubourguet, M.-M. (2008). Modelling a fish passage with SPH and Eulerian codes: The influence of turbulent closure, Proc. 3rd SPHERIC International Workshop, 3rd–6th June 2008, Lausanne, (Switzerland), 85-91.