https://dx.doi.org/10.22161/ijaers.4.12.31 ISSN: 2349-6495(P) | 2456-1908(O)

Validation of Dam-Break Problem over Dry Bed using SPH

Kaan Dal[1], Stefania Evangelista[2], Ada Yilmaz[3], Selahattin Kocaman[4*]

[1,3,4]Department of Civil Engineering, Iskenderun Technical University, Iskenderun, Turkey
[2]Department of Civil and Mechanical Engineering, University of Cassino and Southern Lazio, Cassino (FR), Italy
*Corresponding Author

***Abstract*—*In this study, a comparison was made between experimental and numerical analysis results using the classical dam-break test case over dry bed. DualSPHysics software based on Smoothed Particle Hydrodynamics (SPH) method was used to make the numerical analysis. Experimental data were obtained from Kocaman [1]'s laboratory setup through image processing technique. It was observed that the numerical and experimental results are in good agreement.*

***Keywords*— Dam-Break, SPH, Flood Wave, Experimental Study.**

I. INTRODUCTION

Dams can be built to supply potable water, generate electricity and control flood. Since the massive volume of water may be found upstream, the dams pose risks for the residential areas located downstream. In case dam-break case, the massive volume of water can create a flood and this can cause many losses of life and property. The flood caused by the dam-break is more destructive than a rain induced flood. Agricultural and settlement areas are often those that have experienced floods caused by dam-breaks in the past. Investigating the dam-break problem scientifically is remarkable due to insufficient data of the eventuated dam-break cases. This and this kind of study aims at combining experimental data and numeric analysis to predict the areas that will be affected by the flood and contribute to the improvement of the early warning systems.
Due to difficulties in obtaining real case-data, and although the hydraulic problems are large scale in the field, the data of the numerical analysis can be validated through small-scale experimental studies [2-5]. Most of the early numerical studies used Shallow Water Equations (SWE) to describe dam-break wave propagation [6-8]. Due to recent developments in computer power, it is possible to widely use numerical methods based on Computational Fluid Dynamics (CFD). Some researchers used SWE and Reynolds Averaged Navier-Stokes (RANS) equations to investigate dam-break flow in 2D and 3D [9,10].
Smoothed Particle Hydrodynamics (SPH) method is one of the most commonly used numerical methods in Hydraulics. The method has become popular recently. It was developed to investigate the astrophysics cases at the middle of the 1970s [11].
SPH method is still developing and it is frequently used in the modelling of various engineering problems such as hydraulic [12,13], structural dynamics [14] and solid mechanics [15]. One of the common software used to model the problems is DualSPHysics. This freeware software is used to study free-surface flow phenomena where Eulerian methods can be difficult to apply, such as dam-breaks, sloshing, wave impacts on off-shore structures [16-17].
In this study, the numerical and experimental results were compared for the dam-break flow over a dry bed at initial stages. The comparison was made through water surface profile curves. The numerical results were obtained from the SPH method, whereas experimental results were determined using image processing technique.

II. EXPERIMENTAL SET-UP AND NUMERICAL MODEL

2.1. Experimental Set-up

The experimental studies were carried out at Civil Engineering Hydraulics Laboratory of the Cukurova University, Turkey. The test setup was conducted by Kocaman [1]. Experimental studies were performed in a rectangular channel. Lateral and bottom surfaces are glass. Dimensions of the channel are 8.90 m x 0.3 m x 0.35 m (length, width, height, respectively). The plate representing the dam was placed at 4.65 m from the channel entrance and this part was considered as a reservoir. The water depth in the reservoir was taken 0.25 m. A red dye was added into the reservoir to better determine the free surface levels from recorded images. The downstream side of the channel is kept dry at the beginning of the experiment. The mechanism

https://dx.doi.org/10.22161/ijaers.4.12.31 *ISSN: 2349-6495(P) | 2456-1908(O)*

illustrated in Figure 1 allowed for the instantaneous removal of the vertical plate to model the dam-break. The 4-mm thick plate was used as a dam. A steel rope was attached to the plate top; the rope was drawn over a pulley of 15 kg weight attached at the other end. By releasing the weight from 1.50 m above the floor, the plate was removed instantaneously. The plate removal time was determined as approximately 0.07 s from video records.

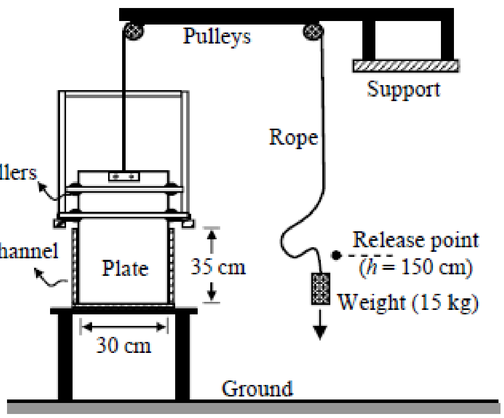


Fig.1: Test setup and dam-break mechanism [18]

Experimental results were obtained through image processing technique. Three adjacent cameras were simultaneously used to record the experiment. The recorded videos were calibrated to obtain the water surface profiles. The cameras are JVC c920e with acquisition rate 25 frames per second and resolution 752x586 pixels. A complete description of the experimental setup and measurement techniques can also be seen in Ozmen-Cagatay and Kocaman [9-18].

2.2. Numerical Model

Dual SPHysics is a numerical model based on the SPH method. This software is capable to study real engineering problems by using graphical processing units (GPU) [16]. In this model, fluid, boundary and solid bodies are represented with particles. The interaction of the neighboring particles depends on the distance between the particles. A kernel function (W) defines to calculate the interaction, which is affected by smoothing length (h). A particle is involved in an interaction with the particles located in a circular area, which size depends on the smoothing length. There is no interaction between particles placed out of the circular area. Physical quantities such as pressure, location, density, velocity etc. are calculated through interpolation depending on the smoothing length. The numeric foundation of the SPH are based on the interpolation of integrals. An F function is defined as shown in Equation 1 [12]:

$$F(r) = \int F(r')\, W\, (r - r', h) dr \qquad (1)$$

where *F(r)* is the average, *W* is the kernel function, *h* is the smoothing length. If this function is interpolated in an "*a*" particle, Equation 2 is obtained;

$$F(r_a) \approx \sum_b F(r_b)\; W(r_a - r_b, h)\Delta v_b \qquad (2)$$

where, subscripts *a* and *b* are individual particles. Δv_b is the volume of the neighboring particle b. Since the volume is the ratio of the density to mass Equation 2 becomes [19]

$$F(r_a) \approx \sum_b F(r_b) \left(\frac{m_b}{\rho_b}\right) W(r_a - r_b, h) \qquad (3)$$

where, m_b and ρ_b are the mass and the density of the particle b respectively.

The accuracy of an SPH analysis depends on the kernel function. The function is defined as $q=r/h$. Here, r is the distance between two particles and q is the dimensionless ratio. The definition of the Wendland kernel used in this study is given in Equation 4:

$$W(r, h) = \alpha_D \left(1 - \frac{q}{2}\right)^4 (2q + 1) \qquad (4)$$

where α_D value is $7/4\pi h^2$ for 2D analysis while, $21/16\pi h^3$ for the 3D analysis.

In the SPH formulation, the Navier-Stokes equations are solved based on the assumption of weakly compressible fluid [20]. Momentum equation is used to define the acceleration for an "a" particle with interaction of the neighboring particles:

$$\frac{dv_a}{dt} = -\sum_b m_b \left(\frac{P_b + P_a}{P_b - P_a} + \Pi_{ab}\right) \nabla \frac{1}{\rho} \nabla_a W_{ab} + g \qquad (5)$$

where, v is velocity, P is pressure, g is the gravity and W_{ab} is kernel function. Π_{ab} is a term based on the artificial viscosity created by Monaghan [20]

Since the mass of the particles is constant, the conservation of mass or continuity equation is used to measure the density variations:

$$\frac{d\rho_a}{dt} = \sum_b m_b v_{ab} . \nabla_a W_{ab} \qquad (6)$$

The relationship between pressure and density is given by the Tait's equation of state, shown in Equation 7 [21]:

$$P = b\left(\frac{\rho}{\rho_0}\right)^\gamma - 1 \qquad (7)$$

$$b = c_0^2 \rho_0 / \gamma \qquad (8)$$

where $\gamma = 7$ and $\rho_0 = 1000$ kg/m^3.

https://dx.doi.org/10.22161/ijaers.4.12.31 **ISSN: 2349-6495(P) | 2456-1908(O)**

The speed of sound c_o is numerically computed like at least ten times the maximum velocity in the system. For the dam-break flow the wave-front velocity was used as follows:

$$c_0 = c_s\sqrt{gh_w} \tag{9}$$

where c_s is the coefficient sound, h_w is the still water depth. If the value of c_s is increased, compressibility of the fluid is decreased according to Equations 7-8.

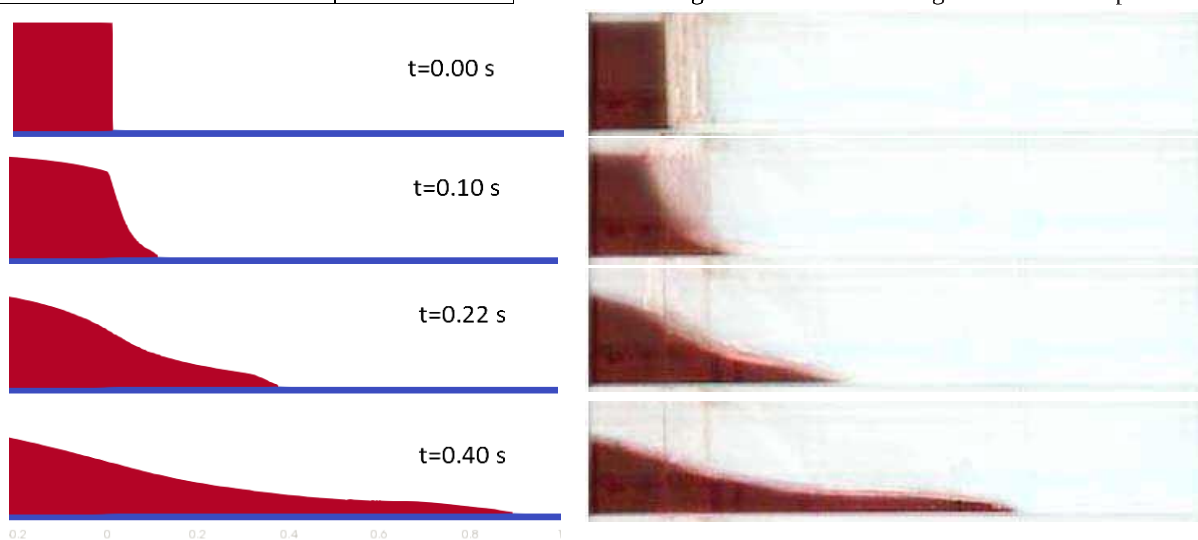
The model parameters set in the DualSPHysics software for the SPH method to make the numerical analysis. are reported in Table 1:

Table.1: Parameters of the numeric model

Artificial viscosity	0.01
Courant coefficient	0.2
Coefficient Sound	20
Distance of the particles	0.002m
DeltaSPH value	0.1
Number of the particles	173650
Smoothing length	0.002828
Step Algorithm	Symplectic
Kernel	Wendland

III. VALIDATION OF THE NUMERICAL MODEL

Comparisons between the SPH numerical and experimental free surface profiles at initial stages can be seen in Figure 2. It is observed that when the plate moves above, the water volume in the upstream rapidly propagates towards downstream. Parabolic wave front at the initial stage becomes hump shaped as time progressed with surface roughness. It can be understood that the surface roughness affects the results at the lately stages more than initial stages. The SPH results agree well with experiment.



(a) (b)

Fig.2: Comparison of free surface profiles by: a) SPH b) experiment

Figure 3 shows the comparison between the computed free surface profiles using the SPH and the experimental data at different times. The flow depths *h* and horizontal distances *x* were non-dimensionalized by the initial water depth h_0. Similarly, time t was multiplied by $(g/h_0)^{1/2}$ to obtain dimensionless time. A good agreement is observed between the measured and SPH-computed free surface profiles at all times. However, the wave-front velocities determined by the SPH model are always higher than experiment except at very early stage (T=1.13).

It was observed that slight changing of some parameters (number of the particles, distance of the particles, artificial viscosity, smoothing length etc.) have remarkable effects on the numerical results. Especially, the particle resolution in the SPH has a great impact on the accuracy and stability. Reducing the distance of the particles causes an increase in the number of the particles. For this reason, solving abilities of the computer becomes insufficient. High-performance computing (HPC) needs to be analyzed for the numerical model with a large amount of particles. However, GPU accelerated computing gives advantages to the SPH method compared to other CFD based numerical methods. DualSPHysics has the GPU option which provides a higher computing power than CPU in order to accelerate SPH simulation with low cost. In this study, runtime of the problem took 556 s for 1.5 simulation time by using Geforce GTX 580 card with 3GB RAM. It was also considered that the variations of the parameters should be investigated in further studies by using different experimental data having obstacles, wave impacts, etc.

https://dx.doi.org/10.22161/ijaers.4.12.31 **ISSN: 2349-6495(P) | 2456-1908(O)**

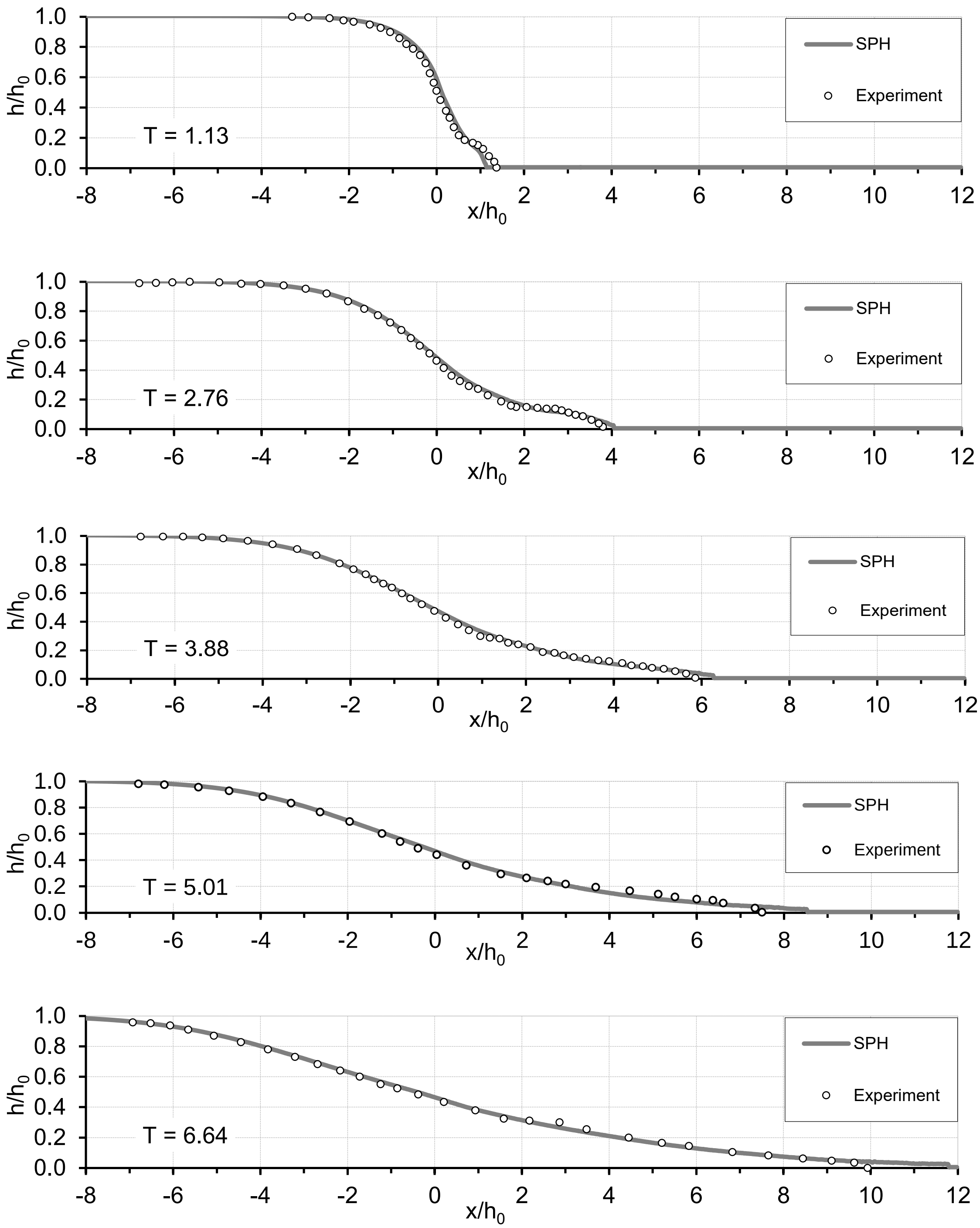


Fig.3: Experimental and numeric results of water surface profiles

IV. CONCLUSIONS

In this study, the flood wave resulted from dam-break phenomenon over a dry bed at initial stages was examined by comparing the SPH based numerical model with experimental one. A very good agreement was observed for free surface profiles. However, it was found that wave-front velocities of the SPH model is higher than the experimental one.

Since the results are in good agreement, it can be concluded that DualSPHysics based on the SPH method can be utilizable and useable to analyze the chaotic hydraulic problem such as dam-break. However, in order to be able to apply large-scale real-life problems, further validation is required with experimental studies representing different topography conditions.

REFERENCES

[1] Kocaman, S. (2007). Experimental and theoretical investigation of dam-break problem. Ph. D. Thesis. Cukurova University, Adana. (in Turkish)

[2] Ozmen-Cagatay, H., & Kocaman, S., (2012). Investigation of dam-break flow over abruptly contracting channel with trapezoidal-shaped lateral obstacles. Journal of Fluids Engineering, 134(8), 081204.

[3] Ozmen-Cagatay, H., Kocaman, S. & Guzel, H. (2014). Investigation of dam-break flood waves in a dry channel with a hump. *Journal of Hydro-environment Research* 8(3): 304-315.

[4] Aureli, F., Maranzoni, A., Mignosa, P., & Ziveri, C., (2008), Dam-Break Flows: Acquisition of Experimental Data Through an Imaging Technique and 2D Numerical Modeling. *J. Hydraul. Eng.*, 134(8), 1089–1101.

[5] Ozmen-Cagatay, H., & Kocaman, S., (2011). Dam-break flow in the presence of obstacle: experiment and CFD Simulation. *Eng. Appl. Comput. Fluid Mech*. 5(4), 541-552.

[6] Mohapatra, P.K., & Bhallamudi, S.M., (1996). Computation of a dam-break flood wave in channel transitions. *Adv. Water Resour.* 19(3), 181-187.

[7] Garcia-Navarro, P., Fras, A., & Villanueva, I., (1999). Dam-break flow simulation: some results for one-dimensional models of real cases. J. Hydrol. 216(3-4), 227-247.

[8] Evangelista S., Altinakar M., Di Cristo C., & Leopardi A. (2013). Simulation of dam-break waves on movable beds using a multi-stage centered scheme. *International Journal of Sediment Research*, 28(3), 269-284, ISSN: 1001-6279.

[9] Kocaman, S., & Ozmen-Cagatay, H. (2015). Investigation of dam-break induced shock waves impact on a vertical wall. *Journal of Hydrology*, 525, 1-12.

[10] Kocaman, S., & Ozmen-Cagatay, H., (2012). The effect of lateral channel contraction on dam break flows: Laboratory experiment. *Journal of hydrology*, 432, 145-153.

[11] Gingold, R.A. & J.J. Monaghan, (1977). Smoothed particle hydrodynamics-theory and application to non-spherical stars,. *Monthly Notices Of The Royal Astronomical Society*, 181: 375-389.

[12] Altomare, C., Oshima, Y., Chen, X., Crespo, A.J.C., & Suziki, T., (2015). Study of the Overtopping Flow Impacts on Multifunctional Sea Dikes in Shallow Foreshores with an Hybrid Numerical Model, *In 36th IAHR World Congress*, The Hague, The Netherlands.

[13] Cunningham, L.S., Rogers, B.D., & Pringgana, G., (2014). Tsunami wave and structure interaction: An investigation with Smoothed Particle Hydrodynamics, *ICE Engineering and Computational Mechanics*, 167:126-138.

[14] Canor, T. & V. Denoël, (2013). Transient Fokker-Planck-Kolmogorov equation solved with smoothed particle hydrodynamics method,. *International Journal for Numerical Methods in Engineering*, 94(6): 535-553.

[15] Randles, P.W. & L.D. Libersky, (1996). Smoothed particle hydrodynamics: Some recent improvements and applications. *Computer Methods in Applied Mechanics and Engineering*, 139(1-4): 375-408.

[16] Crespo, A.J.C., Dominguez, J.M., Rogers, B.D., Gesteira, M.G., Longshaw, S., Canelas, R., Vacondio, R., Barreiro, A., & Feal, O.G., (2015). DualSPHysics:Open-Source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH). *Computer Physics Communications*. 187:204-216.

[17] Shao, J. R., Li, H. Q., Liu, G. R., & Liu, M. B., (2012). An improved SPH method for modeling liquid sloshing dynamics. *Computers & Structures*, 100, 18-26.

[18] Ozmen-Cagatay, H., & Kocaman, S., (2010). Dam-break flows during initial stage using SWE and RANS approaches. *J. Hydraul. Res.* 48(5), 603-611.

[19] Monaghan JJ & Lattanzio JC., (1985). A Refined Method for Astrophysical Problems. *Astron. Astrophys*, 149, 135–143

[20] Monaghan J.J., (1992). Smoothed Particle Hydrodynamics. *Annual Review of Astronomy and Astrophysics*, 30, 543-574.

[21] Users Guide for DualSPHysics code, (2016).