

## NUMERICAL MODEL FOR DAM BREAK OVER A MOVABLE BED USING FINITE VOLUME METHOD

\*Ikha Magdalena<sup>1</sup>, Mohammad Bagus Adityawan<sup>2</sup> and Charles Jonathan<sup>3</sup>

<sup>1,3</sup>Faculty of Mathematics and Natural Sciences, Institut Teknologi Bandung, Indonesia; <sup>2</sup>Water Resources Engineering and Management, Institut Teknologi Bandung, Jatinangor, Indonesia

\*Corresponding Author, Received: 14 Oct. 2019, Revised: 24 Jan. 2020, Accepted: 04 March 2020.

**ABSTRACT:** In this paper, we study dam break phenomenon over a movable bed through a mathematical model. The model dynamically links the hydrodynamic and sediment transport. To solve the model numerically, we use finite volume method on a staggered grid that is simpler than the generalized collocated finite volume method. This is because the Riemann invariants problem in collocated grid approach can be particularly complex and costly in simulating dam break floods with sediment transport. The quality of the dam break flood simulations with our numerical scheme is verified by comparing the results against analytical solutions, laboratory tests and some experimental data available in the literature on fixed and mobile bed conditions. The numerical results reproduce the experimental evidence quite well, proving that the model is capable of predicting the temporal evolution of the free-surface and the bed. Our numerical scheme, which is not only simple to implement but also both accurate and computationally efficient, is proposed as an appropriate tool for simulating dam break problem over a movable bed.

**Keywords:** *Dam break, Movable bed, Finite volume method, Sediment transport.*

### 1. INTRODUCTION

Dam break is an event where a massive volume of water is suddenly released due to the dam failure. Dam failure can be caused by several factors such as flood, earthquake and other factors. The event may leads to a catastrophic disaster to the downstream of the dam. The dam break induced flow may leads to a severe destruction along its flow path. In general, the generated flow shares a common characteristic with tsunami waves. The water depth and velocity are high. This may also leads to the surface erosion along the flow path.

There have been many studies on dam break flow. Yakti et al. [1] has shown the damaging effect of a dam break to its downstream area. They have analysed the overland flow, generated by the Wai Ela Dam failure in 2013. The event nearly destroyed the whole Negeri Lima Village, Ambon, Indonesia. Satellite images and aerial photo also showed severe erosion due to the flow. However, their study did not cover the erosion profile.

The dam break generated flow is generally simulated using the Shallow Water Equations. Finite difference, finite element, or finite volume scheme are used to solve the set of equations. Finite difference scheme is widely used due to its simplicity. Yet, the scheme is less flexible than the others. Thus, its application to a real case scenario may be limited. In addition, the lower order scheme may suffer from instability due to shock waves. Hence, it requires extra term to achieve stability [2]. Finite element provides a better flexibility in terms of grid. Therefore, it may cover a wide range of

application. Nevertheless, it is temperamental in terms of stability in handling shock waves [3, 4]. The finite volume has both grid flexibility and simplicity. In general, it also has a faster computation time than the others. Therefore, in this research, we propose finite volume method to solve the mathematical model. Peng [5] has developed a model to simulate an experimental dam break case. His experiment also includes buildings, situated downstream of the dam. Thus, the effect of the dam break generated flow to these buildings were also observed. The shock waves were captured using a Total Variation Diminishing (TVD) scheme. TVD is commonly used to capture shock [6, 7]. There are other ways to capture shock in finite volume scheme without the need to use a higher order method. Shock capturing in finite volume scheme may also utilize slope limiter function [8, 9, 10]. Pudjaprasetya and Magdalena [11] employed a momentum conservative scheme to handle shock and discontinuity. Moreover, I. Magdalena et. al [12] have investigated radial dam break problem using finite volume method on a staggered grid.

There are limited studies on the dam break induced erosion. Spinewine and Zech [13] have studied the phenomenon by conducting a laboratory experiment in a flume. The bed of the flume was covered with sediment grain. The flume was separated by a gate. The upstream of the gate was filled with water. The dam break was simulated by a sudden opening of this gate. They investigated the velocity profile and sediment grain concentration as the flow propagates. Spinewine and Capart [14] updated the experiment by using a longer flume and

enhancing the gate opening. They also provided a method to estimate the sediment movement.

In this study, a model is developed to simulate a dam break flow and the generated erosion. The model is based on the shallow water equations, solved using a finite volume method with a momentum conservative scheme. It is coupled with a sediment transport equation, thus enhancing its capability for simulating erosion. The model is applied to an experimental dam break case with movable bed.

## 2. GOVERNING EQUATIONS

In this section, the governing equations of transport sediment due to the movement of a fluid in contact with a sediment layer will be discussed. The equation is based on coupled shallow water-Exner equations where Shallow Water Equations simulate the flow of water and Exner equation for movement of sediment layer. The coupled Shallow Water-Exner equations in one-dimensional case read as

$$\partial_t h + \partial_x(hu) = 0, \quad (1)$$

$$\partial_t(hu) + \partial_x\left(hu^2 + \frac{1}{2}gh^2\right) + gh\partial_x(z - H) + S_f = 0, \quad (2)$$

$$\partial_t z + \zeta \partial_x Qs = 0, \quad (3)$$

with  $h(x, t)$  is total water depth,  $u(x, t)$  is horizontal velocity,  $z(x, t)$  is sediment elevation,  $g$  is gravitational acceleration,  $S_f$  is friction term and sediment,  $Qs$  is sediment discharge, and  $H$  is depth of bedrock layer to reference level. The coefficient  $\zeta$  is computed by  $\zeta = (1 - \sigma)^{-1}$ , where  $\sigma$  is bed sediment porosity. In natural systems, the good value of  $(1 - \sigma)$  range for 0.45 to 0.75, so we let  $\sigma = 0.3$ .

To model the friction term  $S_f$  and sediment discharge  $Qs$ , we use some of most used empirical formulas. In most of cases, the flow is turbulent. Therefore, we use Manning's friction law read as

$$S_f = \frac{k^2 u |u|}{h^3}, \quad (4)$$

where  $k$  is Manning's roughness coefficient. For the bedload  $Qs$ , we will use Grass Formula. This formula is proposed by Grass for solid transport discharge read as

$$Qs = A_g |u|^{m-1} u, \quad (5)$$

where  $0 \leq A_g \leq 1$  and  $1 \leq m \leq 4$  are based on experimental data. Letting  $A_g$  be close to 1 shows

there is strong interaction between fluid and sediment. We currently use value  $m = 3$ .

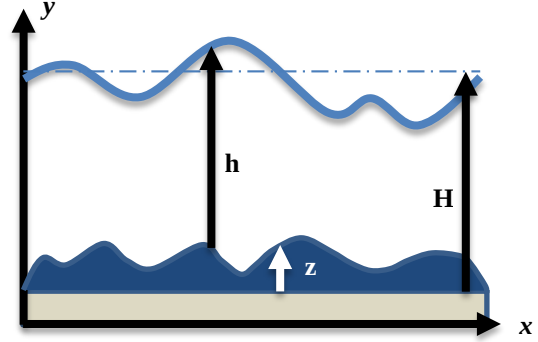


Fig. 1. The domain of this work.

## 3. NUMERICAL APPROXIMATION

In this section, we solve Eqs. (1-3) numerically using finite volume method on a staggered grid. The time interval  $[0, T]$  is divided into  $N_t$  time steps of length  $\Delta t$  and for all  $n \in \{0, \dots, N_t\}$ ,  $t^n = n\Delta t$ . Consider a spatial domain  $[0, L]$  with a staggered grid partition  $x_1 = 0, x_1, x_{\frac{3}{2}}, x_2, \dots, x_{Nx}, x_{Nx+\frac{1}{2}} = L$ . Following the staggered momentum conservative scheme for the Shallow Water Equations as seen in [11]. By modifying, we can implement an analogous scheme for governing equations (1), (2), and (3). The values of  $h$  and  $z$  are computed at every full grid points  $x_j = j\Delta x$ , with  $j = 1, 2, \dots, Nx$ . Whereas velocity  $u$  is computed at every staggered grid points  $x_{j+\frac{1}{2}} = (j + \frac{1}{2})\Delta x$ , with  $j = 0, 1, 2, \dots, Nx$ . This space of discretization is shown in Fig. 2.

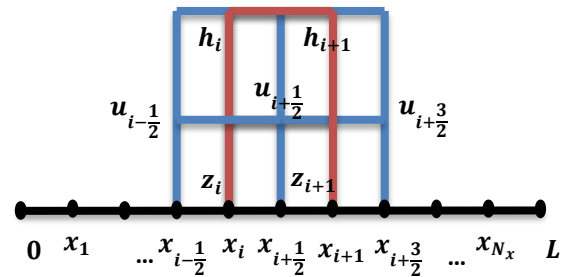


Fig. 2. Illustration of finite volume method on staggered grid.

Then, discretization of mass conservation equation (1) is written as

$$\frac{h_j^{n+1} - h_j^n}{\Delta t} + \frac{*hu|_{j+\frac{1}{2}}^n - *hu|_{j-\frac{1}{2}}^n}{\Delta x} = 0. \quad (6)$$

In Eq. (6), the water depth  $h_{j+1/2}$  and  $h_{j-1/2}$  are unknown, so we approximate their values using the first order upwind method

$$^*h_{j+1/2}^n = \begin{cases} h_j^n, & \text{if } u_{j+1/2} \geq 0, \\ h_{j+1}^n, & \text{if } u_{j+1/2} < 0. \end{cases} \quad (7)$$

Upwind method means when the flow goes to the right, we take the information from the left side. And when the flow goes to the left, we take the information from the right side.

Here, we rewrite the momentum balance equation (2) using simple algebra read as

$$u_t + uu_x + gh_x + g(z - H)_x + \frac{S_f}{h} = 0. \quad (8)$$

Using the same method as used in momentum conservation, the discrete form of Eq. (8) is

$$\begin{aligned} & \frac{u_{j+1/2}^{n+1} - u_{j+1/2}^n}{\Delta t} + g \frac{h_{j+1}^{n+1} - h_j^{n+1}}{\Delta x} \\ & + g \frac{(z - H)_{j+1}^n - (z - H)_j^n}{\Delta x} + \frac{S_f|_{j+1/2}^n}{h_j^n} \\ & + uu_x|_{j+1/2}^n = 0. \end{aligned} \quad (9)$$

It is important to notice that in order to prevent the presence of non-entropic shock, we are using semi-implicit approximation  $g \frac{h_{j+1}^{n+1} - h_j^{n+1}}{\Delta x}$  instead of explicit approximation  $g \frac{h_{j+1}^n - h_j^n}{\Delta x}$  in the momentum conservation [15].

The friction term  $S_f$  is discretized implicitly to reduce the restriction for the stability condition

$$S_f|_{j+1/2}^n = \frac{ku_{j+1/2}^{n+1}|u_j^n|}{(h_j^n)^{4/3}}. \quad (10)$$

Advection term  $uu_x$  related to momentum variable  $q = hu$  is given by

$$uu_x = \frac{1}{h} \left( \frac{\partial(qu)}{\partial x} - u \frac{\partial q}{\partial x} \right). \quad (11)$$

Discretization for advection terms Eq. (11) read as

$$\begin{aligned} (uu_x)_{j+1/2} = & \frac{1}{h_{j+1/2}} \left( \frac{\bar{q}_{j+1} {}^*u_{j+1} - \bar{q}_j {}^*u_j}{\Delta x} \right. \\ & \left. - u_{j+1/2} \frac{\bar{q}_{j+1} - \bar{q}_j}{\Delta x} \right), \end{aligned} \quad (12)$$

whereas

$$\bar{h}_{j+1/2} = \frac{1}{2} (h_j + h_{j+1}),$$

$$\bar{q}_j = \frac{1}{2} (q_{j+1/2} + q_{j-1/2}), \quad q_{j+1/2} = {}^*h_{j+1/2} u_{j+1/2},$$

with the following upwind approximation of  $u_j$

$$^*u_j = \begin{cases} u_{j-1/2}, & \text{if } \bar{q}_j \geq 0, \\ u_{j+1/2}, & \text{if } \bar{q}_j < 0. \end{cases} \quad (13)$$

Using the same method, the discretization of Exner Equation (3) is then

$$\frac{z_j^{n+1} - z_j^n}{\Delta t} + \zeta \frac{Qs_{j+1/2}^{n+1} - Qs_{j-1/2}^{n+1}}{\Delta x} = 0. \quad (14)$$

Discretization for transport sediment  $Qs$  read as

$$Qs_{j+1/2}^{n+1} = A_g \left( |u|_{j+1/2}^{n+1} \right)^2 u_{j+1/2}^{n+1}. \quad (15)$$

Then, we approximate Shallow Water Equations and Exner Equation sequentially. First, we calculate  $h_j^{n+1}$  using Eq. (6), then we calculate  $u_{j+1/2}^{n+1}$  using Eq. (9), and finally we calculate  $z_j^{n+1}$  using Eq. (14). Notice that we are using implicit evaluation in Eq. (14) in order to prevent additional stability condition. Since the sediment transport does not influence the fluid flow, a sufficient stability condition for our scheme is same as stability condition for one-dimensional Shallow Water Equations given by

$$\frac{\Delta t}{\Delta x} \sqrt{gH} \leq 1.$$

Note that we only consider first-order scheme in space and time for all equations. An advantage of this is that it seems to maintain stability properties, while giving sufficiently accurate results, as shown in Section 4.

#### 4. NUMERICAL SIMULATION

In evaluating the accuracy of our proposed numerical scheme, we will conduct some of numerical simulations. First, we will evaluate our scheme with cases from literature. Second, we will compare our scheme to experimental data of dam break simulations.

##### 4.1 Comparison with analytical solution.

To show the performance of our scheme, we consider a case where an analytical solution is

available in [16]. In this case, we use  $A_g = 0.005$ ,  $\Delta x = 0.07$ , and  $\Delta t = 0.01$ . The initial conditions are given by

$$\begin{aligned} (hu)_{ini}(x) &= 1, \\ h_{ini}(x) &= \frac{(hu)_{ini}(x)}{u_{ini}(x)}, \\ u_{ini}(x) &= \left[ \frac{\alpha x + \beta}{A_g} \right]^{\frac{1}{3}}, \\ z_{ini}(x) &= 1 - \frac{u_{ini}^3(x) + 2g(hu)_{ini}^2(x)}{2gu_{ini}(x)}, \end{aligned}$$

where the coefficient  $\alpha$  and  $\beta$  are given by  $\alpha = \beta = 0.005$ . In this simulation, we assume that there is no friction, or  $S_f = 0$ .

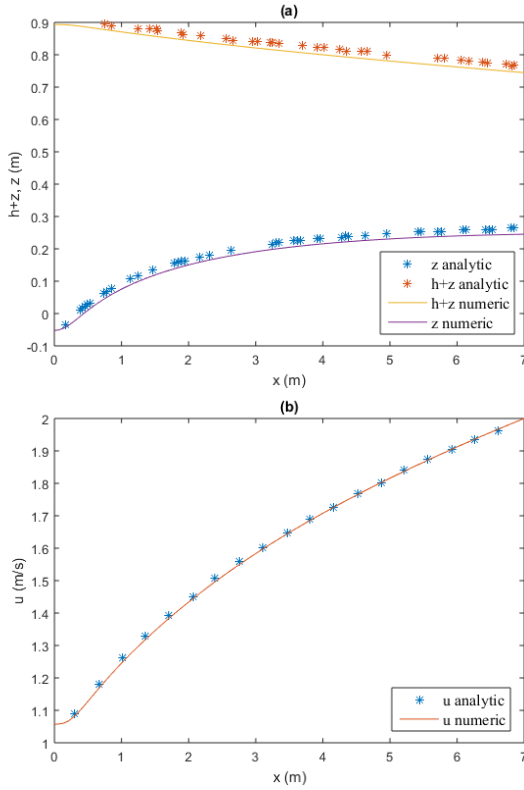


Fig. 3. Comparison between analytical and numerical solutions of (a) water level and sediment profile, (b) velocity profile at the final time  $t = 7$ .

From Fig. (3), we can see that our numerical scheme has successfully approximated the analytical solution for water level, sediment elevation and wave velocity. Using Root Mean Square Error (RMSE) method, we have found that the error between the analytical and numerical results for sediment elevation  $z$  is 0.01663, for water level  $h + z$  is 0.01872, and for the velocity  $u$  is 0.00556. Those error are very small which prove that our numerical scheme fits the analytical model perfectly.

Further, we conduct dam break case to show a comparison between analytical and numerical

solutions for explicit scheme and semi-implicit scheme. Consider domain  $[0, 200]$  and the initial conditions which are given by

$$\begin{aligned} h_{ini}(x) &= \begin{cases} 1 & \text{if } x \leq 50, \\ 0 & \text{otherwise.} \end{cases} \\ u_{ini}(x) &= 0, \quad z_{ini}(x) = 0. \end{aligned}$$

In this simulation, we assume that there is no transport sediment, or  $A_g = 0$ . Using  $\Delta x = 1$  and  $\Delta t = 0.5$ , the simulation gives us Fig. (4) and (5).

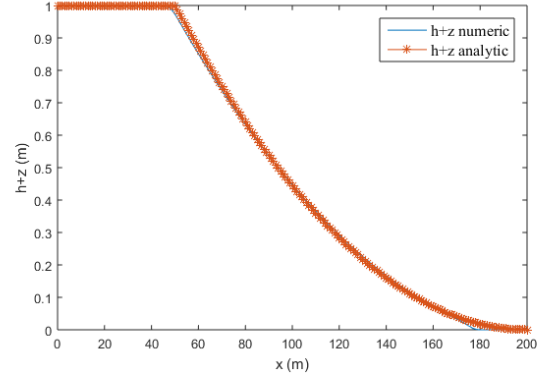


Fig. 4. Comparison between analytical and semi-implicit numerical solutions of water level in dam break case at final time  $T = 50$  s.

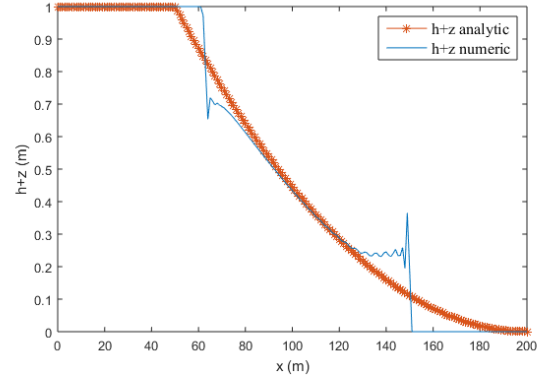


Fig. 5. Comparison between analytical and explicit numerical solutions of water level in dam break case at final time  $T = 50$  s.

As seen in Fig. (5), the analytical solution seems to be well approximated by our semi-implicit scheme compared to the explicit scheme. Using RMSE, the error between analytical and numerical  $h + z$  for semi-implicit scheme is 0.00918. This is much smaller than the error for the explicit scheme which is 0.04803. This proves that our semi-implicit scheme works much better than the explicit scheme to simulate the dam break model.

#### 4.2 Dam break test.

In this section, we will conduct numerical simulation of dam break. The first case is devoted to wet dam break and the second one is for dry dam

break. First, consider domain  $[0, 10]$  with a dam located in the middle of the domain. The initial condition is given by

$$h_{ini}(x) = \begin{cases} 2 & \text{if } x \leq 5 \\ 0.125 & \text{otherwise} \end{cases}$$

$$u_{ini}(x) = 0, \quad z_{ini}(x) = 0,$$

and assume there is no friction term. Using  $A_g = 0.005$  and  $\Delta x = 0.025$ , we will get results as shown in Fig. (6).

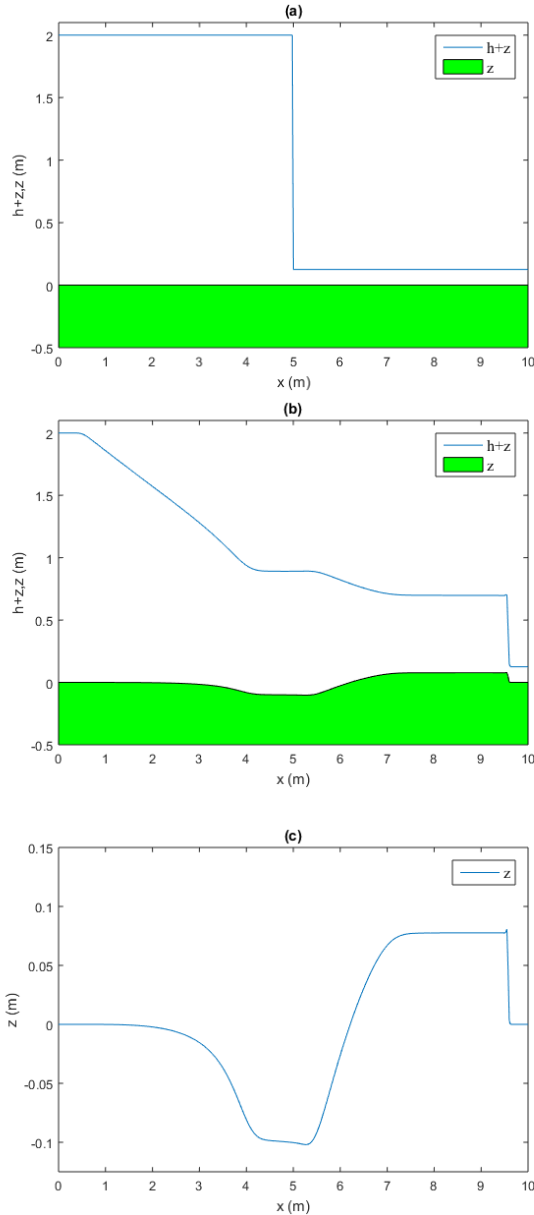


Fig. 6. Wet dam break. (a) Initial condition of dam break. (b) Water level and sediment profile at final time  $t = 1$ . (c) Zoom area of sediment profile.

In Fig. (6.b), we can see that as the dam breaks, the water that propagates to the right side brings along the sediment under it. It makes sense and agree with the real phenomenon of dam break cases. Notice that in Fig. (6.c), at the right side, the

sediment is elevated and becomes about **0.08 m** over the flat bed  $z = 0$ . That is actually **4.267 %** of the water level difference which is **1.875 m**. The sediment elevation could be higher if we consider the energy of the water waves caused by the breaking process of the dam.

Next, consider domain  $[0, 10]$  with a dam located in the middle of the domain. Initial conditions are given by

$$h_{ini}(x) = \begin{cases} 1 & \text{if } x \leq 5 \\ 0 & \text{otherwise} \end{cases}$$

$$u_{ini}(x) = 0, \quad z_{ini}(x) = 0,$$

and assume there is no friction term. This case is dry dam break because there is no water at right side of the dam. Using  $A_g = 0.005$  and  $\Delta x = 0.01$ , we will get result as shown in Fig. (7).

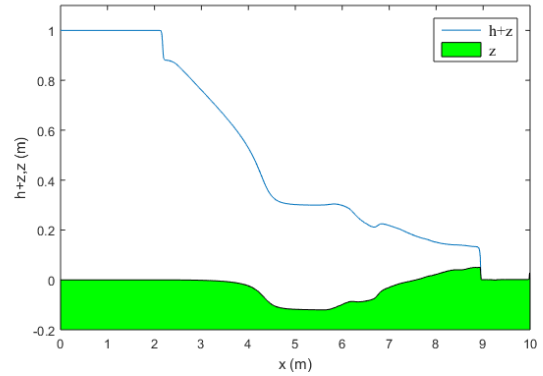


Fig. 7. Water level and sediment profile for dry dam break case at final time  $t = 1$ .

In Fig. (7), we consider the dam break case in the dry condition. From the figure, the sediment transported to the right side is about **0.05 m**. Compared to the water level difference, which is 1 m, the sediment elevation is exactly **5 %** of it. This is bigger than the one produced in the wet dam break condition. This might be caused by the fact that in the wet condition, there are water at the right side which will restrain the flow of the water waves as well as restrain the sediment transport. On the other hand, in dry condition, there are no water to restrain the flow, consequently, the sediment that is transported is higher.

Furthermore, we will show a comparison between numerical solution and experimental data. Some laboratory experiments of dam break over a granular bed have been reported. We focus on the experiment performed in Louvain-la-Nueve, Universite of Catholique de Louvain [14].

Consider the domain of experiment is  $[-3, 3]$  where the dam is located in the middle of the domain. The dam separates two initial water heights  $h_{ini}(x) = 0.35$  at the left side and  $h_{ini}(x) = 0$  at the other side. Initially, the sediment level is  $z_{ini}(x) = 0$ , while the fluid velocity is  $u_{ini} = 0$ . Since we use Grass Formula, we assume that  $A_g =$

**0.005.** In this section, we use  $\Delta x = 0,025$  and coefficient Manning  $k = 0.001$ .

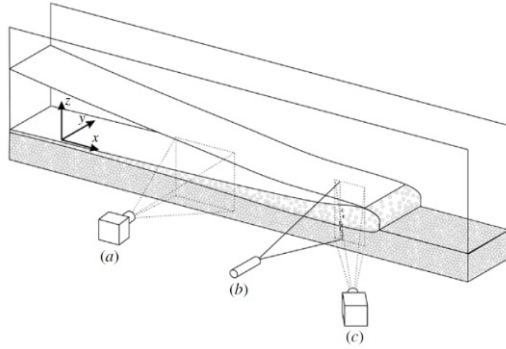


Fig. 8. The experimental set-up with two imaging configuration: (a) the camera normal to the side wall; (b,c) a transverse laser light sheet and an oblique camera.

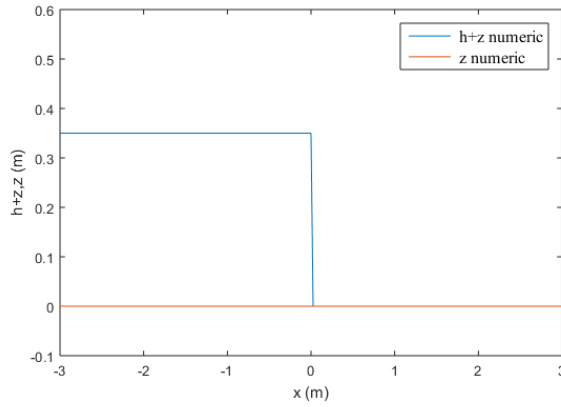


Fig. 9. The initial condition of experiment in one-dimensional.

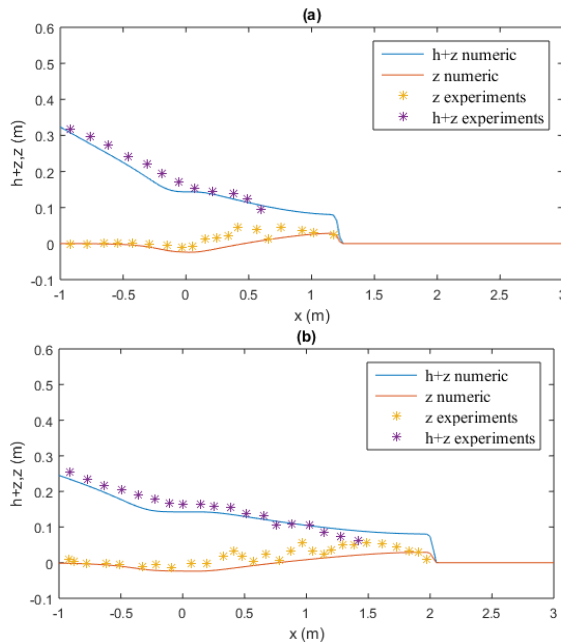


Fig. 10. Comparison between numerical solutions and experimental data of water level and sediment profile at final time (a)  $t = 0.6$  and (b)  $t = 1$ .

The experiment was carried out in the dam-break flume at Universite of Catholique de Louvain. As in Fig. (8), the flume has a horizontal bottom, a total length  $L = 6 \text{ m}$ , an adjustable width set at **25 cm** and a sidewall height of **70 cm**. More details about the flume and experiment are given in [13].

Results at different times are presented in Fig. (10). They show a good comparison between numerical solutions and experimental data in aspect of water level and sediment profiles. We prove this statement by calculate the error between numerical and experimental results in both water level and sediment elevation for both  $t = 0.6$  and  $t = 1$ . Using RMSE, the error for  $h + z$  is 0.02289 at  $t = 0.6$  and 0.02187 at  $t = 1$ . For  $z$ , the numerical-experimental error is 0.02118 at  $t = 0.6$  and 0.02509 at  $t = 1$ .

#### 4.2 Comparison with experimental data.

In this section, we will compare our numerical results to the data collected in a non-dambreak experiment which taken place at Tohoku University [8]. The set-up of the experiment can be seen in Fig. (11).

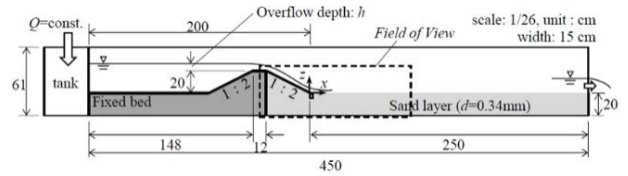


Fig. 11. Set-up of sediment transport experiment conducted at Tohoku University.

The experiment was conducted by generating a constant water flow to the flume to keep  $h$  constant. At the right side, we have an absorbing boundary. In the area behind the structure, sediment with 0.2 m deep is placed, to see how the flow and structure affect the sediment transport.

For simulation, we begin with illustrating the evolution of the incoming flow, before and after its interaction with the structure, using our numerical scheme. The evolution of the water flow for different observation time can be seen in Fig. (12).

Fig. (12.a) shows the water profile when it just started to propagate into the domain with the initial height. After a while, the flow starts to interact with the structure, causing a condition similar to wave run-up condition, showed in Fig. (12.b). In Fig. (12.c), because of the flow that is constantly generated, there will be an overtopping condition occur over the structure.

In order to be able to compare our numerical results to the experimental data, we need to adjust the initial condition. Because there is a limitation regarding the incoming flux that is used in the

experiment, we assume that the flume has already been filled with water when the experiment started. Thus, the initial condition for this comparison is illustrated in Fig. (13).

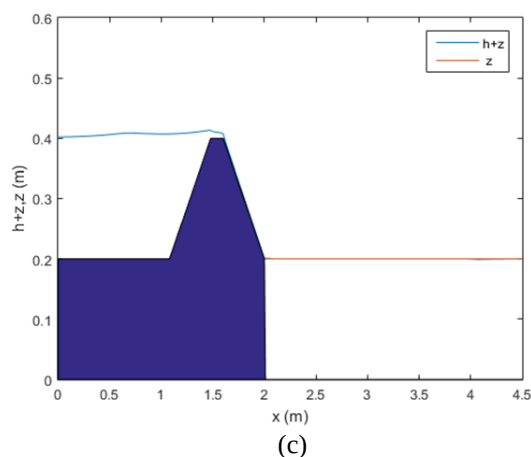
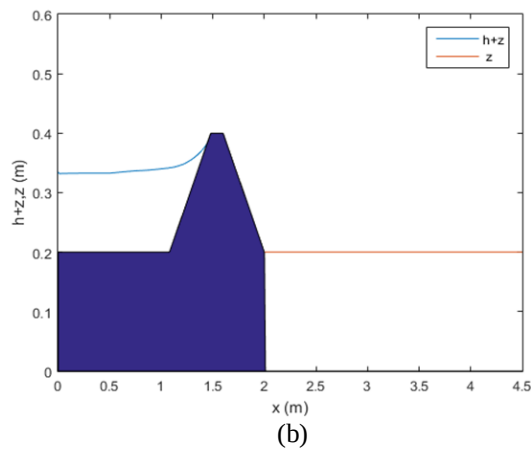
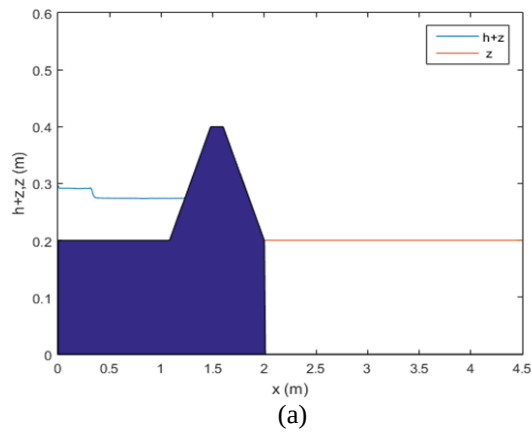


Fig. 12. The evolution of the incoming water flow at (a)  $t = 4.5$  s, (b)  $t = 8.6$  s, and (c)  $t = 13.5$  s.

In this simulation, we focus on observing the domain  $[2.05, 2.5]$  where the sediment profile is change. Then, the result of our simulation is compared to the experimental data, which is presented in Fig. (14).

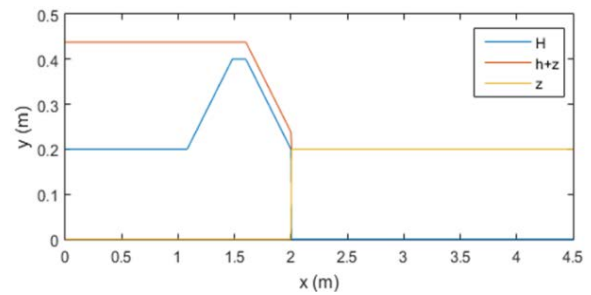


Fig. 13. Initial condition of the simulation.

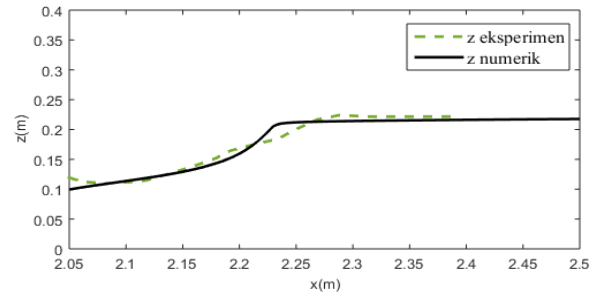


Fig. 14. The comparison between sediment profile simulated using our numerical scheme and the one that based on experimental data.

From Fig. (14), we can see that our numerical scheme has simulated the sediment transport phenomena very well, compared to the experimental data. This statement can be proved by the error of our result calculated using RMSE, which is **0.01055**.

## 5. CONCLUSION

A mathematical model has been developed to simulates a dam break problem with movable bed condition. The model is based on Shallow Water Equations, coupled with sediment transport equation. The governing equations are solved using finite volume method on a staggered grid with a semi-implicit momentum conservative scheme.

The model was verified with a case of analytical dam break. In addition, our semi-implicit scheme was also compared to the explicit scheme. The results from the semi-implicit scheme shows a better comparison to the analytical solution than those of the explicit scheme. The developed model was applied to simulate a hypothetical dam break case with movable bed. The downstream of the dam was simulated under two scenarios, wet and dry. In both scenarios, the model performs well. The model was further applied and verified to an experimental case of dam break with movable bed. The hydraulic parameters as well as the bed level changes produced by the model show good comparisons to the experimental data. In the near future, this model can be developed into a 2-D model to consider the volume of the sediment transport. It also would be good to compare the simulation results to the real

phenomenon of sediment transport in dam break cases.

## 6. ACKNOWLEDGMENTS

The authors would like to express their gratitude to Riset Multidisiplin ITB 2020, Riset PDUPT 2020 and Research, Community Services, and Innovation Program (P3MI) of Faculty of Civil and Environmental Engineering ITB.

## 7. REFERENCES

- [1] Yakti B. P., Adityawan M. B., Farid M., Suryadi Y., Nugroho J. and Hadihardaja I. K., 2D Modeling of Flood Propagation due to the Failure of Way Ela Natural Dam, MATEC Web of Conferences, Vol. 147, 2018, 03009.
- [2] Farid M., Yakti B. P., Rizaldi A., and Adityawan M. B., Finite Difference Numerical Scheme for Simulating Dam Break Flow, The 5th HATHI International Seminar on Water Resilience in a Changing World, 2016.
- [3] Harlan D., Adityawan M. B., Natakusumah D. K., and Zentrato N. L. K., Application Of Numerical Filter To A Taylor Galerkin Finite Element Model For Movable Bed Dam Break Flows, International Journal of GEOMATE, Vol. 16, Issue 57, 2019, pp. 209 – 216.
- [4] Zentrato N. L. H., Harlan D., Adityawan M. B., and Natakusumah, D. K., 1D Numerical Modelling of Dam Break Using Finite Element Method, Proceedings of The 2nd Conference for Civil Engineering Research Network, 2018.
- [5] Peng S. H., 1D and 2D Numerical Modeling for Solving Dam-Break Flow Problems Using Finite Volume Method, Journal of Applied Mathematics, Article ID 489269, 2012, pp.14
- [6] Liang D., Falconer R. A., and Lin B. L., Comparison between TVD-MacCormack and ADI-type solvers of the shallow water equations, Advances in Water Resources, Vol. 29, No. 12, 2006, pp. 1833-1845.
- [7] Tseng M. H., Improved treatment of source terms in TVD scheme for shallow water equations, Advances in Water Resources, Vol. 27, No. 6, 2004, pp. 617-629.
- [8] Adityawan M. B., Tanaka H., Dam break wave simulation using the simultaneous coupling method, Proceedings of 34th IAHR Congress, 2013.
- [9] Pu J., Bakenov Z. and Adair D., Advanced Numerical Solver for Dam-Break Flow Application, Eurasian Chem. Tech. J., Vol. 14, No. 1, 2012, pp. 87-96.
- [10] Wang Y., Liang Q., Kesserwani G. and Hall J. W., A 2D shallow flow model for practical dam-break simulations, Journal of Hydraulic Research, Vol. 49, Issue 3, 2011, pp. 307-316.
- [11] Pudjaprasetya S.R. and Magdalena I., Momentum conservative scheme for dam break and wave run up simulations, East Asian J. Appl. Math, Vol. 4, Issue 2, 2014, pp. 152-16.
- [12] Magdalena I., Erwina N., and Pudjaprasetya S. R., J. Sci. Comput., Vol. 65, Issue 3, pp. 867–874.
- [13] Spinewine B. and Zech Y., Small-Scale Laboratory Dam-Break Waves on Movable Beds, Journal Hydraulics Research, Vol. 45, 2007, pp. 73-86.
- [14] Spinewine B. and Capart H., Intense bed-load due to a sudden dam-break, Journal of Fluid Mechanics, Vol. 731, 2013, pp. 579-614.
- [15] Doyen D. and Gunawan P.H., An explicit staggered finite volume scheme for shallow water equations, Finite Volumes for Complex Applications VII-Methods and Theoretical Aspects, Springer, 2014, pp. 227-235.
- [16] Berthon C., Cordier S., Delestre O., and Le M. H., An analytical solution of the shallow water system coupled to the exner equation, C. R. Math., Vol. 350, Issue 3, 2012, pp. 183–186.

---

Copyright © Int. J. of GEOMATE. All rights reserved, including the making of copies unless permission is obtained from the copyright proprietors.

---