An Alternative Algorithm for Simulating Flash Flood

M F Khaldirian, A P Rahardjo*, D Luknanto, and R D R Sondi

1Civil and Environmental Engineering Department, Universitas Gadjah Mada, Indonesia
2Master Student, Civil and Environmental Engineering Department, Universitas Gadjah Mada, Indonesia
3Undergraduate Student, Civil and Environmental Engineering Department, Universitas Gadjah Mada, Indonesia

*Correspondent author: rahardjo.adam@ugm.ac.id

Abstract. Most of the approaches in numerical modeling techniques are based on the Eulerian coordinate system. This approach faces difficulty in simulating flash flood front propagation. This paper shows an alternative method that implements a numerical modeling technique based on the Lagrangian coordinate system to simulate the water of debris flow. As for the interaction with the riverbed, the simulation uses an Eulerian coordinate system. The method uses the conservative and momentum equations of water and sediment mixture in the Lagrangian form. Source terms represent deposition and erosion. The riverbed in the Eulerian coordinate system interacts with the flow of the mixture. At every step, the algorithm evaluates the relative position of moving nodes of the flow part to the fixed nodes of the riverbed. Computation of advancing velocity and depth uses the riverbed elevation, slope data, and the bed elevation change computation uses the erosion or deposition data of the flow on the moving nodes. Spatial discretization is implementing the Galerkin method. Furthermore, temporal discretization is implementing the forward difference scheme. Test runs show that the algorithm can simulate downward, upward, and reflected backward 1-D flow cases. Two-D model tests and comparisons with SIMLAR software show that the algorithm works in simulating debris flow.

Keywords: flash flood simulation, approach, water and sediment mixture flow

1. Introduction

1.1. Background
Most flash floods propagate downstream with a front because inertia dominates the flow movement [1]. Most of the approaches in numerical modeling techniques are based on the Eulerian coordinate system. This approach faces difficulty in simulating flash flood front propagation. In the Lagrangian approximation, the discretization points on the elements act as moving flow particles. In this Lagrangian approach, elements can compress, stretch and shift because the nodes that create the element are particles that change their positions. The element acts as a moving control volume. While in the Eulerian approach, the discretization points act as control points with the modeling results obtained are the flow parameter values located at the control points. Hence, the Lagrangian approach allows obtaining the behavior of a specific particle that is considered in the modeling of lahar flow.
1.2. Literature Review

Numerical modeling of lahar flows has been carried out by several individuals and institutions. Al Ridlo conducted a one-dimensional lahar flow modeling to compare the Lagrangian approach with the Eulerian approach [2]. In this study, a weighting function in the form of a linear interpolation function was used, with time step discretization in a first-order forward differential scheme and space discretization using the Galerkin method. For the 1-dimensional case, the formulation is identical to the Forward Time Central Space (FTSC) Finite Difference scheme. Several institutions have developed and used lahar flow modeling programs. SIMLAR is a 2D lahar flow modeling software used by Balai Sabo, Ministry of PUPR. It implements an Eulerian coordinate system, a space finite difference method with a leap-frog scheme, and a first-order forward differential scheme for time step discretization so that it can be classified as an FTSC scheme.

Debris flow consists of a mixture of water and sediment with a high concentration in a river with a steep slope. This debris flow moves at high velocity due to gravity. The presence of large materials transported by debris flows, such as rocks, tree trunks, and other materials, can cause damage that can be detrimental [3]. The movement of debris flow is strongly influenced by the forces of solid particles and fluids. This interaction between solids and fluids makes debris flow different from rockslides and water flows [4]. The equations of formation, mass continuity and momentum of a debris flow can be integrated with depth. In a fixed Cartesian coordinate system (x, y, z) with z pointing up against the direction of gravity, the formation equation can be reduced to the average depth relationship in the x-y plane [5]. The finite element method is one of the numerical solutions to obtain an approximate solution to a physical problem [6]. The finite element method divides the area under consideration into small sections called elements. The partial differential equation, a mathematical form of a physical process in the lahar flow, is applied to these elements so that formula is obtained in the form of a variable relationship that one wants to find in the element [7]. The shapes of elements are the simplest and can be made in each of the dimensions concerned. The selection of simple shapes is under the philosophy of the Finite Element Method, which is to simplify the complex shapes of the area under review so that problems can be solved [8].

1.3. Development of 2D Debris Flow Model and the Previous Research

Most 2D depth-averaged numerical debris flow models were developed using the Eulerian approach [9–11]. This approach had difficulties modeling the propagation of debris fronts in the flow field, which may flow uphill. Therefore, an alternative, namely the Lagrangian approach for modeling debris flow, was developed. The 1D model of debris flow has shown promising solutions [2]. However, debris flow problems in the alluvial fan area can’t rely on the 1D model result. This is because the change in debris parameters represents their channel cross-section average values. Therefore, this research was conducted to extend the 1D Lagrangian debris flow model to a 2D model. It is expected that the model can simulate a debris front propagation of 2D topography more realistically.

2. Materials and Methods

2.1. Erosion and Deposition

The following equation obtains the ratio of deposition and erosion in debris flow [12]. For the case of erosion, the sediment concentration in the flow is smaller than the balanced condition or $C_{\infty} \geq C$. The erosion ratio used is defined as follows.

$$i = \delta_e \frac{C_{\infty} - C}{C_{\infty} - C_{\infty} d_x}$$

while for the deposition case, the sediment concentration in the flow is greater than the balanced concentration ($C \geq C_{\infty}$), then the deposition ratio used is as follows.
where $\delta_e$ is the erosion coefficient, $\delta_d$ is the deposition coefficient, $i$ is the erosion and deposition ratio, $C_\star$ is the maximum concentration at the bottom, $u$ is the flow velocity, $h$ is the depth, $d_L$ is the particle diameter, $C$ is the concentration of sediment in the stream, $C_\infty$ is a balanced concentration where the magnitude is affected by the bed slope and internal friction angle.

The balanced concentration ($C_\infty$) versus riverbed slope ($\tan \theta$) for the type of debris flow and hyper-concentrated sediment flow can be formulated in several classifications of debris flow, including stony debris flow, immature debris flow, and bedload transport. For stony debris flow, the balanced concentration and bed stress values are obtained as follows.

$$C_\infty = \frac{\tan \theta}{\left(\frac{\sigma}{\rho} - 1\right)(\tan \phi - \tan \theta)}$$  \hspace{1cm} (3)

$$\tau_b = \rho_s \tan \phi + \frac{\rho}{8} \left(\frac{C_\star}{C_\infty}\right)^{1/3} \left(\frac{d_L}{h}\right)^2 u|u|$$  \hspace{1cm} (4)

where,

$$\rho_s = f(C)(\sigma - \rho)Cgh \cos \theta ; \quad f(C) = \begin{cases} \frac{(C - C_3)}{(C_\star - C_3)} & C > C_3 ; C_3 = 0.5 \\ 0 & C \leq C_3 \end{cases}$$

$C_3$ is the limiting concentration where $\rho_s$ affects. For immature debris flow, the balanced concentration and bed stress values are obtained as follows.

$$C_\infty = 6.7 \left[\frac{\tan \theta}{\left(\frac{\sigma}{\rho} - 1\right)(\tan \phi - \tan \theta)}\right]^2$$  \hspace{1cm} (5)

$$\tau_b = \frac{\rho_T}{0.49} \left(\frac{d_L}{h}\right)^2 u|u|$$  \hspace{1cm} (6)

For bedload transport, the balanced concentration and bed stress values are obtained as follows.

$$C_\infty = \left(\frac{\rho_T}{\rho}g d\right)^{1/3} \left(\frac{(C - C_3)}{(C_\star - C_3)}\right) \left(1 - \alpha_0 \frac{\tau_{ce}}{\tau_*}\right)$$  \hspace{1cm} (7)

$$\tau_b = \frac{\rho_T n_M^2}{h^{1/3}} u|u|$$  \hspace{1cm} (8)

where,

$$\tau_* = \rho \frac{\tau}{(\sigma - \rho)gd} = \frac{u^2}{sgd} ; \quad \tau_{ce} = \frac{\tau_c}{(\sigma - \rho)gd} = \frac{u_{*c}^2}{sgd} = 0.05 ; \quad \alpha_0 = u + 2\sqrt{gh}$$

$\rho$ is the density of water, $\sigma$ is the density of the sediment, $\rho_T$ is the density of the mixture, $\theta$ is the slope of the bed, $g$ is the acceleration of gravity, $\tau_b$ is the basic shear stress, $n_M$ is the Manning coefficient, $\phi$ is the internal friction angle and $i$ is the erosion and deposition ratio.

2.2. Lagrangian Approach

In Lagrangian approximation, partial differential equations are derived for moving particles as follows.
Figure 1. Debris flow moving volume.

In Figure 1, the element acting as the control volume undergoes a compression, shift or stretch following the particle displacement acting as discretization points. For the 2-dimensional case, the base is the area of the element according to the shape of the specified element. The control volume limits particle changes that can shift, compress or stretch, their position coordinates change according to the equation.

\[
\frac{\partial x}{\partial t} = u; \quad \frac{\partial y}{\partial t} = v
\]

\(\frac{\partial}{\partial t}\) is a total differential operator, \(x\) and \(y\) are space coordinates, \(u\) is the \(x\)-axis velocity, \(v\) is the \(y\)-axis velocity, \(t\) is time.

2.3. Governing Equation

In 2D debris flow simulation with uniform parameter values as deep as the flow (depth-averaged), the Lagrangian approach’s continuity equation and momentum conservation equation are determined as follows.

Mixed continuity equation:

\[
\frac{DV}{Dt} = iA[C_* + (1 - C_*)s_b] + rA
\]

Solid continuity equation:

\[
\frac{DCV}{Dt} = iAC_*
\]

\(V\) is the control volume, \(A\) is the area, \(r\) is lateral flow parameter and \(s_b\) is the degree of sediment saturation. In the momentum conservation equation, the forces acting on the flow control volume are arranged.

Figure 2. The component of the force acting on the control volume.

In Figure 2 the working momentum is caused by the components of the force, including \(F_1\) and \(F_2\) which are hydrostatic forces, \(F_3\) is gravity and \(F_4\) is the bottom frictional force of the channel. So, the momentum conservation equation can be formulated as follows.
\[
\frac{DuV}{Dt} = gV \sin \theta_x - gV \cos \theta_x \frac{dh}{dx} - \frac{A \tau_b}{\rho_T} \quad ; \quad \frac{DvV}{Dt} = gV \sin \theta_y - gV \cos \theta_y \frac{dh}{dy} - \frac{A \tau_b}{\rho_T}
\]

In the discontinuous form the depth of flow, the drainage area and the volume are defined by the formulation.

\[V = A \cdot h\]  

(13)

where \(h\) is the flow depth, \(A\) is the control volume, \(x\) and \(y\) are the space coordinates, \(u\) is the \(x\)-axis direction velocity, \(v\) is the \(y\)-axis direction velocity, \(t\) is time, \(\rho_T\) is the lahar flow density, \(\theta\) is the bed slope, \(A\) is the surface area, \(g\) is the acceleration due to gravity, \(\tau_b\) is the bed shear stress in the flow.

2.4. Galerkin Method

This method uses the same function for the weighting function and its basic function, namely the interpolation function (\(N\)) so that \(W = N\). The interpolation of the function is written as follows.

\[
\hat{f} = \sum_{i=1}^{n} \hat{f}_i N_i
\]

(14)

Where \(\hat{f}\) is estimated function value, \(\hat{f}_i\) is the value of the function at node \(i\) which is considered in the element \(N\) is shape function. The interpolation of the first derivative of the \(\hat{f}\) function in Cartesian coordinates is as follows.

\[
\frac{\partial \hat{f}}{\partial x} = \sum_{i=1}^{n} \hat{f}_i \frac{\partial N_i}{\partial x} ; \quad \frac{\partial \hat{f}}{\partial y} = \sum_{i=1}^{n} \hat{f}_i \frac{\partial N_i}{\partial y} ; \quad \frac{\partial \hat{f}}{\partial z} = \sum_{i=1}^{n} \hat{f}_i \frac{\partial N_i}{\partial z}
\]

(15)

2.5. Numerical Discretization

In general vector form, the governing equation can be arranged as follows.

\[
\frac{DP_x}{Dt} = - \frac{\partial F_x}{\partial x} + R_x ; \quad \frac{DP_y}{Dt} = - \frac{\partial F_y}{\partial y} + R_y
\]

(16)

A first-order forward differential scheme according to the Taylor series equation is performed to discretize the time on the left side of the equation. Physically, the Taylor series can be interpreted as the magnitude in space and time which is calculated from the magnitude itself in a certain space and time which has a small difference with the observed space and time [13]. So that the time discretization can be expressed as follows.

\[
U^{t+1} = U^t + \frac{\partial U}{\partial t} \Delta t
\]

(17)

Meanwhile, for the discretization of space, the Galerkin method is used with a weighting function, an interpolation function for the shape of a linear triangular element.

Figure 3. Linear elements of an isosceles right triangle.

Figure 3 shows the basic shape of linear triangular elements. The interpolation function of the elements of a linear triangle with the basic form of an isosceles right triangle is as follows.

\[N_i = \alpha_i + \beta_i x + \gamma_i y\]

(18)

Where,
\[
\alpha_i = \frac{(x_jy_k - x_ky_j)}{2A}; \quad \beta_i = \frac{(y_j - y_k)}{2A}; \quad \gamma_i = \frac{(x_k - x_j)}{2A}; \quad A = \frac{1}{2} \begin{vmatrix}
1 & x_i & y_i \\
1 & x_j & y_j \\
1 & x_k & y_k \\
\end{vmatrix}
\]

\(x\) and \(y\) are the coordinates, \(i, j, k\) are the number of nodes, \(A\) is the area of the element and \(N_i\) is the interpolation function. The discretization of the general form of the governing equation is formulated in the following Galerkin form.

\[
\int_{\Omega} N_j N_i \, d\Omega = \Delta t \left( \int_{\Omega} N_j \frac{\partial N_i}{\partial \Omega} \, d\Omega F_i^t + \int_{\Omega} N_j N_i \, d\Omega R_i^t \right)
\]

or in vector-matrix form.

\[
M \, \Delta \mathbf{U} = \Delta t (K F^t + M R^t)
\]

2.6. Research Procedure

At every step, the algorithm evaluates the relative position of moving nodes of the flow part to the fixed nodes of the riverbed. Computation of advancing velocity and depth uses the riverbed elevation and slope data and the bed elevation change computation uses the erosion or deposition data of the flow on the moving nodes. Spatial discretization implements the Galerkin method and temporal discretization implements the forward difference scheme. In general, this research was conducted with the following steps.

- Define the continuity and momentum equations that will be used as a reference.
- Adjust the equation to the simulated conditions so that a new form is ready to be discretized.
- Compile the numerical discretization of the equation using the finite element method.
- Perform simulation tests.

2.7. Initial and Boundary Conditions

The initial conditions are determined by setting a certain depth value at each point of the debris deposit. The initial condition of the debris deposit is at rest. The velocity of the debris particle for the next time step is obtained due to the momentum component of the flow. The boundary condition applies moving boundary following the displacement of the nearest neighbor point in the internal domain area. The depth remains zero at the boundary points. There is no hydrostatic force applied to the outer vertices.

2.8. Discretization of the Governing Equation

The governing equation is substituted according to equation (16) as below,

\[
\mathbf{P}_x^k = (X \ V \ CV \ uV)^t; \quad \mathbf{P}_y^k = (Y \ V \ CV \ vV)^t
\]

\[
\mathbf{F}_x^k = (0 \ 0 \ 0 \ g \cos \theta_x(h))^t; \quad \mathbf{F}_y^k = (0 \ 0 \ 0 \ g \cos \theta_y(h))^t
\]

\[
\mathbf{R}_x^k = \begin{pmatrix}
\frac{iA[C_x + (1 - C_s)b] + rA}{iAC_x} \\
gV \sin \theta_x - \frac{Ar_{bx}}{\rho_T}
\end{pmatrix}^t; \quad \mathbf{R}_y^k = \begin{pmatrix}
\frac{iA[C_x + (1 - C_s)b] + rA}{iAC_x} \\
gV \sin \theta_y - \frac{Ar_{by}}{\rho_T}
\end{pmatrix}^t
\]

The equations (9) will be formulated in the following discrete in vector-matrix form.

\[
[M][\Delta x] = \Delta t[M][u]; \quad [M][\Delta y] = \Delta t[M][v]
\]

For the constant volume and constant concentration rate, equations (10) and (11) will be equal to zero and variable volume (\(V\)) in equation (12) can be eliminated. So, the discrete formulation form of equations (10) and (11) will be formulated in the following vector-matrix form.
\[ [M][\Delta V] = \Delta t[M][0] \quad ; \quad [M][\Delta CV] = \Delta t[M][0] \]  

In Equation (25), the debris flow volume and concentration will be constant for each time step. The momentum equation can be simplified by removing variable \( V \) in the vector-matrix form as follows.

\[ [M] \Delta u = \Delta t[K]\left[g \cos \theta_x [h]\right] + \Delta t[M]\left[g \sin \theta_x - \frac{f_{bx}}{h \rho_T}\right] \]  

(26)

\[ [M] \Delta v = \Delta t[K]\left[g \cos \theta_y [h]\right] + \Delta t[M]\left[g \sin \theta_y - \frac{f_{by}}{h \rho_T}\right] \]  

(27)

### 2.9. Model Verification

Model verification is done by comparing to the analytic solution of traveled distance of the centroid of a slipping liquid volume with respect to time as follows.

\[ \frac{dx}{dt} = u_0 + \left( g \sin \theta - g \frac{n_M^2 \left( \frac{dx}{dt} \right)^2}{h^{5/3}} \right) t \]  

(28)

Simulations were carried out with a uniform slope value (\( \tan \theta_x = 0.3 \)) and several variations of the Manning bed roughness coefficient values of \( n_M = 0.0 \), \( n_M = 0.03 \), and \( n_M = 0.06 \). The centroid traveled distances of the results of each simulation are then compared with the analytic solutions. The results of the modeling give close values to the analytic solution as shown in the following graph.

![Figure 4. Comparison of modeling results with the analytic solutions of slipping liquid volume travel distances](image)

### 3. Results and Discussion

Modeling using Debris Flow 1-D was carried out with a bed slope of 0.1 and on a parabolic bed. In Figure 5, debris movement seems to be retarding for the flatbed because of the bed friction. As for the parabolic bed, it is shown that the model can simulate climbing flow and reflected backward flow. For the 2-D test, the debris volume and concentration are set to be constant over time. The debris volume discretization points were set at a uniform distance at the initial conditions. The initial velocity was set to zero and the flow depth was set at a certain value. The deposition and erosion rates are set to zero. There were some scenarios to observe the Lagrangian debris flow model, as shown in Table 1.
The longitudinal bed slope of the channel is constant along the channel from upstream to downstream ends, while the slope of the left and right bank forms a V-shaped channel. In all scenarios, the debris volume discretization points are spaced 1 m from each other before the movement begins.

Figure 5. Lagrangian Debris Flow 1-D model test run of flows on a flat and a parabolic beds.

Table 1. Test run scenarios.

| Scenario Code | Bed Slope | Bank Slope |
|---------------|-----------|------------|
| 01S0.1        | 0.1       | 0.0        |
| 01S1          | 1.0       | 0.0        |
| 02S1          | 1.0       | 1.0        |

Figure 6. Lagrangian model run test results on a uniform bed slope (tan $\theta_x = 0.1$) with flat bank slope (tan $\theta_y = 0$). (a) at initial condition, (b) at t = 1s, (c) at t = 3s, (d) at t = 5s.

Figure 6 shows that the volume of debris flows upstream to downstream and spreads to the left and right sides due to the absence of a bank slope. The elements stretch with a fixed volume, and the flow depth decreases and the area is expanded.
Figure 7. Long-section for Lagrangian model run test 01S0.1.

Figure 8. Cross-section for Lagrangian model run test 01S0.1.

Figure 7 and Figure 8 show the maximum depth at the initial conditions is set to 1 m. The maximum distance between the upstream and downstream boundaries is 9.0 m. The maximum distance between the left and right boundaries is 4.0 m. Meanwhile, after a 5 s run, the maximum depth is 0.173 m. The maximum distance between the upstream and downstream boundaries is 17.11 m. The maximum distance between the left and right boundaries is 12.09 m. The front of debris travels 16.22 m in 5 seconds.

Figure 9. Lagrangian run test results on uniform bed slope \((\tan \theta_x = 1.0)\) with flat bank slope \((\tan \theta_y = 0)\). (a) at initial condition, (b) at \(t = 2s\), (c) at \(t = 3s\), (d) at \(t = 4s\).

In the scenario as shown in Figure 9, the element's tendency to expand also occurs dominated by widening towards the left and right side of the channel. The velocity at the upstream boundary is smaller than the downstream velocity due to the difference in hydrostatic resistance in the direction of propagation.

Figure 10. Cross-section for Lagrangian run test 01S1.
According to Figure 11, the debris element slides at high velocity and expands the element lengthwise in the longitudinal section. Figure 10 shows that there is an expansion that tends to be symmetrical in the broader direction, with near-uniform velocity both to the left and to the right, so it appears that the debris element slides relatively in the middle of the channel. The maximum depth at the initial conditions is set to 3 m. The maximum distance between the upstream and downstream boundaries is 9.0 m. The maximum distance between the left and right boundaries is 4.0 m.

Meanwhile, after a 5 s run, the maximum depth is 0.823 m. The maximum distance between the upstream and downstream boundaries is 13.92 m. The maximum distance between the left and right boundaries is 9.53 m. The front of debris travels 57.6 m in 4 seconds. On a steeper slope, the propagation velocity is higher, the longitudinal stretch is much smaller, and the effect of hydrostatic resistance is getting less influencing than that of gravity. The widening of the elements to the left and right sides occurs.

Modeling test using SIMLAR program is carried out for the same scenario to compare the results of modeling debris flow behavior with different approaches. In modeling using SIMLAR program, the elements do not move, the discretization points remain at a certain position (Eulerian coordinate system), so the solution provides parameter changes at the discretization points over time.

![Figure 11. Long-section for Lagrangian model run test 01S1.](image)

![Figure 12. SIMLAR run results on uniform bed slope ($\tan \theta_x = 1.0$) with flat bank slope ($\tan \theta_y = 0$). (a) at t = 1s, (b) at t = 2s, (c) at t = 3s, (d) at t = 4s.](image)
SIMLR test for bed slope of 1.0 is shown in Figure 12. There are differences between the SIMLR input data and that of the 2-D Lagrangian model. In SIMLR, the initiation of movement is based on the input of the flow hydrograph at the inflow point which acts as the upstream boundary. The terrain regular grid points are used as the calculation domain of debris flow. Propagation of debris volume is visually visible due to changes in depth obtained from the solution at grid points over the entire terrain. It can be seen that at time 4 s, the downstream tip of the simulation results of both methods reach nearly the same location. However, the results of the Lagrangian method reach a longer distance (less than ten meters).

![Figure 13](image1.png)

**Figure 13.** Lagrangian model run test results on uniform bed slope (\(\tan \theta_x = 1.0\)) with bank slope (\(\tan \theta_y = 1.0\)). (a) at initial condition, (b) at \(t = 2\)s, (c) at \(t = 3\)s, (d) at \(t = 4\)s.

![Figure 14](image2.png)

**Figure 14.** Long-section for Lagrangian model run test 02S1.

In Figure 13, the volume of debris slides on the bank's slope keeping the flow concentrated, and tends to be slightly elongated and compressed in the middle of the channel during its propagation. However, the velocity at each point of the element's composition tends to be uniform. The elements do not undergo many changes in shape with a fairly small time step range \(dt = 0.01\) s. The shape of the volume is also maintained along with the propagation, the maximum depth at the initial conditions is set to 2 m. Figure 14 shows that at initial movement, the maximum distance between the upstream and downstream boundaries is 9.0 m. The maximum distance between the left and right boundaries is 2.82 m. Meanwhile,
after a 4 s run, the maximum depth is 2.83 m. The maximum distance between the upstream and downstream boundaries is 9.92 m. The maximum distance between the left and right boundaries is 2.69 m. The front of debris travels 55.51 m in 4 seconds.

4. Conclusion
The model test shows that the Lagrangian approach for the debris flow volume and fixed coordinate system for the bed can simulate debris flow in 1D and 2D coordinate systems. The Lagrangian approach provides a promising replication for approaching natural debris flow behavior such as downward, upward, and reflected backward movement. Comparison with the SIMLR model shows that the Lagrangian approach provides more stable results. In the 2D model simulation, the distribution of mass, depth, and momentum of a debris flow can be observed more thoroughly.

Further research is suggested to add algorithms so that it is possible to choose the shape and surface curvature to replicate more complex physical conditions. It is also necessary to conduct simulations for conditions with sediment concentration change and bed elevation changes. Adding model capability in the simulation of a splitting debris flow is also challenging.

References
[1] Widowati A P A 2017 Hydraulic and Hydrologic Modeling of Steep Channel of Putih River, Magelang District, Central Java Province, Indonesia J Civ Eng Forum 3 125–34
[2] Al Ridlo B Z and Rahardjo A P 2013 Progress on Numerical Model Development of ID Debris Flow using Lagrangian Galerkin Method 4th-International Work. MULTIMODAL SEDIMENT DISASTER
[3] Viesca R 2008 Debris Flow: Mechanics, Prediction and Countermeasures by Tamotsu Takahashi vol 165
[4] Iverson R and Denlinger R 1987 The physics of debris flows—a conceptual assessment IAHS-AISH Publ. 155–65
[5] Martinez C E 2009 Eulerian-Lagrangian Two Phase Debris Flow Model (Florida International University)
[6] Smith D E, Huebner K H, Dewhirst D L and Byrom T G 1995 The finite element method for engineers Choice Rev. Online 32 32-5680-32–5680
[7] Patiroi A 2010 Pemodelan Numerik Persamaan Boussinesq Menggunakan Metode Elemen Hingga 2 Langkah Taylor Galerkin (Universitas Gadjah Mada)
[8] Rahardjo A P 2002 Metode Elemen Hingga Hidro. (Yogyakarta: Jurusan Teknik Sipil, Fakultas Teknik, Universitas Gadjah Mada, Yogyakarta)
[9] O’Brien J S, Julien P Y and Fullerton W T 1993 Two-dimensional water flood and mudflow simulation J. Hydraul. Eng. 119 244–61
[10] Rickenmann D, Laigle D, McArdell B W and Hübl J 2006 Comparison of 2D debris-flow simulation models with field events Comput. Geosci. 10 241–64
[11] Uchida T, Nishiguchi Y, Nakatani K, Satofuka Y, Yamakoshi T, Okamoto A and Mizuyama T 2013 New numerical simulation procedure for large-scale debris flows (Kanako-LS) Int. J. Eros. Control Eng. 6 58–67
[12] Takahashi T 2007 Initiation and development of debris flow Debris Flow 102–67
[13] Luknanto D 2003 Model Matematika (Yogyakarta)