Two-dimensional numerical modeling of dam-break flows over natural terrain using a central explicit scheme

Jaswant Singh*1, Mustafa S. Altinakar2, Yan Ding3

*1 Research Scientist, National Center for Computational Hydroscience and Engineering, The University of Mississippi, Oxford, MS, 38655, USA, Email: singh@ncche.olemiss.edu

2Research Professor and Director, National Center for Computational Hydroscience and Engineering, The University of Mississippi, Oxford, MS, 38655, USA, Email: altinakar@ncche.olemiss.edu. [member IAHR]

3Research Assistant Professor, National Center for Computational Hydroscience and Engineering, The University of Mississippi, Oxford, MS, 38655, USA, Email: ding@ncche.olemiss.edu [member ASCE]

*1Corresponding Author

ABSTRACT

A two-dimensional (2D) numerical model has been developed to solve shallow water equations for simulation of dam-break flows. The spatial derivatives are discretized using a well-balanced explicit central upwind conservative scheme. The scheme is Riemann solver free and guarantees the positivity of the flow depth over complex topography if the Courant number is kept less than 0.25. The time integration is performed by Euler's scheme. The model is verified against analytical results for water surface elevation and discharge for three benchmark test cases. A good agreement between analytical solutions and computed results is observed. The property of well-balancing in still water over an uneven bottom is also confirmed. The model is then validated by simulating a laboratory experiment in which a dam break flow propagates over a triangular obstacle. The model performance was found to be satisfactory. A dam break laboratory experimental test case on a frictionless horizontal bottom is also simulated for 2D validation of the model, and good agreement between simulation and the experimental data is observed. The suitability of the proposed model for real life applications is demonstrated by simulating the Malpasset dam-break event, which occurred in 1959 in France. The computed arrival time of the flood wave front and the maximum flow depths at various observation points matched well with the measurements on a 1/400 scale physical model. The overall performance indicates that this model can be applied for simulation of dam-break waves in real life cases.

Key words: Shallow water equations; Central upwind schemes; Dam break flows; Malpasset dam break event; Flood simulations.

Nomenclature

Variable	Description	Dimensions
a^+	Right side local wave speed in a computational cell	[LT^{-1}]
a^-	Left side local wave speed in a computational cell	[LT^{-1}]
B	Bottom elevation	[L]
b^+	Top side local wave speed in a computational cell	[LT^{-1}]
b^-	Bottom side local wave speed in a computational cell	[LT^{-1}]
C	Chezy's coefficient	[$L^{1/2}T^{-1}$]
F	Vector of convective flux in x-direction	[L^2T^{-1}, L^3T^{-2}, L^3T^{-2}]
G	Vector of convective flux in y-direction	[L^2T^{-1}, L^3T^{-2}, L^3T^{-2}]
g	Acceleration due to gravity	[LT^{-2}]
h	Water depth	[L]
n	Manning's coefficient	[$L^{-1/3}T$]
N_{fl}	Courant number	
S	Vector of bottom source terms	[L^2T^{-2}, L^2T^{-2}]
S_{fx}	Friction source terms in x-direction	[L^2T^{-2},]
S_{fy}	Friction source terms in y-direction	[L^2T^{-2},]

Nomenclature

Variable	Description	Dimensions
U	Vector of primitive variables	[L, L^2T^{-1}, L^2T^{-1}]
$\tilde{U}$	Vector of piecewise linear reconstructed primitive variables	[L, L^2T^{-1}, L^2T^{-1}]
u	Velocity component in x-direction	[LT^{-1}]
v	Velocity component in y-direction	[LT^{-1}]
V_f	Wave front velocity	[LT^{-1}]
w	Water surface elevation	[L]
θ	Coefficient for minmod limiter to control the numerical diffusion	
Δx	Cell size in x-direction	[L]
Δy	Cell size in y-direction	[L]
Δt	Time step	[T]
ε	A priori chosen small positive number	

1. Introduction

When a semi-infinite water body initially at rest is released instantaneously by removal of a vertical barrier, such as in case of a dam failure, the resulting unsteady flow over a sloping or horizontal bed is termed as dam-break flow. In order to assess the consequences of a dam failure event, simulation of the resulting flood is of great importance. The spatial and temporal variations of water depths and velocities during a dam break event are important parameters for hydraulic engineers in order to prepare emergency action plans in a risk based framework. In recent years, there has been a substantial research emphasis on the development of numerical models to simulate dam-break flows. For instance, Altinakar et al. [1] developed a two-dimensional (2D) model coupled with a one-dimensional (1D) model for simulation of floods due to overtopping and breaching of levees; Caleffi et al. [2] used a finite volume method for modelling the extreme flood events in natural channels and Valiani et al. [3] modeled the Malpasset Dam break event using 2D finite volume method.

The dam-break flows can be modeled as unsteady free surface flows over a complex topography using depth-averaged, non-linear Shallow Water Equations (SWEs), also known as Saint-Venant equations. Ying et al. [4] developed numerical models for flows generated by a dam failure or levee breaching process using a conservative form of SWEs. There are many challenges in solving SWEs, such as treatment of wet and dry interface, treatment of bed elevation terms, mixed flow regimes and modeling the still water regions generally called as “lake at rest” conditions, etc. Brufau et al. [5] developed a numerical model for dam-break flows and resolved the wetting and drying of irregular terrain to a good extent. Zhou et al. [6] provided an adequate historic revision and those features required for a 2D river flow simulation model, such as treatment of bed elevation terms in sources terms and capturing the subcritical or supercritical conditions. Brufau et al. [7] developed a 2D numerical model for dam-break flows and achieved a zero mass error by modifying the wetting-drying condition which included the normal velocity to the cell edges. Most of these numerical models are based on Godunov-type schemes or lower order up-winding schemes, which produce significant numerical diffusion and do not necessarily admit steady state solutions in case of natural terrains, where the source terms due to bottom elevation gradient may become very important.

A variety of high-order well-balanced schemes for SWEs can be found in (Toro E.F [8], LeVeque [9], Kurganov and Levy [10] and Noelle et al. [11]). These schemes produce good approximation of the quasi-steady solutions and non-stationary steady states. The difficulty may occur where wet-dry interfaces are encountered in the solution domain. In this case, water depth (h) may become negative and the numerical computation will simply break down. Another difficulty with some of those schemes is the limitation of continuous bottom topography.

In the present paper, the 2D numerical model is developed solving SWEs which are members of the general class of equations called hyperbolic conservation laws. This system of equations admits steady-state solutions in which the non-zero flux terms are balanced by the sources terms. They also admit solutions that involve discontinuous and nonlinear waves, such as shocks and rarefactions, as well as wet-dry interfaces generated by dam-break flows [11]. The well-balanced central upwind scheme proposed by Kurganov and Petrova [12] is adopted for the present model as this scheme produces less numerical diffusion. This central scheme achieves up-winding by considering one-sided wave speeds, which further reduces the numerical diffusion.

If the Courant number is kept less than 0.25 for a 2D case, this scheme guarantees the positivity of the depth in simulations of supercritical flows propagating over a dry bed. The source terms due to bottom elevation are discretized in such a way that they cancel the flux terms in case of steady state condition; hence, the lake at rest condition can also be captured. This type of treatment of source terms results in a well-balanced scheme.

Such important properties make this scheme attractive for development of numerical models for simulation of dam break flows. Instead of using simplified linear bottom friction terms considered in the original scheme by Kurganov and Petrova [12], the non-linear bottom friction terms considering Manning's roughness coefficient are adopted in the present model. The non-linear bottom friction terms, which are capable of considering the influence of drag force by the bottom roughness, ensure that the developed model is applicable to simulate the real life dam break flows. By using the digital elevation model (DEM) data as the direct input of natural terrain data, this model can also be integrated with Geographic Information System (GIS) for developing applications to solve real life dam-break flow problems.

This paper is organized as follows: In section 2, the mathematical formulation, numerical scheme including the governing equations, solution scheme, its properties and the treatment of two source terms (bottom elevation gradient and bottom friction terms) are presented. In Section 3, the model verification is presented by testing the performance of the model against the standard benchmark test cases. Two laboratory experimental cases of dam break flow are simulated for 1D and 2D validation of the proposed model. In section 4, the applicability of the model to simulate real life cases is examined by simulating the Malpasset Dam break event. Section 5 gives the conclusions for this research work.

2. Dam-Break Flow Model Development

2.1 Governing Equations

The conservative form of 2D shallow water equations, consisting of a continuity equation and two momentum equations for depth-averaged free surface flows, are written as:

$$\begin{bmatrix} h \\ hu \\ hv \end{bmatrix}_t + \begin{bmatrix} hu \\ hu^2 + \frac{1}{2}gh^2 \\ huv \end{bmatrix}_x + \begin{bmatrix} hv \\ huv \\ hv^2 + \frac{1}{2}gh^2 \end{bmatrix}_y = \begin{bmatrix} 0 \\ -gh\frac{\partial B}{\partial x} - S_{fx} \\ -gh\frac{\partial B}{\partial y} - S_{fy} \end{bmatrix}, \tag{1}$$

where *x* and *y* are Cartesian coordinates describing the horizontal plane, *B(x,y)* is the bottom elevation function describing an arbitrary or natural bathymetry, *t* is time, *h(x,y,t)* is water depth, *u(x,y,t)* and *v(x,y,t)* are the two components of the depth-averaged velocities in *x* and *y* directions, respectively. The gravitational acceleration is denoted by *g*. The components of the non-linear bottom friction terms due to its roughness in *x* and *y* directions are defined as:

$$S_{fx} = gu\sqrt{u^2 + v^2} / C^2, \tag{2}$$

$$S_{fy} = gv\sqrt{u^2 + v^2} / C^2. \tag{3}$$

where *C* is the Chezy's coefficient and is calculated as $C = (h)^{1/6} / n$ and *n* is the Manning's coefficient. The water surface elevation is represented by *w(x,y,t)* such that *w* = *h*+*B* (see figure 1). The bottom elevation *B(x,y)* is assumed not to change with time, i.e., fixed bed is considered in this model. By replacing water depth (*h*) by the difference of surface water elevation and bottom elevation (*w-B),* and carrying out algebraic manipulations, Eq.1 is rewritten in terms of *w(x,y,t).*

$$\begin{bmatrix} w \\ hu \\ hv \end{bmatrix}_t + \begin{bmatrix} hu \\ \dfrac{(hu)^2}{w-B} + \dfrac{1}{2} g (w-B)^2 \\ \dfrac{(hu)\,(hv)}{w-B} \end{bmatrix}_x + \begin{bmatrix} hv \\ \dfrac{(hu)\,(hv)}{w-B} \\ \dfrac{(hv)^2}{w-B} + \dfrac{1}{2} g (w-B)^2 \end{bmatrix}_y = \begin{bmatrix} 0 \\ -g(w-B)\dfrac{\partial B}{\partial x} - S_{fx} \\ -g(w-B)\dfrac{\partial B}{\partial y} - S_{fy} \end{bmatrix}, \quad (4)$$

Eq.4 can be written in vector form as follows:

$$U_t + F(U,B)_x + G(U,B)_y = S(U,B), \qquad (5)$$

where *U, F(U,B), G(U,B)* and *S(U,B)* are the vectors of primitive variables, fluxes in x and y direction and sources. They are defined as follows:

$$U = \begin{bmatrix} w \\ hu \\ hv \end{bmatrix}, \quad F(U,B) = \begin{bmatrix} hu \\ \dfrac{(hu)^2}{w-B} + \dfrac{1}{2} g (w-B)^2 \\ \dfrac{(hu)\,(hv)}{w-B} \end{bmatrix}, \; G(U,B) = \begin{bmatrix} hv \\ \dfrac{(hu)\,(hv)}{w-B} \\ \dfrac{(hv)^2}{w-B} + \dfrac{1}{2} g (w-B)^2 \end{bmatrix},$$

$$S(U,B) = \begin{bmatrix} 0 \\ -gh\dfrac{\partial B}{\partial x} - S_{fx} \\ -gh\dfrac{\partial B}{\partial y} - S_{fy} \end{bmatrix}.$$

2.2 Numerical Scheme

The discretization of the above equations is based on a finite volume method. A staggered grid is used in which the primitive variables are defined at the cell center, and the bottom elevation is defined at the cell corner as shown in figure 2. The bottom is represented as a bilinear surface. In order to facilitate the direct input of a DEM data-set, for natural terrains, a uniform grid of size Δx in x-direction and Δy in y-direction is adopted. The computational cell (i,j) is defined as $\left\lfloor x_{i-1/2,j} x_{i+1/2,j}, y_{i,j-1/2}, y_{i,j+1/2} \right\rfloor$ where $x_i = i.\Delta x$ and $y_j = j.\Delta y$. The locations represented by $\left[x_{i+1/2,} y_j\right]$, $\left[x_{i-1/2,} y_j\right]$, $\left[x_{i,} y_{j-1/2}\right]$, *and* $\left[x_{i,} y_{j-1/2}\right]$ designate the four mid-points of the sides of the cell volume in both x and y direction. These four mid-point locations are superscripted as east (^{E}), west (^{W}), north (^{N}) and south (^{S}) in the equations. The adopted scheme to solve the system of Eq.5 is as follows:

$$\frac{d}{dt} U_{i,j}(t) = -\frac{H^x_{i+\frac{1}{2},j}(t) - H^x_{i-\frac{1}{2},j}(t)}{\Delta x} - \frac{H^y_{i,j+\frac{1}{2}}(t) - H^y_{i,j-\frac{1}{2}}(t)}{\Delta y} + S_{ij}(t), \tag{6}$$

The central-upwind numerical fluxes, H^x and H^y are given below.

$$\begin{aligned} H^x_{i+\frac{1}{2},j} &= \frac{a^+_{i+\frac{1}{2},j} f(U^E_{i,j}, B(x_{i+\frac{1}{2}}, y_j)) - a^-_{i+\frac{1}{2},j} f(U^W_{i+1,j}, B(x_{i+\frac{1}{2}}, y_j))}{a^+_{i+\frac{1}{2},j} - a^-_{i+\frac{1}{2},j}} \\ &+ \frac{a^+_{i+\frac{1}{2},j} a^-_{i+\frac{1}{2},j}}{a^+_{i+\frac{1}{2},j} - a^-_{i+\frac{1}{2},j}} \left[U^W_{i+1,j} - U^E_{i,j} \right], \end{aligned} \tag{7}$$

$$H^{y}_{i,j+\frac{1}{2}} = \frac{b^{+}_{i,j+\frac{1}{2}} f(U^{N}_{i,j}, B(x_i, y_{j+\frac{1}{2}})) - b^{-}_{i,j+\frac{1}{2}} f(U^{S}_{i,j+1}, B(x_j, y_{j+\frac{1}{2}}))}{b^{+}_{i,j+\frac{1}{2}} - b^{-}_{i,j+\frac{1}{2}}} + \frac{b^{+}_{i,j+\frac{1}{2}} b^{-}_{i,j+\frac{1}{2}}}{b^{+}_{i,j+\frac{1}{2}} - b^{-}_{i,j+\frac{1}{2}}} \left[U^{S}_{i,j+1} - U^{N}_{i,j} \right] , \tag{8}$$

Where $a^{\pm}_{i\pm 1/2, j\pm 1/2}, b^{\pm}_{i\pm 1/2, j\pm 1/2}$ are the local one-sided speeds of the propagation in x and y directions respectively and are calculated as follows:

$$a^{+}_{i+1/2,j} = \max\left\{ u^{E}_{i.j} + \sqrt{gh^{E}_{i,j}}, u^{W}_{i+1.j} + \sqrt{gh^{W}_{i+1,j}}, 0 \right\},$$

$$a^{-}_{i+1/2,j} = \min\left\{ u^{E}_{i.j} - \sqrt{gh^{E}_{i,j}}, u^{W}_{i+1.j} - \sqrt{gh^{W}_{i+1,j}}, 0 \right\},$$

$$b^{+}_{i,j+1/2} = \max\left\{ v^{N}_{i.j} + \sqrt{gh^{N}_{i,j}}, v^{S}_{i.j+1} + \sqrt{gh^{S}_{i,j+1}}, 0 \right\},$$

$$b^{-}_{i,j+1/2} = \min\left\{ v^{N}_{i.j} - \sqrt{gh^{N}_{i,j}}, v^{S}_{i.j+1} - \sqrt{gh^{S}_{i,j+1}}, 0 \right\}.$$

$U^{E,W,N,S}$ are the four mid-point values of the piecewise linear reconstruction function $\tilde{U}(x, y)$ at the locations defined as $\left[x_{i+1/2,} y_j\right]$, $\left[x_{i-1/2,} y_j\right]$, $\left[x_{i,} y_{j-1/2}\right]$, *and* $\left[x_{i,} y_{j-1/2}\right]$. This non-oscillatory piecewise polynomial reconstruction function $\tilde{U}(x, y)$ for each cell can be defined as follows:

$$\tilde{U}(x, y) = \overline{U}_{ij} + (U_x)_{ij}\,(x - x_i) + (U_y)_{ij}(y - y_j) \ , \tag{9}$$

where $\overline{U}_{ij}$ is the mean value of the primitive variables (*w*, *hu*, *hv*) at the cell center. The numerical derivatives $(U_x)_{i,j}$ and $(U_y)_{i,j}$ are at least first-order component-wise approximations of $U_x(x_i, y_j, t)$ and $U_y(x_i, y_j, t)$. They are computed using a non-linear limiter function commonly known as the minmod limiter.

$$(U_x)_{i,j} = \text{minmod}\left(\theta\frac{\overline{U}_{i,j} - \overline{U}_{i-1,j}}{\Delta x}, \frac{\overline{U}_{i+1,j} - \overline{U}_{i-1,j}}{2\Delta x}, \theta\frac{\overline{U}_{i+1,j} - \overline{U}_{i,j}}{\Delta x}\right), \tag{10}$$

$$(U_y)_{i,j} = \text{minmod}\left(\theta\frac{\overline{U}_{i,j} - \overline{U}_{i,j-1}}{\Delta y}, \frac{\overline{U}_{i,j+1} - \overline{U}_{i,j-1}}{2\Delta y}, \theta\frac{\overline{U}_{i,j+1} - \overline{U}_{i,j}}{\Delta y}\right), \tag{11}$$

where θ is a coefficient to control numerical diffusion. As Kurganov and Petrova [12] suggested that θ =1.3 can provide the best simulation results, this value is adopted in all the computations in the present study. The minmod function is defined as:

$$\text{minmod}(z_1, \ldots\ldots, z_n) = \begin{cases} \max z_i, & z_i < 0\ \forall i, \\ \min z_i, & z_i > 0\ \forall i, \\ 0, & otherwise. \end{cases} \tag{12}$$

The overshoot and undershoots in the reconstruction of primitive variables are prevented using this limiter.

The proposed scheme (Eq.7 and 8) is a Godunov-type central scheme which is based on integration over Riemann fans and is second order accurate. This scheme does not require characteristic decomposition and Riemann solvers; thus it preserves the advantage of simplicity in central schemes. The upwinding nature of the scheme is demonstrated as the direction of wave propagation is taken into consideration by measuring the one-sided local speeds. That is the reason that this scheme is called a central-upwinding scheme. If the fluxes are monotone, the scheme reduces to standard upwind type.

The discretization of source terms is of crucial importance as the non-zero component of the flux terms needs to be balanced with the bottom elevation source terms in case of steady state solutions. The *w(x,y,t)* remains constant for the steady state condition. The source terms due to bottom elevation gradient are discretized as follows:

$$\overline{S}^{(2)}_{i,j}(t) \approx -g\,\frac{B(x_{i+\frac{1}{2}},y_j)-B(x_{i-\frac{1}{2}},y_j)}{\Delta x}\cdot\frac{(w^{E}_{i,j}-B(x_{i+\frac{1}{2}},y_j))+(w^{W}_{i,j}-B(x_{i-\frac{1}{2}},y_j))}{2}\ ,$$

$$\overline{S}^{(3)}_{i,j}(t) \approx -g\,\frac{B(x_i,y_{j+\frac{1}{2}})-B(x_i,y_{j-\frac{1}{2}})}{\Delta y}\cdot\frac{(w^{N}_{i,j}-B(x_i,y_{j+\frac{1}{2}}))+(w^{S}_{i,j}-B(x_i,y_{j-\frac{1}{2}}))}{2}\ . \qquad (13)$$

In order to guarantee the positivity of the flow depth at the cell center, the slope of newly reconstructed primitive variables is compared with the slope of the bottom elevation. If the reconstructed values of primitive variables at the mid-points of the cell interface is less than that of bottom elevation at the cell interface mid-points, the slope is adjusted explicitly equal to the bottom slope in order to assure that the flow depth remains always positive. Like other explicit schemes, this scheme is subjected to the Courant-Friedrichs-Lewy (CFL) condition for stability. This condition states that during a time step, the wave (or a perturbation) should not travel a distance longer than the quarter of smallest cell size. Given a mesh size, the CFL condition for this model places an upper bound on the time step as follows:

$$N_{cfl} = \text{Max}\left[\frac{\Delta t}{\Delta x}(|u| + \sqrt{gh}), \frac{\Delta t}{\Delta y}(|v| + \sqrt{gh})\right] \le 0.25 \ , \tag{14}$$

where N_{cfl} is the Courant number. When a dam with a reservoir of water depth h breaks instantaneously and completely, the tip of the flood front advancing over a dry terrain has an initial velocity (V_f) that can be calculated as follows:

$$V_f = 2\sqrt{gh} \ , \tag{15}$$

The model searches for the largest initial water depth in the computational domain and uses it as the depth (h) in Eq. 14 to compute the initial time step. Using V_f and the N_{cfl} the time step *(*Δt*)* is determined. If the computed water depth in any cell is infinitely small or zero, the velocities required in flux terms and local speeds of propagation (refer Eq.4) will not be computed correctly. Also the computations will slow down dramatically as the Δt computed from very large velocities will be extremely small. These difficulties are overcome by desingularizing the quantity *(hu)/h* and *(hv/)h* using the formulae (adopted from Kurganov and Petrova [12]) as follows:

$$u = \frac{\sqrt{2}h(hu)}{\sqrt{h^4 + \max(\varepsilon, h^4)}} \ , \tag{16}$$

$$v = \frac{\sqrt{2}h(hv)}{\sqrt{h^4 + \max(\varepsilon, h^4)}} \, . \tag{17}$$

where ε is a small a-priori chosen positive number. When h ≥ $\varepsilon^{1/4}$, Eq. 16 and 17 reduces to (*hu)/h* and *(hv/)h* calculating the exact values of velocities. In the cases where h < $\varepsilon^{1/4}$, the computed velocities are corrected by Eq. 16 and 17. We have used $\varepsilon=10^{-8}$ in all of our simulations in this paper.

The time integration is carried out using Euler's method. In order to keep the positivity preserving quality of the scheme, the Courant number = 0.25. Hence the time step is sufficiently small in order to make the Euler's method stable for all the applications used in this paper. For other details of the proposed scheme such as the order of accuracy and detailed error analysis, we refer the reader to [12].

3. Verification and Validation of Numerical Model

Based on the above numerical scheme, a numerical model for solving SWEs has been developed by using the C++ programming language. This model is verified by simulating three one-dimensional established benchmark test cases of flow over a bump in an idealized channel. Since the analytical solutions exist for these benchmark test cases, the model's accuracy in dealing with various complicated flow situations can be investigated. The verified model is then validated by simulating a dam-break flow over a triangular obstacle in an experimental flume. After verifying the model performance for test cases of one-dimensional flow, simulation of two-dimensional flow was carried out for a 2D test case published in the literature. Finally, the model's applicability in a real life scenario is examined by simulating the case of the Malpasset dam-break event which occurred in 1959 in France.

3.1 Model verification using 1D analytical solutions

We have used three established benchmark test cases for verification of this model. These test cases involve transcritical, supercritical and subcritical flows in a 25m channel with a bump. The bed profile for this bump is defined by Eq. 18.

$$B(x) = \begin{cases} 0.2 - 0.05(x-10)^2 & \text{if } 8 < x < 12, \\ 0 & \text{otherwise.} \end{cases} \tag{18}$$

The computational domain consist of 200 uniform cells of size Δx = 0.125m. The time step is calculated based on the N_{cfl} criteria. The numerical results are compared with analytical results taken from Goutal and Maurel [13].

Case 1: Transcritical flow without a shock

The initial water surface elevation in the channel is 0.33m. A unit discharge of 1.53 m^2/s is imposed at the upstream boundary, and the open boundary conditions ($dw/dx=0$, $dv/dx=0$) are applied at the downstream side. The steady state condition is reached after 41.5 s. Figure 3 shows the analytical and numerical solutions for the water surface elevation profile and discharge after the steady state has been reached. The water surface elevation starts to drop beginning at the upstream end of the bump and reaches a constant level immediately downstream of the bump. The simulated water surface elevation shows good agreement with the analytical results. The simulated discharge is slightly underestimated at the location of the bump. This fluctuation is local and does not propagate in the solution over the domain. The test Case 2 in the following paragraph gives more detailed description about this discrepancy in the simulated discharge.

Case 2: Transcritical flow with a shock

In this case, the initial water surface elevation in the channel is 0.33m. A unit discharge of 0.18 m^2/s is imposed on the upstream boundary and a depth of 0.33m is imposed on the downstream boundary. A steady state is reached at about 175.5 s. Figure 4 shows the analytical and numerical solution for water surface profile and discharge for this case. The drop of water surface elevation due to the bump is captured well in the numerical results. The simulated water surface elevation at the bump matched very well with the analytical solution. There is a local peak in the numerical solution for unit discharge at the location of the jump downstream of the bump.

This discrepancy in discharge has been reported in other schemes as well. (see Zhou *et al*. [6]; Vazquez-Cendon [14]; Hubbard and Garcia-Navarro [15]; Xu [16]; Delis [17]; Rogers *et al.* [18]; Delis and Katsaounis [19]). One of the reasons for the discrepancy in the computed discharge is in the mathematical formulations of the governing equations: The mass flux (*hu*) in the continuity equation and the primitive variable discharge (*hu*) in the momentum equation, have same mathematical form but are different in terms of physical meaning. The mass flux (*hu*) is calculated at the interface of the cell from the continuity equation and the primitive variable discharge *(hu)* is obtained at the center of the cell from the momentum equations. The fluctuation in the primitive variable discharge *(hu)* is expected at the location of the bump because the momentum (*hu*) has to satisfy with the momentum conservation law. However, on the other hand, mass flux (*hu*) in the continuity equation should remain constant for the constraint of mass conservation. We have compared both of these quantities in figure 5 for this case of transcritical flow with a shock. It can be observed that the mass flux is constant (mass-conservative) throughout the channel and the primitive variable discharge *(hu)* varies at the location of discontinuity of water surface to satisfy momentum conservation.

Case 3: Subcritical flow over a bump without a shock

In this case, the subcritical flow conditions are maintained in the same channel as taken in the above two cases. A unit discharge of 4.42 m^2/s is imposed on the upstream boundary and a depth of 2.0 m is imposed on the downstream boundary. The steady state condition reaches in 85.5 s. Figure 6 shows the comparison of the analytical and numerical solution for this case after reaching a steady state. The drop in the water surface elevation profile at the location of the bump is captured well by the model. There is an excellent agreement between the simulated water elevation and the analytical results. The computed discharge matched well with the analytical solution. However, in this case too, local discrepancy of discharge is observed at the location of the bump due to the same reason as explained in Case 2. From the above three test cases, it is verified that the model produced accurate results in all the flow regimes.

Case 4: Well-balance property test

As discussed in section 2.2, the discretization of source terms is performed in such a way that in still water, the non-zero component of the flux terms is balanced by the source terms dues to bottom gradient. To test this well-balancing property of the proposed scheme, a severe test is designed. The test domain is 50 m long and the water surface elevation is 10 m. The bottom is defined as follows:

$$B(x)=\begin{cases}-0.0513x+4 & \text{if } \; 0\le x\le 15,\\ -0.0762x^2+3.2108x-27.766 & \text{if } \; 16\le x\le 29,\\ 0.4824x-14.472 & \text{otherwise.}\end{cases}$$

(19)

The Δx =1.0 m and the initial depth is assigned as the difference of water surface elevation and the bottom elevation. The maximum depth in the domain is 10 m at x =30 m and minimum depth is zero at the downstream end. The maximum slope of the bottom is 0.4824. The simulation was carried out for 10 seconds. Figure 7 shows water surface elevation, depth and velocity. The water surface elevation remains at 10 m and there is no change in the depth. In the whole domain velocity remained zero. The test proved that the proposed scheme is well-balanced.

3.2 Model validation using 1D experimental case

In order to further examine the performance of this model, one case of dam-break flow generated in laboratory experiments is simulated. The experimental data are taken from Liang and Marche [20]. This experimental study was conducted at Hydraulic Research Laboratory, Châtelet, together with The University of Brussels in Belgium under the supervision of J.M. Hiver. The dam-break wave is generated by instantaneous removal of a gate impounding a clear water reservoir. The subsequent evolution of the resulting flood wave over a triangular bump is investigated for water depth as a function of time at various gauge points in the downstream region. The schematic layout of the experiments is shown in figure 8.

The experiments were conducted in a rectangular horizontal channel of 38 m length. The gate is located at 15.5 m and the water depth in the reservoir is 0.75 m. The width of the flume is assumed to be 10 cells in y-direction. A 6.0 m long and 0.4 m high triangular obstacle with symmetric slopes is located at 13.0 m downstream of the dam. The channel has a fixed bed with dry initial conditions. The side boundaries are solid walls, and the down-stream boundary condition is open boundary. The mesh size in x and y directions is 0.01 m. The Manning's coefficient n is 0.0125 which is adopted from Liang and Marche [20]. The value of $\varepsilon = 10^{-8}$. The flow depths are measured at 6 observation stations, i.e., GP2, GP8, GP10, GP11, GP13, GP20 which are located at 2, 8, 10, 11, 13 and 20 meters downstream of the dam respectively, and are shown in figure 8.

3.2.1 Results and discussion

The comparisons of simulated and observed water depths are discussed in this section. Figure 9 shows the comparison of water depths at six selected observation stations. The flow regime changes from subcritical to transcritical, and reaches to supercritical flow at various observation stations as the dam-break flow propagates downstream. As the gate is removed instantaneously, the water depth rises with time and then after about 10 seconds, starts falling sharply at GP2. As the flow passes over the triangular obstacle, a reflective negative wave front is generated, which starts traveling upstream and the depth at GP2 rises forming a peak. The same phenomenon causes the formation of other peaks at GP10 and GP11 observation points. The comparison of the experiments and the simulated results show a good match at the first four observation stations. The temporal variation of water depth at the tip of the obstacle is shown at GP13. At this location, there is no reflective wave front, but progressive wave fronts from the upstream side cause the formation of peaks. The model produced a good match with the experimental water depths at this station also.

A discrepancy in water depth between the experiments and the simulations is observed at GP20. As the model overestimated the flow upstream of the obstacle, more water than the reality passes over the downstream side of the obstacle. This leads to higher depth at GP20. There is a chance of experimental errors too as this discrepancy is also reported by other researchers as well ([5], [19]). Even if the water depth is slightly overestimated at GP20, the times of arrival of flood peaks due to progressive wave fronts after the obstacle is captured well. At all the other observation points, the model produced good results. It can be concluded that, the model is capable of capturing the dam break flow with a reasonable accuracy.

3.3 Model validation using idealized 2D experimental case

The 2D validation of the proposed model is performed by simulating the laboratory experiment case of flood wave propagation on a horizontal bottom due to partial dam break reported by Fraccarollo and Toro [21]. The schematic view of the experimental set is shown in figure 10. The reservoir is 1m wide and 2m long with a downstream dry horizontal flood plain of 3 m length. The dam is 2 m wide with 0.4 m breach located in the middle. In the selected case, the water depth in the reservoir is 0.6 m and the downstream flood plain is dry. The location of 5 measuring gauges is shown in Table 1. The dam breaches instantaneously.

The above experimental case is represented on a mesh of 400 x 200 points with a Δx = Δy 0.01m. The dam is defined by raising the bottom elevation of two columns of cells at the dam location. The value of $\varepsilon = 10^{-8}$. The boundary conditions are no flow type at the location of the reservoir and open along the all sides of the flood plain. The observed water depth at 5 measuring stations is compared with the simulations and the results are shown in figure 10. As the dam breaches instantaneously, a surge wave propagates in the downstream flood plain. The rarefaction wave propagates towards boundaries causing the water level to drop in the reservoir. The stations 5A and C are equidistant from the center of the breach; thus the depth hydrographs at these locations are almost similar. The simulated results of flood depth at these two locations are in good agreement with the experiments. Station 4 is located at the left end of the dam where the streamlines are curved. The model slightly underestimated the flow depth in the initial phase of the dam break wave propagation. The reason for this can be attributed to the inability of shallow water equations to handle non-hydrostatic pressure due to curvature of the streamlines. However, after 2 seconds, the model reproduced the flow depth reasonably well. At measuring station 0, which is in the center of the breach, the model slightly underestimated the flow depth. The slight discrepancy immediately after the opening of the gate is normal considering that the SWEs cannot describe the complex flow conditions created by the friction of the gate on the water impounded behind and the non-hydrostatic pressure. The flow depth at a downstream station, 8A, is reproduced up to a reasonable accuracy. Overall, the performance of the model in simulating a 2D ideal dam break experimental case can be termed as good.

4. Malpasset Dam-Break Simulation

After verification and validation of the model by analytical and experimental solutions, the model is applied to simulate one real life case of the Malpasset Dam break event.

4.1 Malpasset Dam break case

The Malpasset Dam was constructed on Reyran River in 1954 for the purpose of irrigation and drinking water supply to the town of Frejus, a French town upstream of the Côte d'Azur area. This arch dam had a crest height of 66 m and crest length of 223 m. The maximum reservoir capacity was $5.5\text{x}10^7$ m^3. After an intense rainfall in December 2nd, 1959, the Malpasset Dam failed giving rise to a huge flood wave of 40-m height. This catastrophic flood resulted in the death of 421 people and considerable property damage in the inundated region downstream. Several investigations were conducted to investigate the cause of the dam failure. It was reported that exceptional rainfall and insufficient geologic information on the dam site resulted in the disaster.

A physical model of 1/400 scale was built in 1964 for performing a detailed study of this historical dam-break flood event. The front arrival time and the maximum water depth were measured at 9 observation points in the physical model. The location of the observation points and reservoir are shown in figure 11. There were three electric transformers (denoted as A, B and C in this paper) on the downstream side of the dam which were shut down due to arrival of the flood wave. The exact shutdown times of those transformers are available in the records.

4.2 Computational conditions for Malpasset dam-break case

The DEM, initial conditions such as water surface elevation in the reservoir, the coordinates of the dam, transformers and the observation points in the physical model were obtained from Ying et al. [4]. The simulation is carried out on two computational uniform grids for analyzing the grid dependency of the solution. One computational grid has 550×220 cells with a uniform cell size of 30 m, and the other grid has 1100×440 cells with a uniform cell size of 15 m. A spatially constant value of Manning’s coefficient is set equal to 0.033 taken from Valiani et al. [3]. The tolerance $\varepsilon = 10^{-8}$ meter was considered in the simulations. Along the edges of the model, closed boundary condition is used if it is land. Constant water surface elevation boundary condition is used where the edge of the model are in contact with the Mediterranean Sea (upper right hand corner, figure 11). Since there is no data on the actual breaching process of the dam, it was assumed that the dam failure occurred instantaneously.

4.3 Results and discussions

The results of the simulations are presented in this section. Figure 11 (top) shows the location of the observation points and transformers in the domain. There were 9 observation points which were named as S6 to S14. The snapshots of simulated flow depth at 700s and 2000s presented in figure 11 (middle and bottom) show that the model gives a realistic prediction of the flood at the downstream area. Figure 12 shows the complete flow process of Malpasset dam-break event as an animation. The water depth file is super-imposed on the bottom topography file (DEM) of the terrain. The spatial and temporal distribution of flow depths starting from the instantaneous dam break event to the flow reaching at the down-stream boundaries shows that the model is capable of simulating dam-break floods in real life case. It can be observed from the animation that the water remains at zero velocity in the ocean near down-stream boundary. This demonstrates the simulation of “lake-at-rest” capability of the model.

Table 2 shows the comparison of simulated flood propagation time with field-observed data at the location of the three electric transformers. The model predicted the front arrival time to a good accuracy as the relative error shown in Table 2 is very low. When the finer mesh is used, the simulation results are improved and relative error is reduced substantially.

Figure 13 shows the comparison of simulations with physical model observations for maximum depth and flood propagation time at nine observation points. The model reproduced the flood fronts which reached the observation points at almost the same time as the flood in the physical model experiment. At the last point, which is the farthest point in the physical domain, the model slightly over predicted the flood front propagation time. However, the finer mesh results improved this discrepancy. This discrepancy can probably be attributed, at least partially, to the use of a constant Manning's *n* over the entire computational domain. The results of Valiani et al. [3] with high resolution grid are also included in the figure 13 for comparison with the proposed model. It is observed that the maximum water level predicted by the proposed model is matching with the physical model and results of Valiani et al. [3] up to a reasonable accuracy. However, the coarse mesh over predicted the water level at station S9. But the finer mesh produced even better results than Valiani et al. [3] at this observation point. From the comparisons of the simulated results with the physical model observations and with the Valiani et al. [3] simulations, it can be concluded that the model is capable of capturing the hydrodynamic behavior of flood flows to a high accuracy level, and can be used for real life applications for impact assessment of dam-break hazards.

5 Conclusions

A 2D numerical model is developed to numerically solve shallow water equations using an explicit central upwind, well-balanced and flow depth positivity-preserving, second-order accurate scheme. The source terms of the original scheme are modified to represent the bottom friction in natural terrains more realistically. The scheme does not require characteristic decomposition and Riemann solvers, and is easy to implement.

Various 1D and 2D numerical tests are performed in order to establish the accuracy of the proposed model to simulate the flood propagation in real life. The numerical solution shows a good agreement with analytical solutions and jump location is predicted accurately in cases of flow over a bump. The well-balanced property of the proposed scheme is also tested by simulating still water over uneven bottom topography. The model is found to be well balanced.

The performance of the proposed model is verified by simulating 1D and 2D laboratory experiments and the real life case of the Malpasset Dam break event. The results show that there is good agreement between the experiments and simulations. The numerical simulation results are shown to have good agreement with both laboratory experiments and field observation. The present numerical model can, therefore, be used for studying real life problems.

6 Future works

Currently work is underway to include source and sink terms into the model to simulate overland flow, resulting sediment erosion and transport by taking into account spatially and temporally varied precipitation and infiltration.

ACKNOWLEDGEMENT: This research was funded by the Department of Homeland Security-sponsored Southeast Region Research Initiative (SERRI) at the Department of Energy's Oak Ridge National Laboratory, USA. The valuable suggestions provided by the anonymous reviewers are also thankfully acknowledged.

References:

[1] Altinakar MS, Miglio E, Wu W. Coupling 1D-2D shallow water models for simulating floods due to overtopping and breaching of levees. Proceedings of the 8th world congress on computational mechanics. 2008; WCCM8. Venice, Italy, June 30- July 5.

[2] Caleffi V, Valiani A, Zanni A. Finite volume method for simulating extreme flood events in natural channels. IAHR J Hydraulic Res 2003; 41(2): 167-277.

[3] Valiani A, Caleffi V, Zanni A. Case study: Malpasset dam-break simulation using two-dimensional finite volume method. J Hydraul Eng. 2002; 128: 460-472.

[4] Ying X, Wang SSY, Khan AK. Numerical simulation of flood inundation due to dam and levee breach. Proceedings of ASCE world water and environmental resources congress 2003, Philadelphia, USA.

[5] Brufau P, Vazquez-Cendon ME, Garcia-Navarro P. A numerical model for flooding and drying of irregular domains. Int J Numer Methods in Fluids 2002; 39: 247-275.

[6] Zhou JG, Causon DM, Mingham CG. Ingram DM. Surface gradient method for treatment of source terms in the shallow-water equations. J Comput. Phys. 2001;168:1-25.

[7] Brufau P, Garcia-Navarro P, Vazquez-Cendon ME. Zero mass error using unsteady wetting-drying conditions in shallow flows over dry irregular topography. Int J Numer Methods in Fluids 2004;45:1047-1082.

[8] Toro EF. Shock-capturing methods for free-surface shallow flows. New York: Wiley; 2001.

[9] LeVeque RJ. Balancing source terms and flux gradients in high resolution Godunov methods: the quasi-steady wave-propagation algorithm. J Comput. Phys. 1998;146:346-365.

[10] Kurganov A, Levy D. Central-upwind scheme for the saint-venant system. M2AN Math Model Numer Anal 2002;36:397-425.

[11] Noelle S, Pankratz N, Puppo G, Natvig J. Well-balanced finite volume scheme for arbitrary order of accuracy for shallow water flows. J Comput. Phys. 2006;213:474-499.

[12] Kurganov A, Petrova G. A second order well balanced positivity preserving central upwind scheme for the saint-venant system. Commun. Math Sci 2007;133(5(1)):133-160.

[13] Goutal N, Maurel F, editors. Proceedings of the 2nd workshop on dam-break wave simulation. HE-43/97/016/B, France; 1997.

[14] Vazquez-Cendon ME. Improved treatment of source terms in upwind schemes for the shallow water equations. J. Comput. Phys. 1999;148:497-526.

[15] Hubbard M, Garcia-Navarro P. Flux difference splitting and the balancing of source terms and flux gradients. J Comput. Phys. 2000;165:89-125.

[16] Xu K. A well-balanced gas-kinetic scheme for the shallow water equations with source terms. J Comput. Phys. 2002;178:533-562.

[17] Delis AI. Improved application of the HLL Riemann solver for the shallow water equations with source terms. Commun. Numer. Meth. Engrg. 2006;19: 59-83.

[18] Rogers BD, Borthwick AGL, Taylor PH. Mathematical balancing of flux gradient and source terms prior to using Roe's approximate Riemann solver. J Comput. Phys. 2003;192:422-451.

[19] Delis AI, Katsaounis Th. Relaxation schemes for the shallow water equations Int J Numer Methods in Fluids 2003;41:695-719.

[20] Liang Q, Marche F. Numerical resolution of well-balanced shallow water equations with complex source terms. Adv in Water Res 2009; 32:873-884.

[21] Fraccarollo L, Toro EF. Experimental and numerical assessment of the shallow water model for two-dimensional dam-break type problems. J Hyd. Res 1995; 33: 843-864.