IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X.
PP 37-41
www.iosrjournals.org

Numerical Simulation of a Dam Break Flow Using Finite Difference Method

Saudamini Naik[1], K.K. Khatua[2]
[1,2]*(Civil Engineering, National institute of technology Rourkela, India)*

Abstract: *Analysis of a dam break flow numerically is an essential part of hydraulic engineering practice. Estimation of peak flood flow depth, its time of occurrence at a specified location, wave fronts and assessment of its fetch can be done through numerical models. In the following paper a numerical method based on the Mac Cormack finite difference scheme is presented. The approach was followed for simulating two-dimensional shallow water equation presenting a dam break flow. This paper describes the use of the Mac Cormack explicit time-splitting scheme in the development of a two-dimensional (in plan) hydraulic simulation model that solves the St. Venant equations. The basic Mac Cormack scheme is enhanced by using the method of fractional steps forsimplifying application, treating the friction slope, a stiff source term, point-implicitly, for numerical oscillation control and stability which is validated by comparing the current data with the benchmark results, and good agreement is achieved in the case of a partial dam-break simulation. The present numerical analysis is able to resolve shocks, complex bed geometry including the influence of bed slopes and roughness. Here MATLAB software is used for coding and mesh generation.*
Keywords*: Dam break, Maccormack method, SWE, finite difference method*

I. Introduction

A wide variety of physical phenomenons are governed by mathematical models based on shallow water equations. An important class of problems of practical interest involves water flows with the free surface under the influence of gravity. This class includes tides in ocean, breaking of waves in shallow beaches, flood waves in rivers, surges of dam break wave modelling etc. Although safety criteria have been considered in design, construction and operation of dams, dams may be broken under unpredictable events. If the water surface elevations, travel time resulting from dam break, area to be flooded, wave fronts, arrival time can be estimated or approximated in any way then we can go for planning to prevent all the failures and losses due to the dam break. This things can be evaluated by the mathematical or numerical models because real time field measurements are prohibitively difficult to make in case of dam break. Mathematically, the dam-break problem is commonly described by the shallow water equations (also named the Saint Venant equations for the one Dimensional case). Prominent characteristics of hyperbolic equations of this type is the formation of bores (i.e., the rapidly varying non continuous flow). It is a crucial base for substantiating the numerical method whether the scheme can capture the dam-break bore waves exactly or not. This gives rise to heightening the concern in solving such a problem.

Studies to understand the basic mechanics of DBF date back to the earliest attempt by Ritter in 1892[1]. Ritter (1892) derived an analytical solution for the hydrodynamic problem of instantaneous dam break in a frictionless and horizontal channel of rectangular shape. Later, Dressler (1952) and Whitham (1955)[2,3] included the effect of bed resistance in the analysis of DBF and derived analytical expressions for the velocity and height of the wave front. Stoker (1957)[4] extended the Ritter solution to the case of wet-bed conditions on the downriver side. He deduced analytical aspects for the surface profile in terms of the initial depths, upstream and downstream of the dam. Ritter's and Stoker's solutions assume that the reservoir is non-finite. On the other side, the analytical equations has been deduced by Hunt (1982, 1987)[5,6] by considering finite length reservoirs. In this study, the Massflow-2D code based on the MacCormack- TVD scheme with variable computational domain is proposed as a solution for the shallow water equations associated with the mountainous hazard dynamic procedure in natural terrains. The scheme has the following attributes:

1. Being able to simulate discontinuous flows such as those associated with shock propagation.
2. It is suitable to natural terrain as it isconsidering the additional acceleration and deceleration of topography.
3. Time efficiency along with 2^{nd} order accuracy in both time and space.

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X.
PP 37-41
www.iosrjournals.org

II. Research Methodology

2.1 Governing equation

The flow of water after the breaking of a dam can be described by the shallow water equation that is derived from Navier-stokes equation for an incompressible fluid by assuming that the depth of water is sufficiently small compared with some other significant length like wavelength of the water surface. Thus vertical velocity is neglected and the horizontal velocity is assumed to be constant through any vertical line between bottom and water surface. Integrating the N-S equation in vertical direction yields the shallow water equation (SWE). The general form of shallow water equation is given follow:

$$\partial U/\partial t + \partial F/\partial x + \partial G/\partial Y=S$$

Where

$$U=[h\ uh\ vh]^T \quad F=[uh\ \ hu^2+0.5gh^2\ \ uvh]^T \quad G=[vh\ \ uvh\ \ hv^2+0.5gh^2]^T \quad S=[0\ \ gh(S_{0x}-S_{fx})\ \ gh(S_{0y}-S_{fy})]^T$$

Where U is the vector of conserved variables, Fand Gare the flux vectors in the x- and y-directions and Srepresents a source term vector. The vectors U, Fand G can be expressed in terms of the primary variables u, vand h.where g is the acceleration due to gravity, h the liquid depth, u and v are the stream velocity in the xand y-directions, respectively. While S_{0x} and $S_{oy}$are the bed slopes in the two-Cartesian directions[7]. The friction slopes S_{fx} and S_{fy} have been estimated using the Manning resistance lawin which η is the Manning resistance coefficient. If the bed is taken as constant in depth and the friction was ignored, source term is considered to be zero.

$$S_{fx} = [u\eta^2 (u^2+v^2)^{0.5}]/h^{4/3} \quad S_{fy} = [v\eta^2 (u^2+v^2)^{0.5}]/h^{4/3}$$

Discretisation of Governing Equation

Before discussing the various numerical approaches for approximating the SWE, we must define the mesh and then look at the numerical boundary conditions required to implement the numerical approaches correctly. We will use a fixed mesh over the finite region $xo\leq x\leq xI$and $0\leq t\leq tN$, Here, the numerical solution is denoted by $u_i^n = u(i\Delta x, n\Delta t)$where $\Delta x = x_i - x_{i-1}$ and $\Delta t = t_n - t_{n-1}$ for all i and n.The computational domain is discretized as $x_t = i\Delta\ x$ and $t_n = n\Delta t$, where x is the size of a uniform mesh, and is the time increment. The classical finite difference scheme suitable for the discretization of Saint-Venant Equation. With source terms, is the explicit two step predictor-corrector MacCormack method, as follows

2.1.1 Predictor step:

$$[U^{n+1}_{i,j}]_p= U^n_{i,j}+\Delta t(B(U^n_{i,j})-[F(U^n_{i,j})-F(U^n_{i,j})]/ \Delta x- [G(U^n_{i,j})-G(U^n_{i,j})]/\Delta y)$$

2.1.2 Corrector step:

$$(\partial Ui,j/ \partial t)^{n+1}_{cor}=B([U^{n+1}_{i,j}]_p)-[F([U^{n+1}_{i,j}]_p)-F([U^{n+1}_{i-1,j}]_p)]/ \Delta x- [G([U^{n+1}_{i,j}]_p)-G([U^{n+1}_{i,j-1}]_p)]/\Delta y)$$

Then, we have the solution at time step n + 1 with an average between thepredictor and corrector step

$$U^{n+1}_{i,j}=\{U^n_{i,j}+[U^{\ n+1}_{i,j}]p\}/2+\Delta t/2(\partial U_{i,j}/ \partial t)^{n+1}_{cor}$$

Being second order accurate in space and time, it offers a good resolution and has a great abstract easiness, however it is well recognized that definitive second order schemes show oscillatory behavior near discontinuous and Mac Cormack’s scheme is no exception as shown in fig.1. So, this scheme can be reformulated in a way that leads to the improvement of the performances to avoid oscillation.

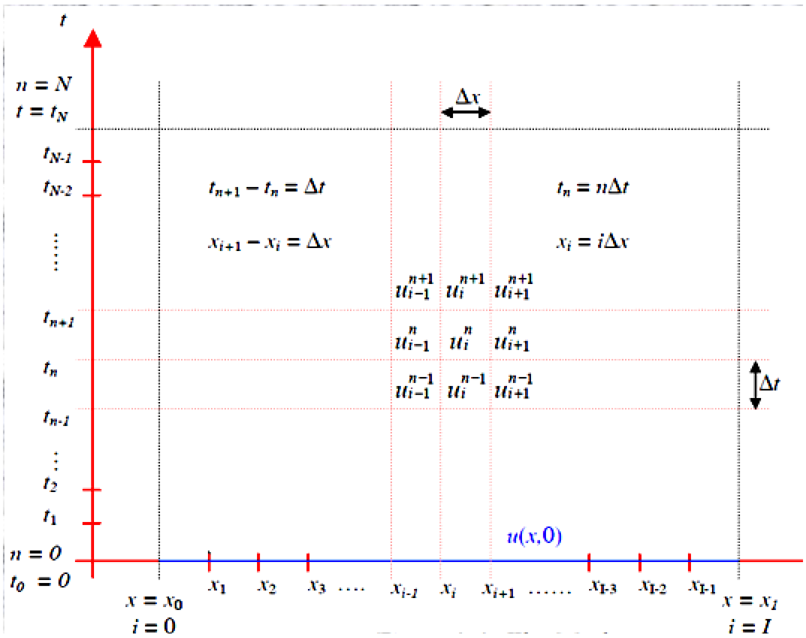


Fig.1. Time Step Discretisation

Initial and Boundary Conditions

To solve discretised equations provision of initial and boundary conditions for inflow and outflow. Prior to dam breach the upstream section acts as a reservoir.

At time t = 0, the initial conditions are given as

$$h(x, y)\big|_{t=0} = h_0(x, y)$$

$$q_x(x, y)\big|_{t=0} = q_{x0}(x, y); \quad q_y(x, y)\big|_{t=0} = q_{y0}(x, y)$$

For a 2D dam-break problem, the initial water depth h_0 is commonly the discontinuous function of coordinates (i.e., it has a water elevation difference from the upstream to the downstream). The initial discharge components q_{x0} and q_{y0} are given as zero here as there is no flow prior to the dam break. For a general shallow water problem, the boundaries of the computational domain have solid boundaries and open boundaries. In the case of solid boundaries, the governing equations do not include the turbulent viscosity, but the bottom friction, free-slip conditions may be considered, and the normal discharge to the wall is set to zero in order to represent no flux through the solid boundaries.The open boundary conditions, in particular need to be treated. The local value of the Froude number, or whether the flow is subcritical or super critical, is the basis for determining the number of boundary conditions. For the 2D subcritical flow, two external conditions are specified at the inflow boundary, and one is specified at the outflow boundary. For the supercritical flow, three boundary conditions at the inflow boundary and none at the outflow boundary have to be specified.

Stability condition

The Courant-Friedrichs-Lewy (CFL)[8] stability condition for 1-D problem is restricted by

$$\Delta t = CFL(\Delta x_{min})/(|q/h| + \sqrt{(gh)})_{max}$$

Where Courant number 0<CFL≤1. FOR 2D problem, the time step is restricted by

$$\Delta t = CFL[\min(\Delta x, \Delta y)]/\max[(|q_x/h| + \sqrt{(gh)}), | (q_y/h| + \sqrt{(gh)}]$$

Where q_x&$q_y$are the unit discharge along x and y directions. “g” is the acceleration due to gravity and “h” is the depth of water.

Wetting/drying Algorithm

In some cases the complex wetting/drying phenomenon is simulated by imposing a thin layer of water across dry cells. In this way, the computation is always carried out everywhere regardless of the wet/dry condition. However, this simple treatment is not suitable for uneven ground conditions, where the dry area must be accepted and omitted from the normal finite difference calculation. Otherwise, water level gradients over comparatively steep dry grounds will induce unreasonably large velocities[9,10].

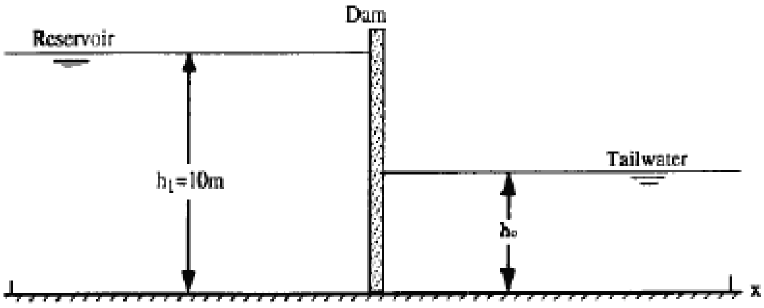
At the beginning of each step, all the grid points whose depths are smaller than a prescribed value, H_{min}, are regarded as being dry. The velocities are set to zero at dry grid points. This drying process is followed by wetting process, where the neighbouring grid points around each dry grid point are examined.

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X.
PP 37-41
www.iosrjournals.org

III. Application And Validation Of The Model

This model is validated with a hypothetical dam break problem for which analytical solution is available. The 2-D asymmetrical dam break problem has been a benchmark test case for shock-capturing numerical models. A square domain is considered, with a side length of 200m. The bed is considered frictionless with negligible slope 0.0001. A dam is located at a distance of 100m from the starting point. This dam separates the headwater, with a depth of 10m, from the tail water with the depth being denoted by $H_{0.}$ In this study H_0 can be either 5, 0.1 0r $10^{3m.}$ Wall situations are mentioned at the outer boundaries of normal with respect to the corresponding dam when first order differentials of all unknowns are fixed to zero at the outer boundaries parallel to the dam. A breach located at 95m-170m along Y direction occurs suddenly in the dam. The water surface 7.2s subsequent to the breach is examined in the following validation, using a time respond of 0.2s and a grid size of 2m.

The water surface elevations worked out by the current MacCormack scheme are provided Fig.2. below, which agree well with those calculated and estimated by other researchers.



0m 100m 200m

Fig. 2. Definition sketch of the model for validation

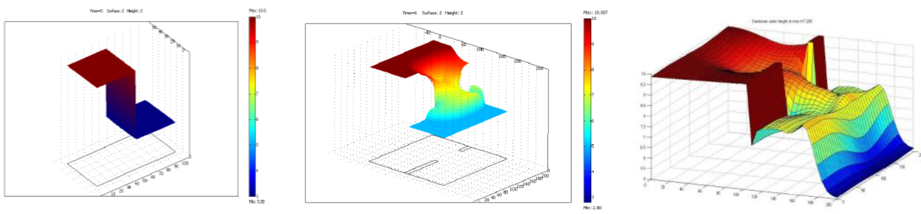


Fig.3. Water surface profile at different time steps

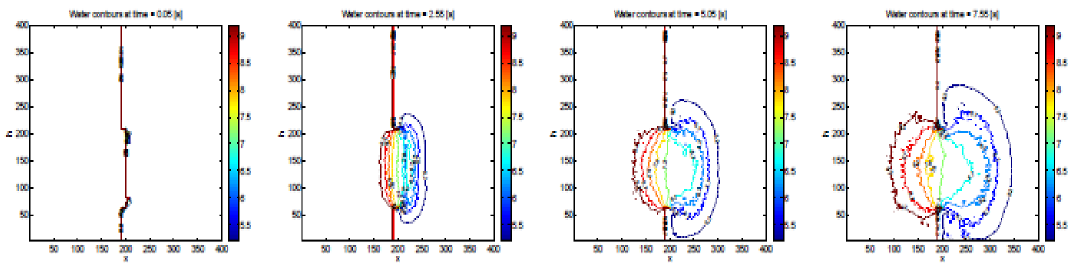


Fig.4. Water depth contours at different time interval

IV. Analysis Of Outputs And Conclusion

From the above Fig.3. and Fig.4. the outputs observed that at both ends of the breach, the water depths are minutely least than that at the center of the breach. Flow separates from the truncated dam walls just downstream of the breach and forms contour rotating eddies. Water depth profiles for wet bed and dry bed conditions are same till mid position of the upstream location and subsequently water depth profile of wet bed increases gradually towards d/s. flood wave travels faster with a decrease in the downstream water depth. The wave front is observed to be steeper when the d/s water depth is larger. Contour plots show that the wave propagation on a wet bed is faster than on a dry bed. On the other hand, the wave front is observed to be steeper when the downstream water depth is larger. In addition, the friction coefficient becomes unimportant in a wet bed case. This is due to the fact that the depressions that are normally filled by the flood wave as it passes by are now already full of water when the computation begins. In real applications, the friction coefficient is much more important, especially for the propagation times. Due to this breach a surge is formed and propagates over the flood plain. Simultaneously, a strong depression wave occurs in the reservoir and causes the water surface near the breach to descend drastically. Here the results got through application of Maccormack method shows good agreement with the previous documented results. This method can be used to further development of the numerical models of the phenomena those use shallow water equations.

Acknowledgement

My sincere thanks to Dr K.K. Khatua, professor of Civil Engineering department, NIT Rourkela for his valuable suggestions and guidance during the period of the project. No words would suffice to him for his inspiring guidance, constant encouragement, immense support and help during the course of this work. I express my heartfelt gratitude to HOD, Dept. of Civil Engineering, and Dr S.K.SAHOO NIT Rourkela (ODISHA) for permitting me to carry my task.

References

[1] C. Zoppou , S. Roberts(1999), Numerical solution of the two-dimensional unsteady dam break, Applied Mathematical Modelling 24 (2000) 457-475

[2] Hubbard, M. E., and Garcia-Navarro, 2000 ‘‘Flux difference splitting and the balancing of source terms and flux gradients.’’ J. Comput. Phys., 165, 89–125

[3] Causon, D. M., Ingram, D. M., and Mingham, C. G. 2001. ‘‘A Cartesian cut cell method for shallow water flows with moving boundaries.’’ Adv. Water Resource.24, 899–911.

[4] E.F. Toro. Shock Capturing Method for Free-Surface Shallow Flows, John Wiley & Sons, Manchester, 2001.

[5] Xing Zheng and Wen-yang Duan (2004), Numerical Simulation of Dam Breaking Using Smoothed Particle Hydrodynamics and Viscosity Behaviour, J. Marine. Sci. Appl. (2010) 9: 34-41 DOI: 10.1007/s11804-010-8037-9

[6] Spine wine, B., and Zech, Y. 2007. Small-scale laboratory dam-break waves on movable beds. Journal of Hydraulic Research, vol. 45: 73-86.

[7] Changhong Hu and Makoto Sueyoshi (2009), Numerical Simulation and Experiment on Dam Break Problem, J. Marine. Sci. Appl. (2010) 9: 109-114 DOI: 10.1007/s11804-010-9075-z

[8] C. Biscarini, S. Di Francesco and P. Manciola(2010), CFD modelling approach for dam break flow studies, Hydrol. Earth Syst. Sci., 14, 705–718, 2010 www.hydrol-earth-syst-sci.net/14/705/2010/.

[9] Sun Guangcai Wei Wenli Y.L. Liu (2010), Numerical Scheme For Simulation of 2D Flood Waves, 2010 International Conference on Computational and Information Sciences

[10] Hamid Reza Vosoughifar, Azam Dolatshah, and Seyed Kazem Sadat Shokouhi(2013), Discretization of Multidimensional Mathematical Equations of Dam Break Phenomena Using a Novel Approach of Finite Volume Method, Journal of Applied Mathematics Volume 2013, Article ID 642485, 12 pages