International Journal of Engineering, Mathematical and Physical Sciences
ISSN: 2517-9934
Vol:4, No:11, 2010

Simulation of Dam Break using Finite Volume Method

A.Roshandel, N.Hedayat, H.kiamanesh

Abstract**—**Today, numerical simulation is a powerful tool to solve various hydraulic engineering problems. The aim of this research is numerical solutions of shallow water equations using finite volume method for Simulations of dam break over wet and dry bed. In order to solve Riemann problem, Roe's approximate solver is used. To evaluate numerical model, simulation was done in 1D and 2D states. In 1D state, two dam break test over dry bed (with and without friction) were studied. The results showed that Structural failure around the dam and damage to the downstream constructions in bed without friction is more than friction bed. In 2D state, two tests for wet and dry beds were done. Generally in wet bed case, waves are propagated to canal sides but in dry bed it is not significant. Therefore, damage to the storage facilities and agricultural lands in wet bed case is more than in dry bed.

Keywords**—**dam break, dry bed, finite volume method, shallow water equations.

I. Introduction

A wide range of physical phenomena by shallow water equations are expressed. Some examples of physical phenomena are tides in the ocean, the flow in open channels, flow around hydraulic structures and dam break problem. Shallow water equations, is a set of nonlinear hyperbolic equations and forbears of viscosity in this equations. The key supposition included in the shallow water equations is that the vertical component of water particles acceleration has a little effect on the pressure, or in other words the hydrostatic pressure is considered [5].

Numerical solution of shallow water equations for propagation of flood waves with three special issues arises: the first issue is the simulation of sudden wave fronts. The second issue arising from the sudden changes in bed depth is measured (rough bed). The last issue, particularly occurs when the present scheme are exerted to survey the wave front progress over dry bed [2].

In a wide range of flow situations, it may be assumed to be always remains uniform response. The development and application of numerical methods for shallow water equations is such special form, is the basic of research work [5].

Amir Roshandel, postgraduate researcher, Islamic azad university dezful branch; e-mail: amir83s2003@yahoo.com.

Najaf Hedayat, assistant professor, Islamic azad university dezful branch.

Hasan .Kiamanesh, associate professor, Islamic azad university dezful branch.

II. Governing Equations

The mathematical model used here includes two-dimensional shallow flow equations, obtained by the incompressible flow continuity equation and the momentum navier-stokes equations. Effects of complex turbulence are not entered in the equations [1].

Shallow water equations in vector and conservation form are as follows:

$$\frac{\partial U}{\partial t} + \nabla . F = S \tag{1}$$

Where $F =$ Flux vector, including components of E & G:

$$G = \begin{pmatrix} vh \\ uvh \\ v^2 h + \frac{1}{2} g h^2 \end{pmatrix}, E = \begin{pmatrix} uh \\ u^2 h + \frac{1}{2} g h^2 \\ uvh \end{pmatrix} \tag{2}$$

$$U = \begin{pmatrix} h \\ uh \\ vh \end{pmatrix}, S = \begin{pmatrix} 0 \\ -gh(S_{0x} - S_{f_x}) \\ -gh(S_{0y} - S_{f_y}) \end{pmatrix} \tag{3}$$

h, u and v respectively: the depth of flow and the average vertical velocity components along the x and y directions.

$g =$ acceleration of gravity and t is time. S_{0x} and S_{0y}, are the bottom slopes in x and y directions, also S_{fx} and S_{fy} are the friction slopes in the same directions.

In the model, the slope of friction is calculated by the empirical strength relationships:

$$(4)\; S_{fy} = \frac{n^2 v \sqrt{u^2 + v^2}}{h^{1.33}} \;;\; S_{fx} = \frac{n^2 u \sqrt{u^2 + v^2}}{h^{1.33}}$$

Where $n =$ Manning's roughness coefficient. Depth (h) and discharges per unit width (hu & hv) are dependent variables that classified in a column vector of U.

III. FINITE VOLUME METHOD

Finite volume method is a method for demonstrating and evaluating the partial differential equations in a form of algebraic equations, and similar to the finite difference method, values are calculated in discrete locations on a geometric grid. In the finite volume method, volume integral consists of divergence term using the divergence theorem transformed to surface integral. The features that make select this approach is summarized in the following are mentioned:

1- Development of algorithms using finite volume method is able to work with complex geometries.
2- If compared with other methods, such as finite element method it was easier method.
3- This method needs less computational work in comparison with finite element method.

The numerical demonstration of physical domain is obtained using quadrilateral mesh. Each of cells is considered as a main control volume. The boundary of each cell is formed by the four direct walls dS_r surrounding it. The outward normal vector to each wall is called n_r [2].

A single control volume that is enclosed by four sides is shown in Fig. 1.

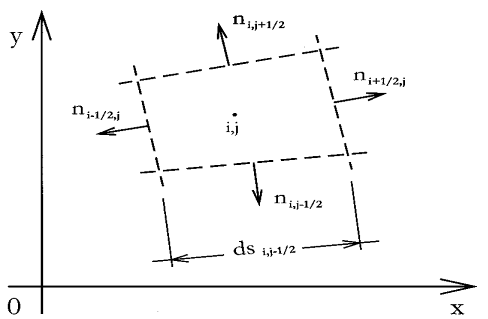

Fig. 1 Universal control volume and statements

Equation (1) can be integrated over optional control volume V:

$$\int_V \frac{\partial U}{\partial t} + \int_V \nabla . F dV = \int_V S dV \tag{5}$$

By applying the Gauss theorem for the second integral of equation (4), and also if calling ΔU the time increscent of average of U over each cell (in the time interval Δt), obtains:

$$\Delta U = -\frac{\Delta t}{\Delta V}\sum_{r=1}^{4}(F_r^{*}.n_r)dS_r + \frac{\Delta t}{\Delta V}\int_V S dV \tag{6}$$

The integral discretisation of the flux is obtained by sum over four sides of numerical flux function F^*.

IV. ROE'S APPROXIMATE SOLVER

The problems which consist discontinuities in the solution are called Riemann problems.

For accessing to exact response of discontinuities of the flow movement, a Godunov-type scheme is used. In the Godunov method, the conservative variables are considered as piecewise constant in cells for each time step and time evolution is specified by solution of Riemann problem at cells interface. In this research, the Roe's approximation Riemann solver is used. The application of this method to the two-dimensional scheme gives the following phrase for the numerical flux [5]:

$$F^{*}.n = \frac{1}{2}\left(F_L + F_R\right) - \frac{1}{2}\sum_{i=1}^{m}\overline{\alpha}_i\left|\overline{\lambda}_i\right|\overline{K}^{(i)} \tag{7}$$

Where $F_L = F(U_L)$ and U_L are the flux and the conservation variable vectors in the left side of each cell interface. Also $F_R = F(U_R)$ and U_R are the same values at the right side of each cell interface.

$$\tilde{\lambda}_1 = n_x\tilde{u} + n_y\tilde{v}, \quad \tilde{\lambda}_2 = \tilde{\lambda}_1 + \tilde{c}, \quad \tilde{\lambda}_3 = \tilde{\lambda}_1 - \tilde{c} \tag{8}$$

$$\left|\tilde{\Lambda}\right| = \begin{pmatrix} \left|\tilde{\lambda}_1\right| & 0 & 0 \\ 0 & \left|\tilde{\lambda}_2\right| & 0 \\ 0 & 0 & \left|\tilde{\lambda}_3\right| \end{pmatrix}, \quad \tilde{A} = \frac{\partial F}{\partial U} = R\left|\tilde{\Lambda}\right|R^{-1} \tag{9}$$

$$R = \begin{bmatrix} 0 & 1 & 1 \\ -n_y & \tilde{u} + \tilde{c}n_x & \tilde{u} - \tilde{c}n_x \\ n_x & \tilde{v} + \tilde{c}n_y & \tilde{v} - \tilde{c}n_y \end{bmatrix}$$
$$R^{-1} = \frac{1}{2\tilde{c}}\begin{bmatrix} 2\tilde{c}(\tilde{u}n_y - \tilde{v}n_x) & -2\tilde{c}n_y & 2\tilde{c}n_y \\ \tilde{c} - \tilde{u}n_x - \tilde{v}n_y & n_x & n_y \\ \tilde{c} + \tilde{u}n_x + \tilde{v}n_y & -n_x & -n_y \end{bmatrix} \tag{10}$$

$\tilde{A}$ is the jacobian matrix of F vector at the vertical direction in boundary. n_x and n_y are components of external unit vector in x and y directions.

Average values of $\tilde{h}, \tilde{u}, \tilde{v}$ and $\tilde{c}$ are defined as follow:

$$\tilde{u} = \frac{\sqrt{gh_L}\times u_L + \sqrt{gh_R}\times u_R}{\sqrt{gh_L} + \sqrt{gh_R}}; \tilde{v} = \frac{\sqrt{gh_L}\times v_L + \sqrt{gh_R}\times v_R}{\sqrt{gh_L} + \sqrt{gh_R}} \tag{11}$$

$$\tilde{h} = \sqrt{h_L . h_R}, \quad \tilde{c} = \sqrt{\frac{1}{2}g(h_L + h_R)} \tag{12}$$

V. STABILITY AND BOUNDARY CONDITIONS

In order to be sure of numerical stability of presented scheme, the time step (Δt) should be limited using the stability condition of Courant number that is expressed by following phrase:

$$CFL = \frac{\Delta t}{\Delta x}\left|\left(\sqrt{gh} + \sqrt{u^2 + v^2}\right)\right|_{\max} \leq 1 \qquad (13)$$

That Δx is the distance between centers of two neighbor cells.

In this research the used boundary conditions are divided into two different kinds, which called them open boundary and closed boundary. For closed boundary condition, the normal velocity is set to zero. For the open boundary condition, the Riemann invariables is used for velocity and depth [3].

VI. RESULTS & DISCUSSIONS

A. 1D dam break over dry bed without friction bed

A channel with 2000 m length is considered. The dam is situated at 1000 m downstream channel. The initial upstream and downstream water levels are 6 m and 0 m respectively and the simulation time is 40 seconds. The results are shown in Fig. 2, which demonstrates a satisfactory agreement with analytical solutions. In this state, wave advance is too more. Therefore possibility of structural failure around the dam is very high. Moreover, after dam breaking great volume of water is released into the downstream which causes a flood for downstream lands and eventually economic losses.

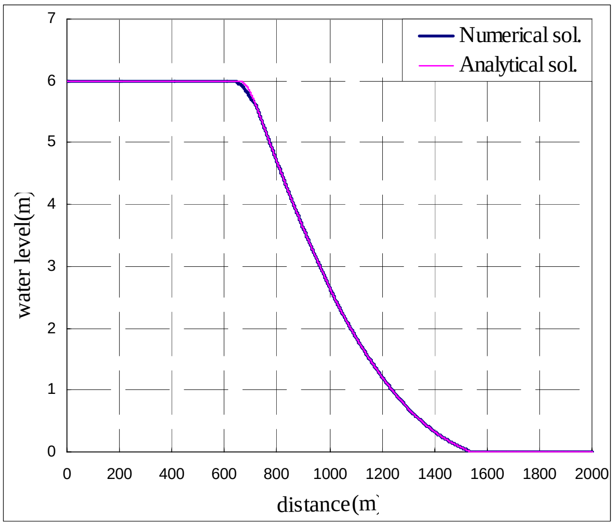

Fig. 2 Comparison of analytical and numerical solutions

B. 1D dam break over dry bed with friction

This test is similar to previous test, but in this state the bottom is flat, with a Manning roughness coefficient equal to $0.03\,m^{-\frac{1}{3}}.s$. The initial upstream and downstream water levels are 6 m and 0 m respectively and the simulation time is 40 seconds. Comparison of exact solutions with simulated depths is shown in Fig. 3. The results are in satisfactory Correspondence with analytical solutions. As shown in Fig. 3, the wave progress in this test is seldom in comparison with what is resulted in the case without friction. Instead wave height at the end of progress point is more comparing to the previous test. Therefore, this state is less hazardous for agricultural lands and structures.

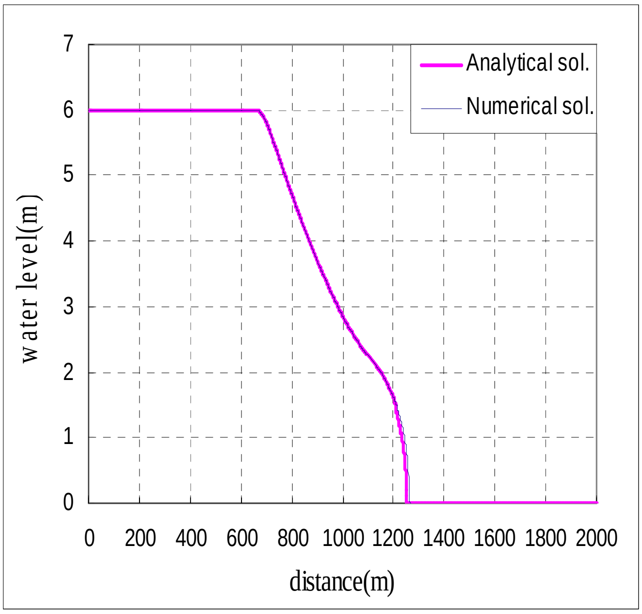

Fig. 3 Comparison of analytical and numerical solutions

C. 2D partial dam break over wet bed

The computational domain consists of 200 m × 200 m zone and the dam is located at 100 m. the breach is 75m wide and the initial water depths at upstream and downstream are 10 m and 5 m respectively, Also the duration of the simulation is 40 seconds. The water surface profile which is in satisfactory agreement with other numerical results such as [1], [2] and [7] is shown in Fig. 4. From the Fig. 4, a shock wave progress to downstream, because in this case the flow is subcritical everywhere, also the waves from dam break are propagated to canal sides that cause severe damages to the downstream structures and storage facilities around the dam and agricultural lands.

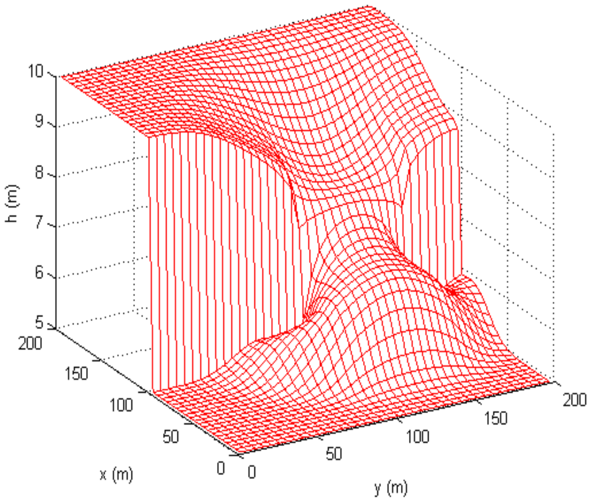

Fig. 4 water surface profile over wet bed

D.2D partial dam break over dry bed

This test case is almost similar to the previous test, except that the tailwater is dry. Channel dimensions are 200 m × 200 m flat region, and the nonsymmetrical breach is 75 m wide. The initial water level at upstream and downstream is 10 m and 5 m respectively, and the period of simulation is 6 seconds. Fig. 5 shows the water surface profile for dam break at $t = 6$ seconds over dry bed. The result is in well correspondence with other numerical results such as [2], [4] and [6]. Achieved In this mood, no shock front propagated to downstream but advantage of the waves is more compared to the previous test. As well as wave propagation into canal sides is negligible than in the case wet bed, because in this state the flow is supercritical. Therefore, damage to the hydraulic structures, utilities and agronomy lands is too less than in previous test.

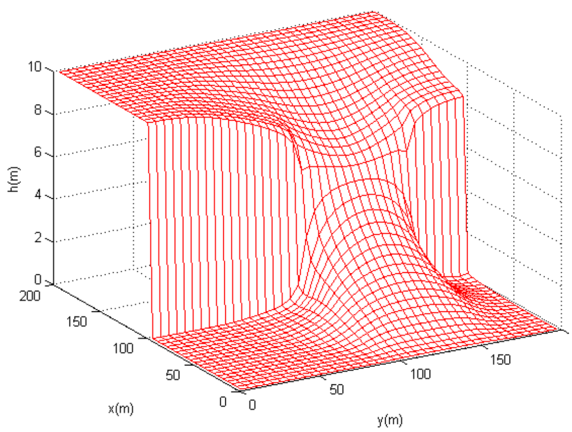

Fig. 5 water surface profile over dry bed

References

[1] M. H. Chaudry, “Open-Channel flow”, Prentice-Hall, Englewood Cliffs, NJ,USA, 2008.

[2] A. Valiani, V. Caleffi, and A. Zanni, “Case Study: Malpasset Dam-Break Simulation Using a 2D Finite Volume Method,” JHE, 128(5), 2002, pp 460–472.

[3] D. H. Zhao, and H. W. Shen, “ Approximate Riemann solvers in FVM for 2D hydraulic shock wave modelling,” JHE, Vol. 122(12), 1996, pp 692-702.

[4] P. A. Sleigh and P. H. Gaskell, M. Berzins, and N. G. Wright, “An Unstructured Finite-volume Algorithm for Predicting Flow in Rivers and Estuaries,” Computers & Fluids, 27(4), 1998, pp 479–508.

[5] E. Toro, “Riemann Problems and theWAF Method for Solving the Two-dimensional Shallow Water Equations,” Philosophical Trans. Royal Soc. London, U.K., A338, 1992.

[6] J. S. Wang, H.G. NI and Y. S. He, “Finite-difference TVD Scheme for Computation of Dam-break Problems,” JHE, 126(4), 2000, pp 253–262.

[7] M. Kianian, M. A. Banihashemi, 2D model for dam break using fread & McCormack method, 4th international congress on civil engineering, Tehran university, 2008.