Flood and Shock Wave Simulation by Using Finite Volume Method on Unstructured Meshes in Domains with Complex Topography

Burak Turan and K.-H. Wang
Department of Civil and Environmental Engineering, University of Houston
Houston, TX 77204-4003
Phone: 713-743-4277, E-mail: khwang@uh.edu

Abstract

A 2D flow solver using the finite-volume method on unstructured triangular meshes was developed to model shock waves and flood flows in domains with irregular topography. The solver used the second-order Godunov type schemes in space and fourth-order Runge-Kutta scheme in time domain. The wet and dry algorithm was integrated in the model to handle the exchange of wet and dry elements in transient simulations. The solver was tested with various dam break problems and by performing flood routing simulation in a complex river network system. The present model results showed fairly good agreement with other published experimental and numerical solutions.

1 Introduction

Two-dimensional (2D) shallow water equations (SWE) used to model water waves traveling on rivers and floodplains can be solved numerically by using robust finite volume schemes. The scheme's capability to model irregular boundaries and complex topographies attracted many researchers working in the area of hydraulics. While modeling the flow traveling into dry river beds or floodplains, the cells changing from dry to wet or vise versa may cause numerical errors in the velocity calculations. To prevent these numerical problems, a wet/dry tracking and treating algorithm or a so called check program are included in the solvers. The bed slope gradient of the cells becomes more important while solving SWE on a computational domain with irregular topography as it is challenging to preserve the balance between flux and source terms.

In this study, up to second order finite volume schemes are developed and coded with C++ object oriented language to solve 2D SWE in conservative form. Higher order correction terms can be included in the bed slope term to have a second order scheme in space for the non-flat domains. The unstructured triangular meshes are used as spatial domain in the solver. Friction slope terms are written in terms of Manning's roughness coefficients in SWE to account for the effect of the bed friction. To get rid of the spurious oscillations in the solution, slope limiters are used for the second order scheme. Time integration is handled by using the fourth-order Runge Cutta Scheme. The stability of the Godunov scheme is controlled by satisfying a suitable Courant- Friedrichs-Levy (CFL) condition. In this paper, the verification of the solver is achieved by testing various dam break cases and flood event occurred on Ulus Basin with irregular topography. The present model results are compared with published experimental and numerical solutions given by other researchers (Anastasiou and Chan, 1997; Brufau and Garcia-Navarro, 2000; Usul and Turan, 2006)

Test Cases

Selected test cases are presented to demonstrate the performance of the developed 2-D flow solver. A dam break problem in a 200m by 200m domain (Fig. 1) is simulated. The water depth of the dam is 10m and tail water depth is 5m. The water is released downstream from a 75m wide opening (Fig. 1). The wave propagation in 3D and the water depth contours at 7.2 seconds is given in Fig. 2. The similar plots and the solutions can be seen in the literature (Anastasiou and Chan, 1997). A dam break problem in a 45^0 channel bend (Fig. 3) is simulated with inputs of n=0.0095 for the channel bottom and 0.0195 for the channel walls. There is no bottom slope in this case. The water depth of the dam is 0.25m and tail water depth is 0.01m. The water is released from a 2.44m by 2.39m reservoir to a 45^0 bend downstream channel from 0.495m opening (Fig. 3). The water depths calculated at point P5 are compared with the experimental results (Brufau and Garcia-Navarro, 2000) (Fig. 4). The match between the present numerical solutions and experimental results is good. The wave propagation in 3D view at 3 secs is given in Fig. 5. The results agree with the plots and the results given by Brufau and Garcia-Navarro (2000).

Watershed Flood Modeling

The flood routing analysis is carried out to evaluate the performance of the present solver in modeling flood flows. The domain of Ulus basin (Fig. 6) is divided into 3694 triangular cells. The min and max Δt calculated in the solver simulations are 0.30 and 2.2 secs, respectively. The solver is run for the whole flood duration which is 144 hours from 07/04/91 to 07/10/91. The flood depths and the extent of the flood can be predicted. Simulated peak discharge is 585 m^3/s whereas the observed one was 596 m^3/s. The comparison between simulated and observed flood hydrographs at the outlet of the basin is presented in Fig. 7. It can be noticed that the predicted flood hydrograph agree reasonably well with observed data.

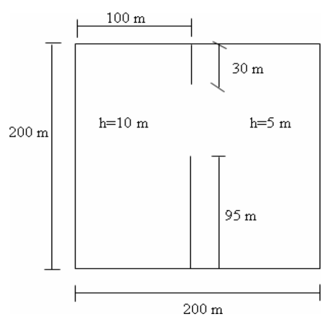

Fig. 1: Dam break problem in a 200m x 200 m domain.

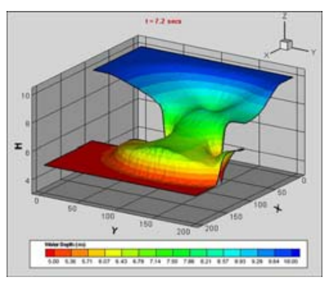

Fig. 2: Wave propagation in 3D view at 7.2 secs.

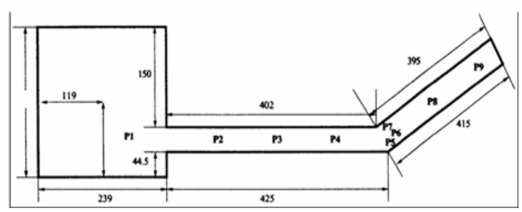

Fig. 3: Dam break problem in a 45^0 channel bend.

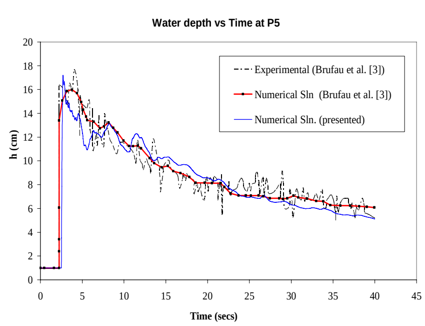

Fig. 4: Comparison of results at location P5.

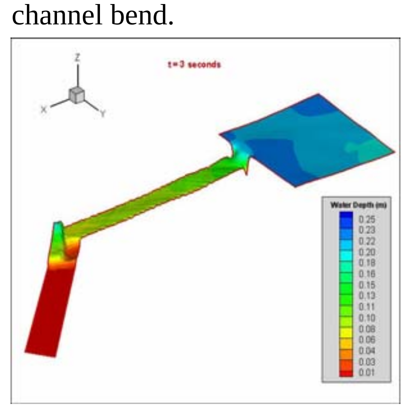

Fig. 5: 3D view of wave propagation in the channel at t =3secs.

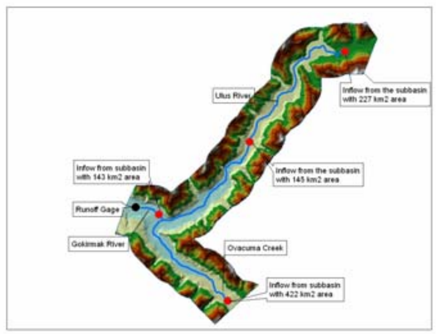

Fig. 6: River network at Ulus Basin and the locations of inflow hydrographs.

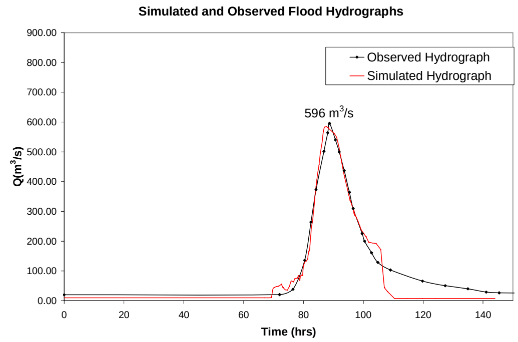

Fig. 7: Comparison of simulated and observed flood hydrographs.