Discussion of

Two-dimensional depth-averaged finite volume model for unsteady turbulent flows

By Chunshui Yu & Jennifer Duan, Journal of Hydraulic Research, 50(6), 599-611

Discussers

ERPICUM Sébastien (IAHR Member), University of Liege, HECE research unit, Chemin des chevreuils 1 B52/3, 4000 Liège, Belgium
Email:s.erpicum@ulg.ac.be (Corresponding author)

PIROTTON Michel (IAHR Member), University of Liege, HECE research unit, Chemin des chevreuils 1 B52/3, 4000 Liège, Belgium

ARCHAMBEAU Pierre, University of Liege, HECE research unit, Chemin des chevreuils 1 B52/3, 4000 Liège, Belgium

DEWALS Benjamin (IAHR Member), University of Liege, HECE research unit, Chemin des chevreuils 1 B52/3, 4000 Liège, Belgium

Discussion of

Two-dimensional depth-averaged finite volume model for unsteady turbulent flows

By Chunshui Yu & Jennifer Duan, Journal of Hydraulic Research, 50(6), 599-611

The Discussers read the Research Paper with interest. The development of a state of the art two-dimensional (2D) depth-averaged flow model coupled with a k-ε turbulence model such as presented by the Authors is challenging. The attention paid by the Authors to the problems of wet-dry fronts and bed slope source term is particularly relevant.

The Research Paper is focused on the importance of turbulence modelling to properly predict measured data from 2D dam-break flow experimental tests. In particular, the conclusions are based on the analysis of the location of a moving hydraulic jump in the experiment of a dam-break flow against an isolated obstacle conducted in the scope of the IMPACT project (Soares-Frazão and Zech, 2007). Simulations were performed with grid sizes of 10, 6.7 and 5 cm. Best results are found using a grid size of 5 cm. Without the turbulence model, the numerical results are significantly deteriorated regarding the hydraulic jump propagation at gauge 2. The Authors state that the simulation errors are grid-independent for grids smaller or equal to 5 cm.

The Discussers applied a finite volume 2D flow model based on Cartesian multiblock grids (Erpicum and al., 2010), similar to the model proposed by the Authors, to the same test case proposed by Soares-Frazão and Zech (2007). Although a k-ε turbulence model is also implemented in the Discussers model (Erpicum et al. 2009), it was not used in this study. Indeed, validation of the flow model for dam break flow propagation suggested that turbulence modelling is not necessary to properly reproduce most of the wave propagation characteristics (Erpicum and al., 2010). A specific attention has instead been paid to the grid size influence. Grid sizes of 10, 5, 2 and 1 cm have been used. In the latest case, to decrease the number of cells and thus the computation time, the reservoir and the downstream part of the channel were modelled with a grid size of 5 cm, using the solver multiblock capacities. The 1 cm grid size was thus used to discretize the area between the dam location and the obstacle, as well as around the obstacle.

Boundaries, geometry, bottom roughness and initial conditions are the same as those depicted by the Authors. The discretization scheme is first order accurate and the time integration scheme is second order accurate (Runge-Kutta 22 with a CFL number equal to 0.1).

Figure 1 compares the simulated water depths, with the four different grid-sizes, and the corresponding measurements at six gauges.

Grid size has little influence on the computed results, except at gauge 2. Using a turbulence model is not necessary to get numerical results in good agreement with the measured value at gauges 1 and 3 to 6. This is consistent with the results from the Authors.

At gauge 2, without turbulence modelling, the best agreement with measured data is found with the coarsest grid (10 cm). Other grid sizes do not succeed in predicting the wave front propagation. For the two finest grids, the hydraulic jump does not reach the gauge during the simulation time.

As shown by these results, the grid size has an important influence on the propagation of moving hydraulic jumps. Gauge 2, located where the dam-break wave reflects against the obstacle and results in an oblique hydraulic jumps, is the only gauge in the experiment where such a moving shock is observed.

Figure 2 shows the simulated water depths computed 20 cm downstream of gauge 2 along the channel centreline, i.e. two cells downstream of gauge 2 for a grid size of 10 cm. This location is closer to the obstacle than gauge 2. Observed water depths at gauge 2 are also represented. These results show that grid convergence is not totally reached for the smallest grid sizes considered. However, they highlight again the strong influence of the grid size on the simulation of the moving hydraulic jumps.

As a consequence, grid size convergence is a key point to properly validate a 2D numerical model for the propagation of moving shocks. Therefore, the conclusion of the Authors as regards with the need for a turbulence closure in dam-break flow modelling may be questioned since the Authors fail to demonstrate by appropriate data the grid-convergence of their simulations.

References

Erpicum, S., Meile, T., Dewals, B., Pirotton, M. & Schleiss, A. (2009). 2D numerical flow modeling in a macro-rough channel. *Int. J. Numer. Methods Fluids,* 61(11)**,** 1227-1246.

Erpicum, S., Dewals, B., Archambeau, P., & Pirotton, M. (2010). Dam-break flow computation based on an efficient flux-vector splitting. *Journal of Computational and Applied Mathematics*, 234(7), 2143-2151.

Soares-Frazão, S., & Zech, Y. (2007). Experimental study of dam-break flow against an isolated obstacle. *Journal of Hydraulic Research*, 45(suppl1), 27-36.

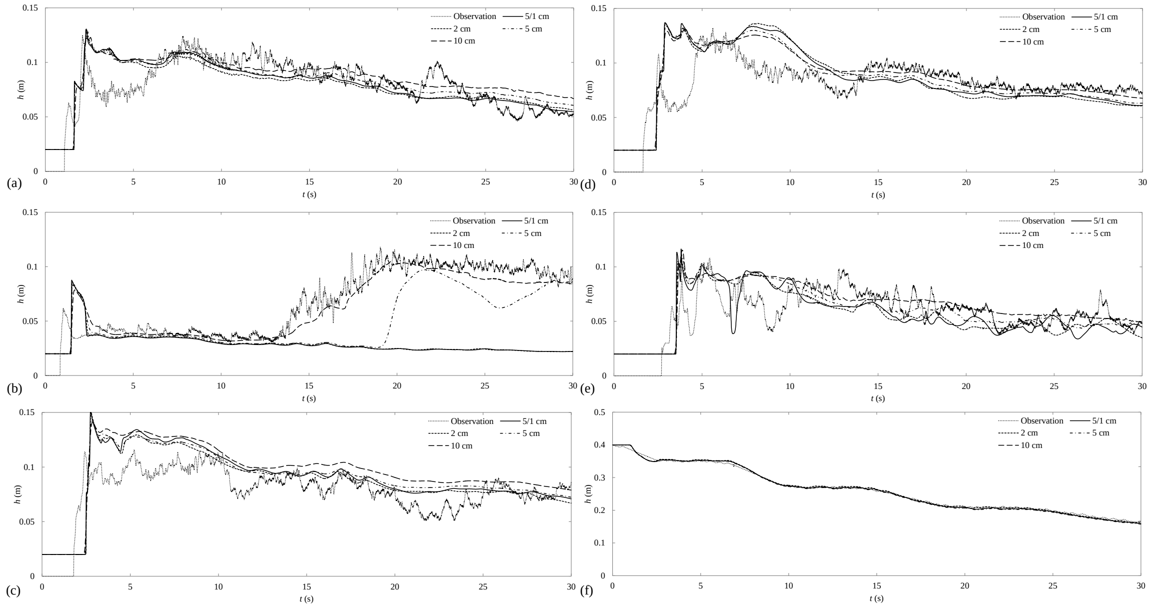

Figure 1: Comparison of measured and computed flow depths with different grid sizes at (a) gauge 1, (b) gauge 2, (c) gauge 3, (d) gauge 4, (e) gauge 5 and (f) gauge 6.

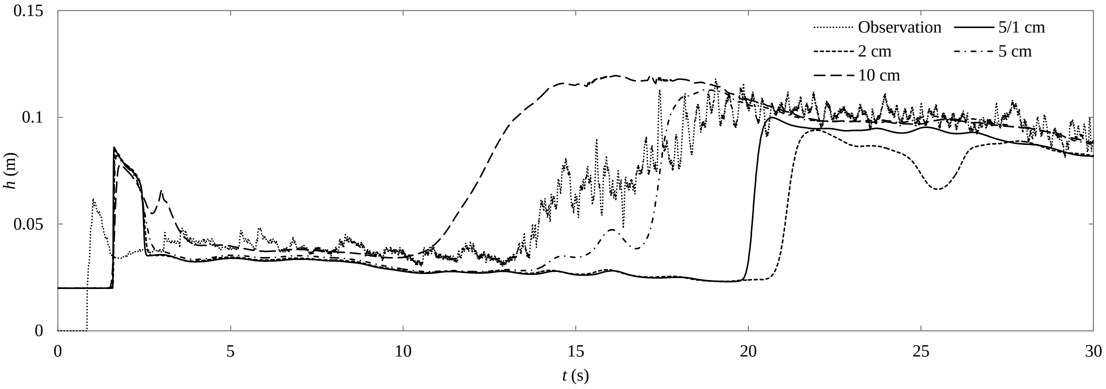

Figure 2: Comparison of flow depths computed with different grid sizes 20 cm downstream of gauge 2 along the channel axis and flow depths observed at gauge 2.