CFD modelling approach for Dam break flow studies

Biscarini C., Di Francesco S. and Manciola P.

Rebuttal to J. M. Hervouet (Referee)

The authors cordially thank the referee for his positive and constructive comments and for the comprehensive and detailed review of the paper. In the following, we respond to each reviewer's note individually.

R: There is no friction in the two test-cases. It is however the most important factor in dam-break computations. A real life case with friction, like the Malpasset test-case used in the CADAM concerted action would be an interesting extra test-case (this is an idea for further articles...).

A: We thank the referee for the valuable suggestion. By the way, we are already working in implementing friction in our numerical models in order to simulate real test cases and these would be the subject of future articles.

R: The 2 test-cases are comparisons to small scale physical models. In this case we have small cells and the k-epsilon model is probably applicable (though almost all validations are done in internal flows at very small scale). This would probably not be the case in real life applications, where much larger cells are used and the numerical velocity gradients loose their physical meaning. In such cases only the turbulence model for diffusion on the vertical is relevant. Generally speaking the efficiency of VOF methods on highly distorted meshes (horizontal versus vertical) is also a point to be tested.

A: The next step of our research, after testing different numerical models, is to test them on real-life applications. In that case the effects of different turbulence models as well as the behavior of the VOF method coupled to the turbulence model itself will be further studied. The possibility of using LES instead of RANS is under consideration.

R: the fact that non hydrostatic equations will give higher celerity of waves is questionable and may not be always the case. A recent PhD thesis by Edmond E. Tossou (edjrosse-edmond.tossou.1@ulaval.ca) at the university of Laval in Quebec compares Serre equations and SW equations. It is shown that non hydrostatic terms slow down the flood wave and this is presented as a well known phenomenon. In my own experiments on the Malpasset test-case, I see no difference between 2D and 3D. It is then an open question with probably not a single answer

A: We thank the referee for arising this point, that is now included in the discussion of the revised version of the paper.

R: at the top of page 6766, and then in page 6770, there is a little ambiguity on the k-epsilon model used : it is said that the standard k-epsilon model is used, and that the depth-integrated k-epsilon model is implemented and included in the software, but nowhere it is really said that the depth-integrated version has been duly used in 2D. As the depth-integrated version accounts for dispersion which may sometimes be more important than turbulence, it indeed should be used.

A: The depth-integrated k-epsilon model is used for the SW while standard k-epsilon has been used for the 3D NS-VOF. This point has been clarified in the text

R: the time-steps, 0.02 and 0.01 s seem to be very small. Are explicit schemes used, maybe giving the corresponding CFL numbers based on celerity or velocity would be more informative.

A: The small time step is due to the fact that the surface-tracking algorithm is considerably more sensitive to the Courant number than in standard fluid flow calculations. The indication of the time step has been given for sake of clarity. As the simulations have been performed at a fixed time step, the Courant number varied in the whole simulation but never exceeded 0.3. To follow the suggestion of the referee, the maximum CFL numbers are now reported in the revised version of the paper.

R: page 6766 the word "specie" is probably a misprint, is it species or special ?

A: It is “species”. Thanks for the correction.

R: anglo-saxons would rather say "consists of" than "consists in".

A: Changed.