

A 2D FLOOD INUNDATION MODEL BASED ON CELLULAR AUTOMATA APPROACH

F. Dottori^{*}, E. Todini^{*}

^{*} Università di Bologna, Dipartimento di Scienze Della Terra e Geologico-Ambientali
Via Zamboni 67, 40126 Bologna, Italy
e-mail: francesco.dottori@unibo.it

Key words: flood model, cellular automata

Summary. The European Directive 2007/60 introduces important modifications on flood risk evaluation and management, and calls for an extensive application of 2D models. Responding to this need of accurate and fast methods for the simulation of flood events, in the past years different hydraulic models based on reduced complexity approaches have been realized and successfully applied. The present work describes a reduced complexity model based on the cellular automata approach and the diffusive wave equations. The model was tested in different numerical cases, providing good results in terms of stability, computational speed and accuracy of solution.

1 INTRODUCTION

The Flood Directive 2007/60 of European Union⁽¹⁾ introduces important modifications on flood risk evaluation and management. The new directive prescribes the definition of flood hazard maps and flood risk maps, along with the development of flood risk management plans. According to the directive, “assessments, maps and plans should be based on appropriate ‘best practice’ and ‘best available technologies’ not entailing excessive costs in the field of flood risk management”. Therefore, to comply the directive, methods for modeling flood inundation should be reliable and capable of generating the required hydraulic information in an appropriate level of detail, but also practicable in terms of computational time and costs as well as input data requirement, since application over large inundation areas will be inevitably needed.

In order to meet all these requirements, the use of 2D hydraulic models which describe flow processes using various degrees of approximation, either in flow equations or in computation scheme, seems recommendable. The reduced complexity allows these models to be faster and less demanding in terms of computational burden and data requirement with respect to more complex models, moreover different works showed that the use of simplified models does not necessarily implies a loss of accuracy and reliability in results^{(2) (3)}.

2 THE PROPOSED MODEL

2.1 The cellular automata approach

To date, several computational schemes for 2D hydraulic modeling have been proposed in

literature. Flood inundation models are generally based on the finite differences, finite elements or finite volumes methods. More recently the Runge-Kutta Discontinuous Galerkin method has been successfully applied to 2D/3D problems in the case of hypercritical, transcritical and convection dominated diffusive problems ⁽⁴⁾. In the present paper a model based on a version of the finite volumes method, called the cellular automata approach, is proposed for representing diffusion dominated phenomena such as flood plain inundation events.

A cellular automaton (CA) may be defined as a complex system made of a finite number of elements, called cells, which are characterized by a number of properties (i.e. a colour, a shape, ecc). A cellular automaton evolves in time by changing the status of all the cells simultaneously; at every time step each cell change its state depending on the state of a set of cells within a certain distance, called vicinity index, according to a particular state function⁽⁵⁾.

The cellular automata approach is used to describe physical phenomena regulated only by local laws, however it has been successfully applied also to fluidodynamics phenomena like debris flows. When applied to reproduce two-dimensional flow of water, the CA approach originates a computational scheme equivalent to the finite volumes method: the study area is schematized using a regular grid of polygonal elements (normally squares), while the state functions describing the system evolution are the momentum and continuity equations, written in a full or simplified version.

Since the overall state of the system only depends on the previous time step state, the CA approach implies the use on an explicit solution scheme. Moreover every cell is influenced only by a limited number of neighborhood cells, therefore flow equations may be solved separately for each cell. It is clear that such computation scheme is inherently very fast and, moreover, it is particularly suited for parallelization, which allows a further increase of computational speed.

Two well known hydraulic models based on CA-like approach are LISFLOOD⁽⁶⁾ and FLO2D⁽⁷⁾. So far, this group of models proved to be very effective in simulating flood events, even when compared with models based on more complex computational schemes ⁽²⁾ ⁽⁸⁾; in particular, their code structure and computational speed allows an extensive use of high resolution, remote sensing data, like airborne laser altimetry (or LiDAR), and airborne and satellite Synthetic Aperture Radar (SAR) data ⁽⁹⁾.

2.2 The proposed CA model

The hydraulic model herein proposed is based on the shallow water equations written in diffusive form. In the literature, the ability of diffusive approximations to correctly simulate flooding phenomena is still under discussion; however, to date models based on diffusive equations have been tested successfully against measurements from actual flood events, and they often performed as well as models based on complete equations ⁽²⁾ ⁽³⁾. As such, it is hypothesized that uncertainties over the data set (especially topography and roughness), dominate and thus influence model results to a greater extent than errors and approximations due to simplified mathematical description ⁽³⁾ ⁽¹⁰⁾. Lastly, models based on diffusive equations provided good results when tested against both analytical and results from physical or alternative numerical solutions ⁽⁶⁾ ⁽⁹⁾ ⁽¹⁰⁾. In the proposed model the study area is schematized

by a grid of polygonal elements, which can be of both regular and variable shape; considering two adjacent cells i and j , the discharge passing through the contact face is computed as:

$$Q_{i,j} = \frac{bh_m^{5/3}}{n} \left(\frac{H_i - H_j}{\Delta x} \right)^{1/2} \quad (1)$$

where H_i, H_j are the water stages in the two cells, Δx is the distance between cells centroids, b is the width of the contact face, n is the Manning roughness coefficient; h_m is the arithmetic mean between water heights in the two cells.

The model uses the Euler explicit computation scheme: first, discharges between cells are computed according to equation (1); by summing incoming and outgoing discharges from i -th node to the connected m nodes the total discharge Q is obtained:

$$Q(V_t) = \frac{\Delta V_i}{\Delta t} = \sum_{j=1}^m Q_{i,j} \quad (2)$$

then volumes V and water heights in cells are updated according to this expression:

$$\Delta V_{i+\Delta t} = \Delta t \cdot Q(V_t) + O(Q^2) \quad (3)$$

where $O(Q^2)$ is the truncation error deriving from the scheme.

Every discharge calculation step includes a control on volumes, which avoids that the volume of water flowing out of a cell is greater than the sum of the volume stored in the cell itself and the incoming volumes from adjacent cells.

2.3 Why a new model?

The proposed CA model, although similar to LISFLOOD and FLO2D under some aspects, differs for a number of details and for the code structure which offers significant advantages. First, the model can work indifferently both with regular and irregular meshes, thus taking advantage from both approaches. For example, since the computation scheme is applied both for 1D and 2D flow, a drainage network may be simulated as a part of the study area, without the need of specific input data, while the main channels may be reproduced using a cross section scheme, like 1D hydraulic models. Irregular meshes can be of great advantage in simulating urban areas, allowing for a better reproduction of buildings and road network.

Also, wetting and drying effects are reproduced with a very simple algorithm, based on the control of volumes (see par. 2.2). As a matter of fact, the majority of reduced complexity models use some kind of approximated formulae⁽⁶⁾, since the primary objective is to maintain model stability. It should be noted that in real world the combined effects of surface storage and roughness, infiltration and evapotranspiration should be considered to properly deal with wetting and drying effects; moreover it has been shown that the shallow water equations lose validity when water depths are very reduced, since surface roughness and storage plays a major role in this situation⁽¹¹⁾. Consequently, it was felt that a simplified algorithm could be the best solution to deal with drying effects without complicating the model structure. More

detailed solution for this issue will be considered in future research work.

3 NUMERICAL CASES

After some preliminary tests, the CA model has been applied to a number of numerical cases, comparing the results to both theoretical solutions and numerical results from other models. Two of these cases are herein described. The aim is to investigate the influence of some major approximations of CA models in reproducing flood events: for example, the model structure does not consider the influence of upstream or downstream conditions, since every point in the flow is directly influenced only by adjacent elements. Moreover, in the CA approach the components of momentum equation are decoupled, thus introducing a further approximation when dealing with 2D flows.

It must be noted that the evaluation of simplified models in numerical cases has been rarely applied in literature⁽⁶⁾; indeed, the majority of research works describe model applications in real world cases, but very often this does not permit to carry out an in-depth analysis of applied models. For example, in a real flood event the calibration procedure plays a major role and may compensate for model limitations if the quantity and quality of available data are limited. On the contrary, numerical cases are based on simplified and well controlled conditions, reducing the number of variables to be considered and the uncertainty on their influence on model performance.

3.1 Case 1: channel

The first presented case deals with 1D flow over a regular channel with a mild slope. The geometry, resumed in Table 1, has been chosen to test the ability of CA model to simulate a diffusive wave, since significant lamination effect is expected.

Bed slope	10^{-4}	Section shape	rectangular
Channel width	250 m	Manning Roughness	$0.05 \text{ m}^{-1/3} \text{ s}$
Channel length	50 km		

Table 1 : Geometry of case 1

The hydrogram starts from an initial condition of uniform flow along all the channel, with a discharge of $10 \text{ m}^3 \text{ s}^{-1}$; at $t=0$, the discharge is linearly increased from upstream, going from 10 to $100 \text{ m}^3 \text{ s}^{-1}$ in 5 hours; the discharge values remains constant for the next 30 hours and then decrease linearly in 5 hours from 100 to $10 \text{ m}^3 \text{ s}^{-1}$. The geometry and the flow conditions were chosen in order to have a very large channel, so that the form of momentum equation used in CA model (eq. 1) may be considered a good approximation. The downstream boundary condition is the uniform flow. The produced result were compared with the well known model HEC-RAS⁽¹²⁾.

Three computation grids have been considered: one with 400x1 cells of 125x250m size, a second with 200x1 cells of 250x250m and another with 100x1 cells of 500x250m. This means that each grid maintains a channel width of 250m, while the longitudinal grid resolution varies. For each grid, simulations with different time steps have been performed (see Table 2).

Figure 1 shows the results in terms of flow profile, compared with HEC-RAS results.

$\Delta X - \Delta t$	1s	2s	5s	10s	15s	30s	60s	NC ts
125m	Y	Y	N					$\approx 1.2s$
250m	Y	Y	Y	N				$\approx 5s$
500m	Y	Y	Y	Y	Y	Y	N	$\approx 22s$

Table 2 : results for case 1. “N” indicates that the simulation produces significant oscillations on solution, while “Y” indicates a stable solution. *NC ts* indicates the minimum time step computed by the Neumann condition (eq.4, see par.3.3).

As expected, the reduction of cell size decrease the model stability, and vice versa; however, the accuracy of solution is not compromised by spatial discretization until water level oscillation becomes significant (see Table 2). The computational time is very small: the simulation with the 250x250m grid and a time step of 5s over a period of 72 hours took approximately 4s on a computer with a 1.86 Ghz processor and 2Gb RAM.

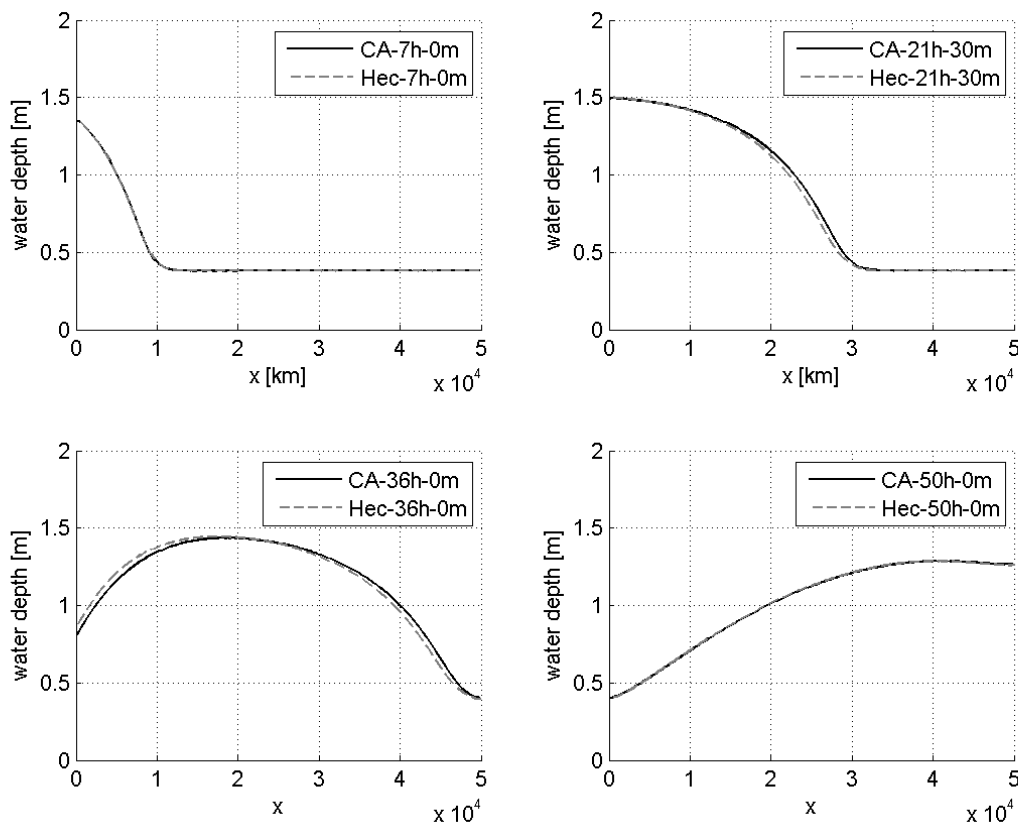


Figure 1: case 1: comparison between flow profiles computed by CA and HEC-RAS models.

As shown in Figure 1, the profiles computed by the two models are very regular and

almost coincident, both in general shape and position of wave front at the different time intervals. The wave simulated by the CA model is slightly faster, however the RMSE between CA and HEC-RAS results is always small (the maximum value, computed for time step $t = 35h$, is well below 4cm). The downstream discharges are also very similar. Most important, the CA model reproduces with a very good accuracy the wave attenuation, even if the computation scheme is approximated with respect of the complete scheme based on full De Saint Venant Equations used by HEC-RAS.

3.2 Case 2: horizontal plane

The wave routing on a flat slope is a case in which hydraulic models may be more subjected to instability, particularly when flow velocity and water surface slope are reduced. Considering explicit diffusive models like CA models, other two instability factors are the use of high resolution grids (cell size below 10m) and the presence of deep water stages⁽⁸⁾.

In order to test the proposed CA model in these conditions, a series of simulations on an horizontal plane were performed using two computation grids: one of 20x20 cells with a 10x10m size and a second of 10x10 cells with a 20x20m size; a initial water stage of 3m is assigned over the whole plane, without subsequent incomings of water, and the area is drained by a weir located in one corner. Such initial conditions has been chosen because tests with incoming discharge have shown that the magnitude of oscillation seem to be only a function of water stage and cell size.

The graphics in Figure 2 show the water stages computed by CA model, when using a computational time step of 0.1s. The water surface shows significant oscillations, however, their amplitude does not vary if stage remains constant, that is, the solution does not “explode”. Also, their magnitude decreases as the stage decreases. As expected, the instability increases when increasing the time step or decreasing the cell size, and vice versa (see Table 3). The simulation with the 10x10m grid and a time step of 0.01s over a period of 5 hours took approximately 19min on a computer with a 1.86 Ghz processor and 2Gb RAM.

$\Delta X - \Delta t$	0.01s	0.02s	0.05s	0.1s	0.2s	NC ts
10m	Y	N				$\approx 5 \cdot 10^{-4} s$
20m	Y	Y	Y	N	N	$\approx 2 \cdot 10^{-3} s$

Table 3 : results for case 2. “N” indicates that the simulation produces significant oscillations on solution, while “Y” indicates a stable solution. *NC ts* indicates the minimum time step computed by the Neumann condition (eq.4, see par.3.3).

3.3 Discussion

Results of case 1 and 2 have shown that the stability of CA model obviously depends on the spatial and time resolution, like any other explicit model; therefore the establishment of a stability condition according to flow conditions is needed.

First, we considered the stability conditions used in similar models. In particular the LISFLOOD model, based as well on the diffusive equations, considers the Neumann stability condition, in the following expression⁽⁶⁾:

$$\Delta t = \frac{\Delta x^2}{4} \min \left(\frac{2n_{i,j}}{h_{m-i,j}^{5/3}} \left(\frac{\partial H_{i,j}}{\partial x} \right)^{1/2} \right) \quad (4)$$

that is, for every cell i , the term between parenthesis is computed in all the j directions, and the minimum time step found is chosen to go on with the simulation.

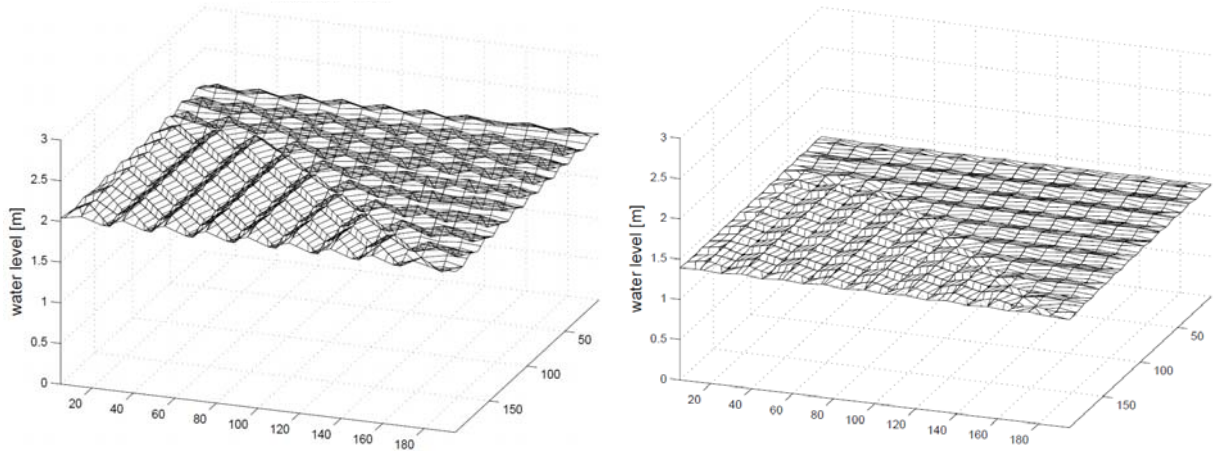


Figure 2. Case 2: water stages computed by CA model after 30 mins (left) and 1 hour (right) from simulation start . The outlet is located in the lower right corner.

In a similar way, the FLO2D model, based on the full shallow water equations, uses the Courant-Levy-Friedrich (CLF) condition⁽⁷⁾:

$$\Delta t = C\Delta x / (v + c) \quad (5)$$

where C is a coefficient which depends on the adopted explicit algorithm and c is the celerity.

Both the conditions have been embedded in CA model code and the computed time steps have been compared with the experimental values found in the two presented numerical cases. (see tables 2 and 3). As expected, the CLF condition (eq. 5) is not suitable for CA model, since the diffusive approximation needs more a stringent condition; however, it was found that the model maintains the stability with greater time steps than those computed by the Neumann stability conditions (eq. 4). The difference is very reduced for case 1 (table 2), but in case 2 the experimental time step may be more than 10 times greater than the stability condition value; this is of primary importance since the computational time needed by the model is considerably reduced with respect to other similar models. So far, it has not been possible to establish a stability criterion suitable for all the tests performed with the model: further research on this issue will be needed.

4 CONCLUSIONS

The present work describes a model based on the cellular automata approach and the diffusive approximation of flow equations. The model was tested successfully in two

numerical cases, proving to be very accurate when compared with numerical solutions. Moreover, the code structure seems to be significantly more stable, and therefore faster, compared to similar models based on diffusive equations. Further research is needed to establish a stability condition for the realized model, which could be used for implementing a variable time step.

Next work will focus on presenting the results of the proposed CA model in real world applications, and on realizing a parallelized version of the model.

ACKNOWLEDGEMENTS

The work described in this paper in part of a research program supported by the Civil Protection Agency of Emilia Romagna.

REFERENCES

- [1] “Directive 2007/60/EC on the assessment and management of flood risks”, *Official Journal of the European Union* (2007).
- [2] M.S. Horritt and P.D. Bates, “Evaluation of 1-D and 2-D numerical models for predicting river flood inundation”. *J. Hydrol.* 268 (1–4), 87–99 (2002).
- [3] D. Yu, S.N. Lane, “Urban fluvial flood modelling using a twodimensional diffusion-wave treatment. Part 1: mesh resolution effects”. *Hydrological Processes* 20 (7), 1541–1565 (2006).
- [4] B. Cockburn, C.W. Shu, “Runge-Kutta Discontinuous Galerkin methods for convection-dominated problems”, *J. Sci. Comput.* 16 (3) 173–261 (2001).
- [5] J. L. Schiff, *Cellular Automata: A Discrete View of the World*, Wiley & Sons Inc, 256 pp.
- [6] N.M. Hunter, M.S. Horritt, P.D. Bates, M.D. Wilson, M.G.F. Werner, “An adaptive time step solution for raster-based storage cell modelling of floodplain inundation”, *Advances in Water Resources* 28 (9), 975–991 (2005).
- [7] FLO-2D Software Inc., *FLO-2D Users Manual* (2007).
- [8] N.M. Hunter, P.D. Bates, S. Neelz, G. Pender, I. Villanueva, N.G. Wright, D. Liang, R.A. Falconer, B. Lin, S. Waller, A.J. Crossley, D.C. Mason, “Benchmarking 2D hydraulic models for urban flooding”, *Water Management* 161, 13–30 (2008).
- [9] N.M. Hunter, P.D. Bates, M.S. Horritt, M.D. Wilson, “Simple spatially-distributed models for predicting flood inundation: A review”. *Geomorphology* 90, 208–225 (2007).
- [10] T. Xanthopoulos, C. Koutitas, “Numerical simulation of a two-dimensional flood propagation due to dam-failure”, *J. Hydraul. Res.*, 14(4), 321-331 (1976).
- [11] G. Tayfur, M. Levent Kavvas, R.S. Govindaraju, “Applicability of St. Venant Equations for two-dimensional overland flows over rough infiltrating surfaces”. *Journal of Hydraulic Engineering*, 1, 51-63 (1993)
- [12] Hydraulic Engineering Centre (HEC), *HEC RAS Hydraulic Reference Manual*, U.S. Army Corps of Engineers, Davis, California, USA, 377 pp. (2001).