XIX International Conference on Water Resources
CMWR 2012
University of Illinois at Urbana-Champaign
June 17-22,2012

NUMERICAL SIMULATION OF GROUNDWATER-SURFACE INTERACTIONS BY EXTERNAL COUPLING OF THE 3D RICHARDS EQUATION AND THE FULL 2D SHALLOW-WATER EQUATIONS

Daniel Caviedes-Voullième*, Javier Murillo* and Pilar García-Navarro*

*LIFTEC-University of Zaragoza
María de Luna 3, CP 50018 SPAIN
e-mail: daniel.caviedes@unizar.es, web page: http://ghc.unizar.es/

Key words: Richards equation, coupled surface-subsurface model

Summary. A method that solves both groundwater and surface domains by means of modeling physical processes is presented. Numerically, the submodels are an explicit 2D finite volume shallow-water equations solver and an implicit 3D finite volume Richards equation solver. To link the submodels a non-iterative external coupling algorithm is used. The integrated model handles naturally dry or flooded conditions at the surface, as well as saturated or unsaturated conditions in the soil.

The algorithm is built upon considering the solution of the surface water field as the top boundary pressure-head condition of the subsurface domain. Once the boundary condition is set, the solution of the subsurface domain is obtained, and the resulting flows at the surface are computed and transferred as source terms for the next time step of the surface model. Because of the explicit nature of the surface water solver, the time step is constrained by the CFL condition. On the other hand, the implicit nature of the subsurface solver allows for larger time steps. Hence, time steps may be different for each model, which requires careful treatment of coupling in time.

The algorithm is evaluated with a series of test cases to assess its applicability, accuracy and sensitivity.

1 INTRODUCTION

Some environmental, ecohydraulics, irrigation and engineering applications have a special interest in the accurate simulation of the interactions between surface and subsurface flow systems. The numerical simulation of 2D surface flows has been efficiently and accurately done using 2D shallow-water equations[1]. Groundwater flows have been often simulated by solving the 2D Dupuit equation or, when necessary and possible, by solving the Richards equation in 1D, 2D or even 3D. In the present work, unsteady surface and subsurface flows are solved conjunctively. In earlier worksy, the coupling has been done by empirical models, such as those for infiltration. Nowadays, several coupling strategies have been proposed[2]: external coupling, iterative coupling and full internal coupling. Apart from the coupling, the surface flow has usually been modelled using diffusive-wave approximations to the shallow water equations[3], or the kinematic wave approximation[4], or even using the Richards equation in both domains[5]. However, to the authors knowledge, no model has used these strategies for conjunctive solutions of the full 2D shallow-water and 3D Richards equations.

2 SURFACE FLOW MODEL

The surface water model is based on the 2D shallow-water equations. In this section the 2D formulation of the shallow-water equations is presented, expressing volume conservation and momentum conservation in vector form

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} + \frac{\partial \mathbf{G}(\mathbf{U})}{\partial y} = \mathbf{S} + \mathbf{H} + \mathbf{I} \tag{1}$$

where $\mathbf{U} = (h, q_x, q_y)^T$ are the conserved variables with h representing depth (L) and $q_x = hu$ and $q_y = hv$ the unit discharges (L^2/T), with u and v (L/T) the depth averaged components of the velocity vector $\mathbf{u}$ along the x and y coordinates respectively. The fluxes are given by

$$\mathbf{F} = \left(q_x, \frac{q_x^2}{h} + \frac{1}{2}gh^2, \frac{q_x q_y}{h}\right)^T \quad \mathbf{G} = \left(q_y, \frac{q_x q_y}{h}, \frac{q_y^2}{h} + \frac{1}{2}gh^2\right)^T \tag{2}$$

where g (L/T^2) is the acceleration of the gravity. The source terms of the system are split in two terms. The term $\mathbf{S}$ is defined as

$$\mathbf{S} = (0, -ghS_{fx}, -ghS_{fy})^T \tag{3}$$

wher the friction losses are written in terms of the Manning's roughness coefficient n $(TL^{-1/3})$:

$$S_{fx} = n^2 u\sqrt{u^2+v^2}/h^{4/3} \quad S_{fy} = n^2 v\sqrt{u^2+v^2}/h^{4/3} \tag{4}$$

The term $\mathbf{H}$ defined as

$$\mathbf{H} = \left(0,\ -gh\frac{\partial z_b}{\partial x},\ -gh\frac{\partial z_b}{\partial y}\right)^T \tag{5}$$

expresses the pressure force variation along the bottom in the x and y direction, formulated in terms of the bed slopes of the bottom level z_b (L).
Source term $\mathbf{I}$ is

$$\mathbf{I} = \Big(i, 0, 0\Big)^T \tag{6}$$

where i is the infiltration (negative) or exfiltration (positive) rate at the bottom surface. System (1) is time dependent, non linear, and contains source terms. Under the hypothesis of dominant advection it can be classified and numerically dealt with as belonging to the family of hyperbolic systems. The mathematical properties of (1) include the existence of a Jacobian matrix, $\mathbf{J_n}$, of the flux normal to a direction given by the unit vector $\mathbf{n}$, $\mathbf{E} \cdot \mathbf{n} = \mathbf{F}n_x + \mathbf{G}n_y$ defined as[1]

$$\mathbf{J_n} = \frac{\partial\left(\mathbf{E} \cdot \mathbf{n}\right)}{\partial \mathbf{U}} = \frac{\partial \mathbf{F}}{\partial \mathbf{U}} n_x + \frac{\partial \mathbf{G}}{\partial \mathbf{U}} n_y \tag{7}$$

The presented equations are solved by an explicit, first order, upwind, finite volume scheme[1]. Because of the explicit nature of the numerical scheme, the time step must satisfy the Courant-Friedrichs-Lewy condition to guarantee numerical stability.

3 GROUNDWATER FLOW MODEL

The mixed form of Richards equation in 3D is

$$\frac{\partial \theta(h_p)}{\partial t} = -\nabla \mathbf{J} = \nabla \left[\boldsymbol{K}(h) \nabla \Big(h_p + z \Big) \right] \tag{8}$$

where θ (L^3/L^3) is the volumetric water content in the soil, z (L) is the vertical coordinate in reference to a certain datum, h_p (L) is water column pressure at a point with elevation z and $\boldsymbol{K}$ (L/T) is the hydraulic conductivity tensor. The model described herein has been implemented for 3D variably saturated flow[6]. The spatial discretization is based on finite volumes. In terms of time discretization, it is an implicit scheme linearized by means of Picard iterations[7], which allow large time steps while keeping numerical stability. Note that equation (8) implicitly carries the definition of flux from Darcy's Law.

4 ALGORITHM DESCRIPTION

The coupling of the subsurface model with the surface flow model is done by two mechanisms: boundary conditions and source terms. The subsurface model receives information from the surface model through boundary surfaces, which are in fact the 2D cells of the surface mesh and depend on the surface flow state. On the other hand, the bed surface is not a boundary in the surface flow model which implies that the interaction with the subsurface model cannot be through boundary conditions. Instead, it is done by means of source/sink terms. Whenever water infilitrates into the ground, a sink term arises in the shallow water mass equation, and whenever water exfiltrates from the ground a source term appears. In summary, pressure conditions (water depth) computed from the surface model act as boundary conditions for the subsurface model, and the flows computed at the surface boundary act as source/sink terms in the surface model.

Time coupling is done by an externar non-iterative procedure with non-equal time steps (different time steps for the surface and subsurface submodels). The solution is obtained for each model separately and is not iterated to reconsider the conditions which have evolved during the time step. Furthermore, the explicit nature of the surface model requires to comply with the CFL condition, resulting in small time steps. Meanwhile, the implicit solution of the subsurface model allows for large time steps. Although this implies that the surface flow time step could be used for subsurface computations, doing so might result in very large computational time. This is due to the fact that a subsurface time step, in general, is more costly than a surface time step. In consequence, the time steps are most likely to be different.

The time sequence of the coupling algorithm is as follows. Initial conditions are given for both models, and a subsurface time step Δt_G is specified, equal or greater that the surface flow time step Δt_S. Global simulation time is denoted t, and current simulation time for surface flow is denoted t_s and for subsurface flow t_g. First, n_s surface flow time steps are computed until $t_s \gtrsim t_g + \Delta t_G$. The actual subsurface time step which will be used now is $\Delta t_g = t_s - t_g$. At this point the surface solution at time t_s is used as boundary condition for the subsurface model, imposed at time $t_g + \Delta t_g$. The subsurface time step is solved and the current subsurface time is then $t_g = t_s + \Delta t_g$. Finally, the flow exchange computed from the subsurface domain is set in the surface flow model as source/sink terms in time t_s and the procedure is performed again.

The coupling method is summarized as follows,

$$h_{p,i}^{n+1} = \begin{cases} h_j^{n+1} + \delta z_{ij}, \text{ if } h_j^{n+1} > h_\epsilon \\ \text{equation (8), if } h_j^{n+1} \leq h_\epsilon \end{cases} \tag{9}$$

$$i_j^n = \frac{V_{bs,i}^n}{A_j \Delta t_g} \tag{10}$$

where $h_{p,i}$ is the water column pressure in the subsurface boundary cell i, h_j is the water depth in surface cell j which lies above subsurface boundary cell i. Furthermore, $\delta z_{ij} = z_{b,j} - z_i$ is the height between the subsurface cell center elevation and the surface cell center elevation, and h_ϵ is a numerical threshold to discrimante wet/dry surface cells. Note that a positive value of V_{bs} implies exfiltration.

The volume exchanged through the boundary surface V_{bs} is cannot be defined with either equation (1) or (8). Instead, it must be computed from enforcing mass conservation in the subsurface boundary cell. It is therefore related to the volumes exchanged through other faces of such boundary cell. It is necessary to obtain such volumes V_ω, by integrating Darcy's flux in time

$$V_\omega = \int_0^{\Delta t_g} \mathbf{J}_\omega(h(t)) \cdot \mathbf{n}_\omega A_\omega \, dt \tag{11}$$

where A_ω is the area of the cell face ω where the flow is computed. The integral in equation (11) must be done numerically because of the high non-linearity of $\mathbf{J}(h(t))$.

Once all the non-boundary exchange volumes V_ω are known, it is possible to enforce mass conservation conditions upon the boundary cell and compute the boundary surface exchange volume with

$$V_{bs} = \left(\theta_i^{n+1} - \theta_i^n\right) V_i - \sum_{\substack{\omega=1 \\ \omega \neq bs}}^{N_\omega} V_\omega \tag{12}$$

In most cases $\theta_i^{n+1} - \theta_i^n = 0$, except for the initial moments of infiltration or exceptional drainage conditions in which there is still interaction while the surface boundary cell has become unsaturated. Furthermore, there are some exceptions to be considered, such as if there is enough water available at the surface to infiltrate the computed exchange volume during the time step, but these are not discussed here for brevity.

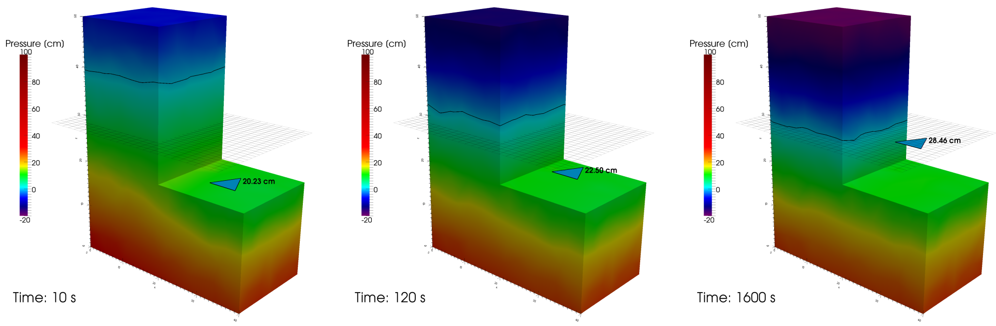
There are additional issues to be addressed, such as the coupling errors which are introduced by the changing surface water depth during the groundwater time step (when they are different), as well as the effects on the exchange volumes generated by mesh resolution (in particular, near the soil surface) and submeshing (several surface cells which lie over a single face of the subsurface 3D cell) and mesh resolution.

5 TEST CASES

5.1 Artesian well

This case was designed to verify that pressures in both domains are correctly coupled. It was also intended to track the global mass conservation during the transient process. The setup resembles the conditions of an artesian well. The surface domain consists of only one cell with impervious boundaries, which is connected to the subsurface domain. All other boundaries of the subsurface domain are impervious. The surface is initially dry, while the subsurface is fully saturated with a hydrostatic pressure profile.

The results of the simulation were excellent. By the end of the process, the phreatic surface was coplanar with the free water surface, as can be seen in figure 1c. Global mass conservation was also excellent, with an error of 0.00125%.



(a) $t = 10\,s$ (b) $t = 120\,s$ (c) $t = 1600\,s$

Figure 1: Results for Artesian Well test case

5.2 Dam Break over pervious bed

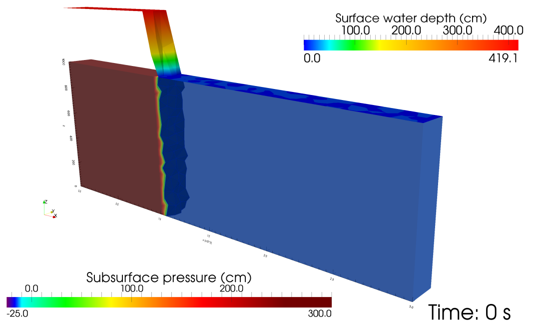
The setup for this case is a $30 \times 2 \times 10\,m$ soil block saturated from $x = 0$ to $x = 10$. The surface condition is a dam break setup, with water depth $4\,m$ in $0\,m \leq x \leq 10\,m$ and initially dry in $10\,m < x < 30\,m$. All boundary conditions for both domains are impervious. The subsurface time step is $10\,s$, significantly larger than the surface time steps (less than $1\,s$).

The results are very satisfactory and mass conservation error is in the order of 0.0025%. Some results are shown in figure 2. The surface shown inside the subsurface domain is the phreatic surface.

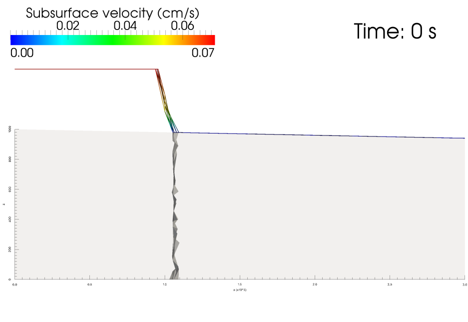
The simulation was performed again with a subsurface time step of $0.25\,s$ to observe the dam break wave in detail. Some results are shown in figure 3. This time scale allows to observe particular dynamics in the subsurface flow, specially near the advancing surface wave. In figure 4 a close up of such region is shown. Note the complex subsurface dynamics and in particular, the vertical circulation near the surface front. Mass conservation error in this case is in the order of 0.00003%.

6 CONCLUSIONS

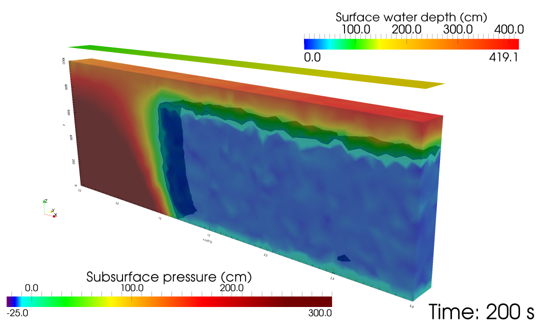
The proposed coupled model makes use of the full 2D shallow-water equations for surface flow and the 3D Richards equation for subsurface flow. The surface submodel is solved with an explicit, finite volume scheme requiring small time steps, while the subsurface submodel is discretized implicitly, allowing for larger time steps. Following these numerical properties the coupling approach is external and non-iterative, meaning that the submodels are executed in sequence, exchanging information only once per subsurface time step. The coupling approach is based on conservation laws only, without the need of empirical infiltration equations. The coupled model was tested with two academic test cases which yield satisfactory results, showing that the coupling approach is mass conservative and allows to simulate complex interaction dynamics near the soil surface, even during high velocity phenomena such as dam break events.



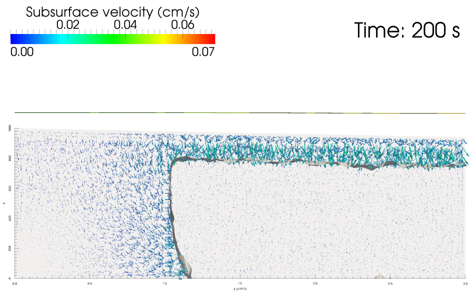
(a) Pressure $t = 0\,s$



(b) Velocity $t = 0\,s$

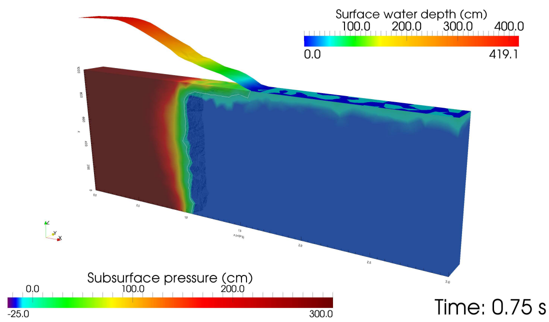


(c) Pressure $t = 200\,s$

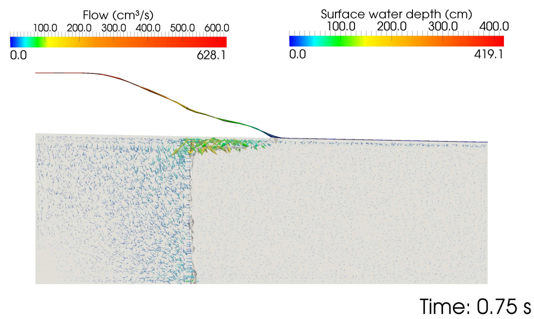


(d) Velocity $t = 200\,s$

Figure 2: Results for Dam Break test case with $\Delta t_G = 10\,s$



(a) Pressure $t = 0.75\,s$



(b) Velocity and Flow discharge $t = 0.75\,s$

Figure 3: Results for Dam Break test case with $\Delta t_G = 0.25\,s$

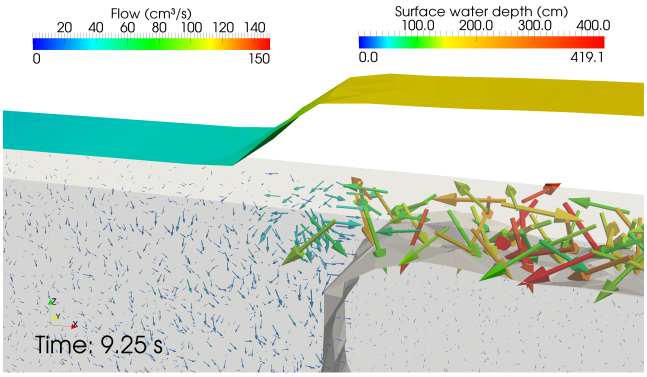


Figure 4: Velocity vectors below the wave

References

[1] J. Murillo and P. Garcia-Navarro, "Weak solutions for partial differential equations with source terms: Application to the shallow water equations," *Journal of Computational Physics*, vol. 229, pp. 4327–4368, JUN 1 2010.

[2] A. Furman, "Modeling coupled surface-subsurface flow processes: A review," *Vadose Zone Journal*, vol. 7, pp. 741–756, MAY 2008.

[3] S. Panday and P. Huyakorn, "A fully coupled physically-based spatially-distributed model for evaluating surface/subsurface flow," *Advances in Water Resources*, vol. 27, pp. 361–382, APR 2004. Fall Meeting of the American-Geophysical-Union held in honour of George F Pinder, San Francisco, CA, DEC 10-14, 2001.

[4] M. Sulis, S. B. Meyerhoff, C. Paniconi, R. M. Maxwell, M. Putti, and S. J. Kollet, "A comparison of two physics-based numerical models for simulating surface water-groundwater interactions," *Advances in Water Resources*, vol. 33, pp. 456–467, APR 2010.

[5] S. Weill, E. Mouche, and J. Patin, "A generalized Richards equation for surface/subsurface flow modelling," *Journal of Hydrology*, vol. 366, pp. 9–20, MAR 15 2009.

[6] D. Caviedes-Voullième, J. Murillo, and P. García-Navarro, "Numerical simulation of three-dimensional transient variably saturated flow," in *Numerical Methods for Hyperbolic Equations: Theory and Applications* (Vázquez-Cendón, M.A. and Cea, L. Bermúdez, A., ed.). NHMETA. An international conference to honour Professor E.F. Toro. Santiago the Compostela, Spain. 2011.

[7] M. Celia, E. Bouloutas, and R. Zarba, "A General Mass-Conservative Numerical-Solution for the Unsaturated Flow Equation," *Water Resources Research*, vol. 26, pp. 1483–1496, JUL 1990.