

Contents

1	Introduction	2
2	Background: Equations and Numerical Methods	2
2.1	The 2D Saint Venant Equations	2
2.2	SVE numerical methods	3
3	SVE-R model Fortran code	7
3.1	SVE-R model structure	7
3.2	Initializing the SVE solver	7
3.3	The predictor step	10
3.4	Corrector step	12
3.5	The <code>source</code> subroutine	13
3.6	<code>timestep</code> The Richards equation subroutine	15
4	Running the model	16
4.1	Output files	16
5	The Simulated Domain	17
6	Python wrapper scripts	18
6.1	<code>sim_input.py</code> writes Fortran input files	18
6.2	Parameters specified in <code>params.json</code>	19
6.3	<code>runmodel</code> : compiles and execute <code>dry.f</code>	20
6.4	<code>read_sim</code> : reads the Fortran output files:	20
7	Examples	21
7.1	<code>example_2by2</code>	21
7.2	<code>example_inflow</code>	21
8	Other iPython notebook files	21
8.1	SVE-R model	21
8.2	Giraldez and Woolhiser	22
8.3	Roughness	22
8.4	Richards stand-alone code	22
8.5	<code>params.dat</code>	24
8.6	<code>boundary.dat</code>	24
8.7	<code>coords.dat</code>	25
8.8	<code>vanG.dat</code>	25
8.9	Model overview/summary	26
8.10	Source pseudocode	28
8.11	<code>timestep</code> pseudocode	30

1 Introduction

This model couples two pre-existing models: a 2D SVE solver [Bradford and Katopodes, 1999] and a 1D Richards equation solver [Celia et al., 1990]. The SVE solver uses a finite volume method involving two steps, predictor and corrector, to achieve second order accuracy. The SVE model is coupled at each grid cell and timestep to the Richards equation solver. The core of the model is implemented in Fortran (`dryR.fort`), and Python scripts are provided to write and read the Fortran files.

This document is divided into the following sections:

- Section 2: A brief summary of the Saint Venant and Richards Equations.
- Section 3: An overview the SVE-R numerical methods, and how the two model components are coupled.
- Section 3: An overview of the Python wrapper scripts, which write the Fortran input files, compile and execute the Fortran code, and read and visualize the Fortran outputs.
- Additional Python code: a stand-alone Richards solver
- Additional Python code: an implementation of ?.

2 Background: Equations and Numerical Methods

2.1 The 2D Saint Venant Equations

The SVE are written in integral form:

$$\frac{\partial}{\partial t} \int_{\Omega} \mathbf{U} d\Omega + \oint_{\partial} (\mathbf{F} dy - \mathbf{G} dx) = \int_{\Omega} \mathbf{Q} d\Omega \quad (1)$$

where $\mathbf{U}^T = (h, hU, hV)$ is the vector of conservative variables.

$$\mathbf{F} = \begin{bmatrix} hU \\ hU^2 + \frac{1}{2}gh^2 \\ hUV \end{bmatrix}; \quad \mathbf{G} = \begin{bmatrix} hV \\ hUV \\ hV^2 + \frac{1}{2}gh^2 \end{bmatrix}$$

where h is the flow depth, z is the bed elevation, u and v are the vertically averaged velocities in the x and y directions, respectively. The source terms are defined as:

$$\mathbf{Q} = \begin{bmatrix} p - i \\ -gh \frac{\partial z}{\partial x} - gh S_{f,x} + \frac{u(p-i)}{2} \\ -gh \frac{\partial z}{\partial y} - gh S_{f,y} + \frac{v(p-i)}{2} \end{bmatrix}$$

where p is the rainfall rate, i is the infiltration rate of water into the bed, and $S_{f,x}$ and $S_{f,y}$ are the x and y components of the friction slope.

2.2 SVE numerical methods

The model is adapted from *Bradford and Katopodes (1999)*, referred to here as BK, and uses predictor-corrector time-stepping to provide a second-order accurate solution. The only significant difference between the BK model and the model described here is the coupling to a Richards equation solver. The following summary of the numerical methods is adapted from *Bradford and Katopodes (1999)*, where a better and more complete explanation can be found.

A predictor is computed at time level $n + 1/2$ by solving the primitive equations in generalized coordinates, and a corrector is computed at the $n + 1$ time level by solving the integral equations.

Predictor Step

The predictor solution is computed by solving the equations in primitive form. In generalized coordinates:

$$\frac{\partial \mathbf{W}}{\partial t} + \mathbf{A}_W \frac{\partial \mathbf{W}}{\partial \xi} + \mathbf{B}_W \frac{\partial \mathbf{W}}{\partial \eta} = \mathbf{Q}_W \quad (2)$$

where ξ and η are in the directions of increasing j and k indices, respectively. The j, k indices indicate the column and row numbers of a given cell, respectively (see Figure 1 schematic). $\mathbf{W}_T = [h, U, V]$ is the array of primitive variables, and the matrices \mathbf{A}_W and \mathbf{B}_W are defined as:

$$\mathbf{A}_w = \begin{bmatrix} U_\xi & h\xi_x & h\xi_y \\ g\xi_x & U_\xi & 0 \\ g\xi_y & 0 & U_\xi \end{bmatrix} \quad \mathbf{B}_w = \begin{bmatrix} U_\eta & h\eta_x & h\eta_y \\ g\eta_x & U_\eta & 0 \\ g\eta_y & 0 & U_\eta \end{bmatrix}$$

where $U_\xi = U\xi_x + V\xi_y$ and $U_\eta = U\eta_x + V\eta_y$. ξ_x , ξ_y , η_x and η_y are the grid transformation metrics for mapping x and y to ξ and η . In Cartesian coordinates, the generalized coordinates simplify to: $\xi = x$ and $\eta = y$.

The predictor solution in cell j, k at $t + \Delta t/2$ is given as:

$$\mathbf{W}_{j,k}^{n+1/2} = \mathbf{W}_{j,k}^n - \frac{\Delta t}{2} (\mathbf{A}_W \overline{\Delta \mathbf{W}}_\xi + \mathbf{B}_W \overline{\Delta \mathbf{W}}_\eta - \mathbf{Q}_W)_{j,k}^{n+1/2} \quad (3)$$

where the overbar denotes a cell-average gradient of \mathbf{W} in cell j, k , which is computed with a flux limiter (nonlinear average) in order to preserve solution monotonicity. Flux limiters become first-order accurate near discontinuities while remaining second-order accurate elsewhere, and several options are included in the code, described in Section 3.

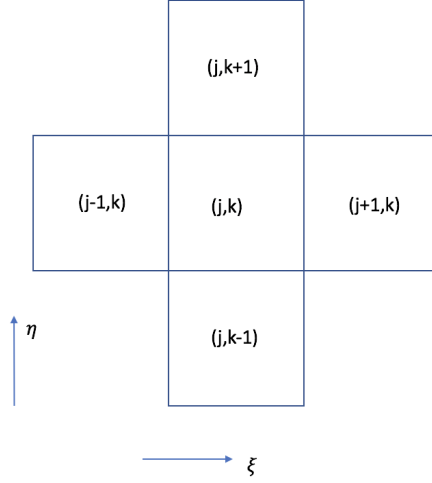


Figure 1: Sketch of a computational cell, after BK Figure 2.

Corrector Step

The corrector solution is obtained from the conservative form of the governing equations.

The predictor solutions are reconstructed to the left and right of each cell face using the monotone upstream scheme for conservation laws (MUSCL), which achieves second-order spatial accuracy. The reconstructed predictor values define a Riemann problem at each cell face, which are used to compute the interfacial fluxes:

$$\frac{\mathbf{U}_{j,k}^{n+1} - \mathbf{U}_{j,k}^n}{\Delta t} + \frac{1}{\Omega_{j,k}} \left[-\mathbf{F}_{\perp 1}^{n+1/2} \Delta s_1 + \mathbf{F}_{\perp 2}^{n+1/2} \Delta s_2 + \mathbf{F}_{\perp 3}^{n+1/2} \Delta s_3 - \mathbf{F}_{\perp 4}^{n+1/2} \Delta s_4 \right] = \mathbf{Q}^{n+1/2}$$

where \mathbf{Q} and \mathbf{U} are the cell-center values in cell j, k with area Ω , \mathbf{F} and \mathbf{G} are the average boundary values on each cell face, and δs is the length of the cell faces. The indices 1 through 4 denote the four cell faces: index 1 corresponds to the bottom cell face, and the remaining cell faces are numbered in counter-clockwise order.

The flux \mathbf{F} is normal to the cell boundary and positive in the direction of increasing cell coordinates. \mathbf{F} is defined as:

$$\mathbf{F}_{\perp} = \begin{bmatrix} hu_{\perp} \\ huu_{\perp} + \frac{1}{2}gh^2 \cos \phi \\ hvu_{\perp} + \frac{1}{2}gh^2 \sin \phi \end{bmatrix}$$

where u_{\perp} is the velocity perpendicular to the cell face, and ϕ is the angle between the face normal vector and the x -axis.

The fluxes are evaluated using a Godunov-type upwind scheme in which a Riemann problem is solved across each cell face, using the method of Roe (1981). Further details can be found in BK.

The source term

The source term \mathbf{Q} contains the parameterization of the surface roughness (via the friction slope S_f and the lateral inputs (rainfall p and infiltration i). Infiltration is independently modeled at each timestep and grid cell with the 1D Richards equation, described in the next subsection.

The friction slope is specified in the generalized form:

$$S_{f,x} = \left(\frac{\alpha U}{h^m} \right)^{1/\eta} \frac{|U|}{U}; \quad S_{f,y} = \left(\frac{\alpha V}{h^m} \right)^{1/\eta} \frac{|U|}{V}$$

where α is a roughness parameter, and m specifies the flow regime ($m = 2$ for laminar flow, $1/2$ for turbulent flow). $\eta=1/2$ for most roughness schemes, with the exception of laminar flow, for which $\eta=1$. For Manning's equation, $\alpha = n$, $m = 2/3$ and $\eta = 1/2$:

$$S_{f,x} = \frac{n^2 U}{h^{4/3}} |U|; \quad S_{f,y} = \frac{n^2 V}{h^{4/3}} |U|$$

More generally, with $\eta = 1/2$:

$$S_{f,x} = \frac{\alpha^2}{h^m} U |U|; \quad S_{f,y} = \frac{\alpha^2}{h^m} V |U|$$

Richards equation

Richards equation is solved following the approach outlined by *Celia et al.* (1990), which involves a backward Euler approximation in time coupled with a simple Picard iteration scheme. The solver used the discrete approximation of the mixed H - θ form:

$$\frac{\partial \theta}{\partial t} - \nabla \cdot K \nabla H - \frac{\partial K}{\partial z} = 0 \tag{4}$$

where z denotes the vertical dimension (assumed positive upwards), θ is the soil moisture content, H is the matric potential and K is the unsaturated hydraulic conductivity. θ and matric potential H are related via the Van Genuchten water retention curve. The volumetric soil moisture content, θ , and effective saturation, S_e , are computed as:

$$\theta = \frac{\theta_S - \theta_R}{1 + (\alpha |H|)^n)^m} + \theta_R$$

$$S_e = \frac{\theta - \theta_R}{\theta_S - \theta_R}$$

where θ_S and θ_R are the saturated and residual soil moisture content; n is a measure of the pore size distribution; $m = (1 - 1/n)$; and α is related to the inverse of the air entry suction. The unhydraulic conductivity K is computed as:

$$K = K_s \sqrt{S_e} [1 - (1 - S_e^{1/m})^m]^2$$

where K_s is the saturated hydraulic conductivity. The infiltration rate is solved with Darcy's law:

$$q = -K \left(\frac{\partial H}{\partial z} + 1 \right) \quad (5)$$

where the 1 (second term) on the RHS of Equation 5 reflects the fact that H is the matric head, as opposed to the hydraulic head.

Coupling the SVE and Richards Equation models

The model components are coupled at each grid cell in two steps: the depth from the SVE solver provides the surface boundary condition to the Richards equation solver, and the infiltration rate from the Richards equation solver is used by the SVE source term. This requires that several cases be accounted for: (1) no rain and no ponding, (2) rain but no ponding, and (3) ponding (with or without rain). In case (1), a no flux boundary condition is applied at the surface. In case (2), the Richards equation solver computes a potential infiltration rate (PI), defined as the infiltration rate that would occur with $H = 0$ cm at the surface, and compares this value to the rainfall intensity, p . If p exceeds the potential infiltration rate, ponding begins and the boundary condition switches to case (3). Otherwise, the potential infiltration rate is greater than p , and $i = p$. Finally, in case (3), the upper boundary condition H is equal to the ponding depth h . These cases are schematically illustrated in Figure 2.

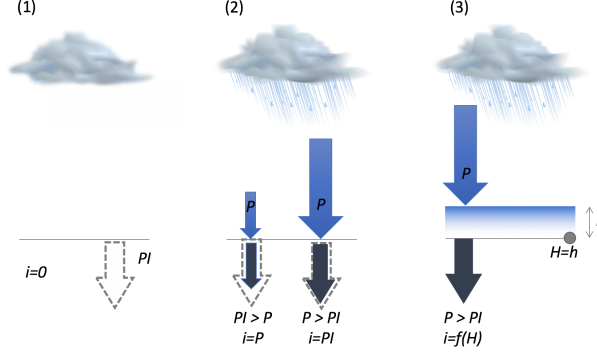


Figure 2: Schematic to illustrate the boundary condition cases for Richards equation. PI is the potential infiltration rate, which is computed to determine whether the rain intensity exceeds the infiltration capacity of the soil (in which case ponding occurs).

3 SVE-R model Fortran code

3.1 SVE-R model structure

The core of the model, `dry.f`, is written in Fortran, and involves a number of auxiliary files (see screenshot in Figure 3), including:

- `dry.inc` : specifies common variables used by `dry.f` (a number of variables are “common” variables, and not explicitly returned by the Fortran subroutines).
- `input` : files to initialize `dry.f` are located in this folder
- `output` : `dry.f` saves the outputs to this `output` folder.
- `.ipynb` files: python notebooks to visualize the simulation outputs.
- `params.json` : a used supplied parameter dictionary
- `sw` : the compiled Fortran executable.

Figure 4 summarizes the workflow of `dry.f`, which can be divided into main two steps: (1) initializing and (2) running the model. The following subsections describe `dry.f` components in greater detail.

3.2 Initializing the SVE solver

The `init` subroutine initializes the SVE component of the model, which involves reading a number of scalar parameters from `params.dat` and setting up the grid. The infiltration and parameters are initialized in the subroutine `initVG`. The following briefly describes the input files, and further details are included in Appendix (including an illustrative example of a 2×2 grid)



Figure 3: Summary of the file structure associated with `dry.f`.

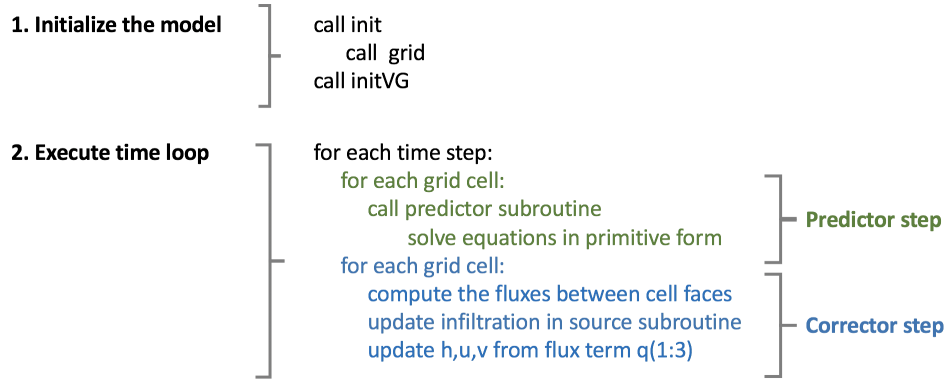


Figure 4: Summary of the SVE-R model structure in `dry.f`.

Input files

- `params.dat` : `params.dat` specifies a number of scalar parameters, and is read by the `init` subroutine of `dry.for`. See Appendix for a list of the input parameters and sample `params.dat` file.
- `coords.dat` : contains the x,y,z coordinates at the cell nodes.
- `boundary.dat`: Describes the boundary types and locations.
- `veg.dat` : Contains vegetation pattern. Written by `input_veg.py`.
- `nodes.dat` : Numbered list of the nodes surrounding each grid cell.
- `vanG.dat` : Van Genuchten parameters and initial H as a function of depth for vegetated and bare soil cells.

`coords.dat`

`coords.dat` The first line contains the number of points (`npt = (ncol+1)*(nrow+1)`) and the number of cell edges (`ne = nrow*ncol`).

`boundary.dat`

The first row contains the number of boundary cells, defined as a grid cell with one or more boundaries. columns are `j k inum itype ipos`.

`input.boundary.py` loops over all of the boundary cells, and records the boundary location `j,k`, number `inum`, type `itype`, and position `ipos`. Following this, the following rows record: the number of columns and rows, and, for each column, the beginning and final row (i.e. `kbeg = 1` and `kend = nrow`).

Boundary types are:

- `itype = 1` for a closed boundary
- `itype = 0` for an open boundary
- `itype = 4` for a subcritical fixed flux boundary (limited support)
- `itype = 5` for a supercritical influx boundary (no support).

For subcritical fixed flux boundary, the normal depth is computed with Manning's equation with $n = 0.1$. The input parameter `influx` is specified in units m^2/s . The rainfall rate (in cm/hr) can be related to `influx` as: $p \text{ (cm/hr)} = q \text{ (m}^2/\text{s)} \cdot 3600 \text{ s/hr} / (L_y \text{ m}) \cdot 100 \text{ cm/m}$ or $q \cdot 3.6e5 / L_y$.

Equivalent `influx` is $p * L_y / 3.6e5 = \text{equivalent } q \text{ (m}^2/\text{s)}$

Grid Set-up

- `nn` : space allocated to grid (`nn > np`).
- `np`: the number of nodes.
- `x(nn)`, `y(nn)`, `zz(nn)` : x, y, z coordinates of the cell nodes.
`x, y, zz` are read as 1D arrays, and interpolated to a 2D grid with the help of the nodes file `nodes.dat`.
- `xc`, `yc`, `zc` : coordinates at the cell centers, with dimensions (`ncol, nrow`)
- `dxi = dξ`, `dxi(1), dxi(2) = ξx, ξy`
- `deta = dη` `deta(1), deta(2) = ηx, ηy`
- `sx = dz/dx`, `sy = dz/dy`
- `nop(nx, ny, 4)` : node numbers defining each grid cell (see Figure 5).

- **inum** : number of boundary interfaces of each grid cell.
- **itype** : interface type of each boundary (**itype** = 1 for wall boundaries, 0 for open boundaries,..).
- **ipos** : Position of each boundary, with dimensions (ncol, nrow, 4)

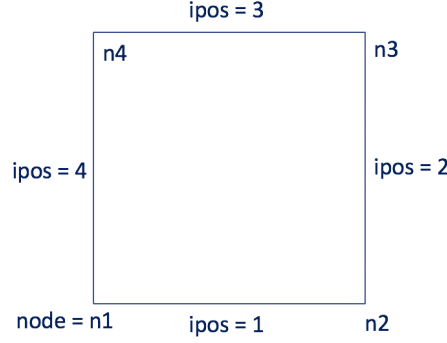


Figure 5: Schematic of the nodes and boundary naming convention.

2x2 Grid example

`nop(1,1,:) = [1,4,5,2]` means that cell (1,1) is bounded by nodes **n1=1**, **n2=4**, **n3=5**, **n4=2**, starting from the lower left node **n1** moving counterclockwise. `itype(j,k,2)=1`, `itype(j,k,3)=4`, means that cell (j,k) has a closed boundary at position 2 (right) and a subcritical influx boundary at position 3 (top). See Figure 5.

vertical cell faces have index 1, horizontal faces have index 2

3.3 The predictor step

The **predict** subroutine is called for each cell to obtain **hp**, **up**, **vp** (corresponding to $h^{n+1/2}, u^{n+1/2}, v^{n+1/2}$). **predict** modifies the common variables **hp**, **up**, **vp**, **dh**, **du**, **dv**, **qs**.

The following provides an (incomplete) correspondence between BK equations and Fortran code, with the mathematical notation in *italics* and the equivalent code in **typewriter**. For example, for the array of primitive variables:

$$\mathbf{W}_T = \begin{bmatrix} h \\ U \\ V \end{bmatrix} = \begin{bmatrix} \mathbf{h} \\ \mathbf{u} \\ \mathbf{v} \end{bmatrix}$$

The matrices **A_W** and **B_W** are defined as:

$$\mathbf{A}_w = \begin{bmatrix} U_\xi & h\xi_x & h\xi_y \\ g\xi_x & U_\xi & 0 \\ g\xi_y & 0 & U_\xi \end{bmatrix} = \begin{bmatrix} \text{uxi} & h \text{ dxi}(1) & h*\text{dxi}(2) \\ g \text{ dxi}(1) & \text{uxi} & 0 \\ g \text{ dxi}(2) & 0 & \text{uxi} \end{bmatrix}$$

$$\mathbf{B}_w = \begin{bmatrix} U_\eta & h\eta_x & h\eta_y \\ g\eta_x & U_\eta & 0 \\ g\eta_y & 0 & U_\eta \end{bmatrix} = \begin{bmatrix} \text{ueta} & h*\text{deta}(1) & h*\text{deta}(2) \\ g \text{ dxi}(1) & \text{ueta} & 0 \\ g \text{ dxi}(2) & 0 & \text{ueta} \end{bmatrix}$$

$$\partial_\xi \mathbf{W} = \begin{bmatrix} \partial_\xi h \\ \partial_\xi u \\ \partial_\xi v \end{bmatrix} = \begin{bmatrix} \text{dh}(1) \\ \text{du}(1) \\ \text{dv}(1) \end{bmatrix}; \quad \partial_\eta \mathbf{W} = \begin{bmatrix} \partial_\eta h \\ \partial_\eta u \\ \partial_\eta v \end{bmatrix} = \begin{bmatrix} \text{dh}(2) \\ \text{du}(2) \\ \text{dv}(2) \end{bmatrix}$$

where $\text{dh}(1)$ implies $\text{dh}(j, k, 1)$, and similarly for $\text{du}(1)$ and $\text{dv}(1)$.

$\mathbf{A}_w \partial_\xi \mathbf{W}$ is computed as:

$$\mathbf{A}_w \partial_\xi \mathbf{W} = \begin{bmatrix} U_\xi & h\xi_x & h\xi_y \\ g\xi_x & U_\xi & 0 \\ g\xi_y & 0 & U_\xi \end{bmatrix} \begin{bmatrix} \partial_\xi h \\ \partial_\xi u \\ \partial_\xi v \end{bmatrix} = \begin{bmatrix} U_\xi \partial_\xi h + h\xi_x \partial_\xi u + h\xi_y \partial_\xi v \\ g\xi_x \partial_\xi h + U_\xi \partial_\xi u \\ g\xi_y \partial_\xi h + U_\xi \partial_\xi v \end{bmatrix}$$

Equivalently in code:

$$\mathbf{A}_w \partial_\xi \mathbf{W} = \begin{bmatrix} \text{uxi} & h \text{ dxi}(1) & h \text{ dxi}(2) \\ g \text{ dxi}(1) & \text{uxi} & 0 \\ g \text{ dxi}(2) & 0 & \text{uxi} \end{bmatrix} \begin{bmatrix} \text{dh}(1) \\ \text{du}(1) \\ \text{dv}(1) \end{bmatrix} =$$

$$\begin{bmatrix} \text{uxi}*\text{dh}(1) + h*(\text{dxi}(1)*\text{du}(1) + \text{dxi}(2)*\text{dv}(1)) \\ g*\text{dxi}(1)*\text{dh}(1) + \text{uxi}*\text{du}(1) \\ g*\text{dxi}(2)*\text{dh}(1) + \text{uxi}*\text{du}(1) \end{bmatrix}$$

The `limitr` subroutine contains the flux limiter that the predictor step uses to compute the cell-average spatial gradients (which preserves solution monotonicity by becoming first-order accurate near discontinuities, yet remains second-order accurate elsewhere). Various choices are included, with the parameter `ilim` specifying the averaging type. `ilim=5` instructs the code to use the β family of averages, and is given by:

$$\overline{\Delta \mathbf{W}} = \text{sign}(a) \min[\max(|a|, |b|), \beta(|a|, |b|)] \text{ if } ab > 0$$

$$0 \text{ if } ab \leq 0$$

$\beta = 1$ yields the relatively more dissipative Minmod average, and $\beta = 2$ yields the less dissipative Superbee average.

3.4 Corrector step

The corrector step computes the fluxes between the cell interfaces by calling the `fluxes` subroutine for each cell j,k and for each interface.

The interfacial cell fluxes at a given time-step are stored in the common variable `f(0:nx,0:ny,1:3,1:2)`, where the first two indices of `f` contain the j,k coordinates, the third index correspond to the components of \mathbf{F}_\perp (see Equation), and the final index denotes the cell face (1 for vertical cell faces and 2 for horizontal). `f(j,k,1:3,1)` represents the flux from cell $(j-1,k)$ to (j,k) , and `f(j,k,1:3,2)` represents the flux from cell $(j,k-1)$ to (j,k) (see Figure ??). Calling `fluxes(j-1,j,k,k,1)` modifies `f(j,k,1:3,1)` and calling `fluxes(j,j,k-1,k,2)` `f(j,k,1:3,2)`.

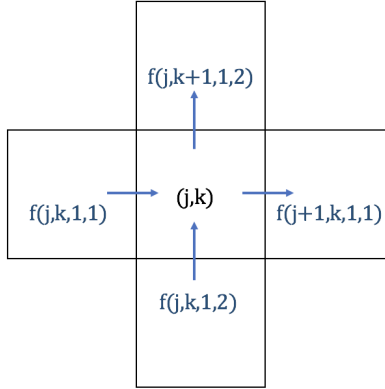


Figure 6: Definitional sketch for the interfacial fluxes, indicating how the fluxes into and out of cell (j,k) are stored in the array `f(0:nx,0:ny,1:3,1:2)`

Fluxes subroutine

Within the `fluxes` subroutine, `fluxes(jl,jr,kl,kr,i1)` computes the flux from cell (jl,kl) to (jr,kr) , modifying the common variable `f(jr,kr,1:3,i1)`. `i1 = 1` indicates that the flux is between vertical cell faces, i.e. from cell $(j-1,k)$ to (j,k) . Similarly, `i1 = 2` indicates that the flux is between horizontal cell faces, i.e. from cell $(j,k-1)$ to (j,k) .

`fluxes` first reconstructs the predicted values to the left and right of each face (`hl,hr,ul, ur,vl,vr`) using the monotone upstream scheme for conservation laws (MUSCL). These reconstructed predictor values define a Riemann problem at each cell face, and `fluxes` calls the `solver` subroutine to compute $\mathbf{F}_{\perp\mathbf{I}}$ from Equation ??, reproduced below:

$$\mathbf{F}_{\perp\mathbf{I}} = \frac{1}{2}(\mathbf{F}_{\perp\mathbf{L}} + \mathbf{F}_{\perp\mathbf{R}} - \hat{\mathbf{R}}|\hat{\mathbf{\Lambda}}|\Delta\hat{\mathbf{V}})$$

Solver subroutine

This subroutine uses the monotone upstream scheme for conservation laws (MUSCL) to compute \mathbf{F}_\perp at each cell face. The code in `solver` corresponds to the Equation ?? as:

- $\hat{\mathbf{R}} : \mathbf{e}(3,3)$
- $|\hat{\mathbf{\Lambda}}| : \mathbf{a}(3) \times$
- $\Delta \hat{\mathbf{V}} : \mathbf{ws}(3)$

$$\hat{\mathbf{R}} = \begin{bmatrix} 1 & 0 & 1 \\ \hat{u} - \hat{a} \cos \phi & -\sin \phi & \hat{u} + \hat{a} \cos \phi \\ \hat{v} - \hat{a} \sin \phi & \cos \phi & \hat{v} + \hat{a} \sin \phi \end{bmatrix} = \begin{bmatrix} 1 & 0 & 1 \\ \text{uhat-chat*cndum} & -\text{sndum} & \text{uhat+chat*cndum} \\ \text{vhat-chat*sndum} & \text{cndum} & \text{vhat+chat*sndum} \end{bmatrix}$$

$$|\hat{\mathbf{\Lambda}}| = \begin{bmatrix} |\hat{u}_\perp - \hat{a}| & & \\ & |\hat{u}_\perp| & \\ & & |\hat{u}_\perp + \hat{a}| \end{bmatrix} = \begin{bmatrix} |\text{uperp} - \text{chat}| & & \\ & |\text{uperp}| & \\ & & |\text{uperp} + \text{chat}| \end{bmatrix}$$

$$\Delta \hat{\mathbf{V}} = \begin{bmatrix} \frac{1}{2} \left(\Delta h - \frac{\hat{h} \Delta u_\perp}{\hat{a}} + \frac{\hat{h} \Delta c_T}{2c_T} \right) \\ \hat{h} \Delta u_\parallel \\ \frac{1}{2} \left(\Delta h + \frac{\hat{h} \Delta u_\perp}{\hat{a}} + \frac{\hat{h} \Delta c_T}{2c_T} \right) \end{bmatrix} = \begin{bmatrix} 0.5*(\text{dh dum} - \text{hhat*duperp/chat}) \\ \text{hhat*dupar} \\ 0.5*(\text{dh dum} + \text{hhat*duperp/chat}) \end{bmatrix}$$

- $\Delta h : \text{dh dum} = \text{hr} - \text{hl}$
- Left interface: $\text{hl} = \text{hp}(\text{j1}, \text{k1}) + 0.5*\text{dh}(\text{i1})$
- Right interface: $\text{hr} = \text{hp}(\text{j1}, \text{k1}) - 0.5*\text{dh}(\text{i1})$

3.5 The source subroutine

The source term is computed with the subroutine `source`, which modifies the common variable `qs` (or $Q(j, k)$). `source` is called by both the predictor and corrector steps; however, Richards equation is only solved during corrector step. The input arguments are the cell indices `j, k`, the primitive variables, and an indicator variable `lstep` for the step type (`lstep = 0` for the predictor step and 1 for the corrector step).

In the predictor step, `source` is called for each cell before updating the predictor values (`hp, up, vp`):

```
call source(j, k, h(j,k), u(j,k), v(j,k), 0)
```

`source` is similarly called in the corrector step before updating the fluxes:

```
call source(j, k, hp(j,k), up(j,k), v[(j,k), 0])
```

Inside `source`, the primitive variables are labeled `depth`, `udum`, `vdum`. The source term in Fortran code is given as:

$$\mathbf{Q} = \begin{bmatrix} \text{winflt} \\ -\text{grav*depth*sx}(j,k) - \text{grav*depth*fricSx} + 0.5*\text{udum*winflt} \\ -\text{grav*depth*sy}(j,k) - \text{grav*depth*fricSy} + 0.5*\text{vdum*winflt} \end{bmatrix}$$

where $S_{f,x} = \text{fricSx}$ and $S_{f,y} = \text{fricSy}$, and $p - i = \text{winflt}$.

Infiltration details (corrector-step)

In the corrector step, `source` calls the infiltration-specific subroutines (`timestep` and `potential` to update the infiltration rate i and the surface boundary conditions at cell j,k . `timestep` solves Richards equation at cell (j,k) and returns the updated H , θ and K , which `source` uses to compute the infiltration rate i . `potential` is used to prescribe the boundary conditions.

The Richards equation solver is implemented separately for each grid cell, and the 3-dimensional H , θ and K fields are saved as the common variables `r8H` (cm), `r8Theta`, and `r8K` (cm/s), respectively, with dimensions (`nrow` \times `ncol` \times `nz`).

From Darcy's law, the infiltration rate i (cm/s) is computed as:

$$i = K \left(\frac{\partial H}{\partial z} + 1 \right)$$

which corresponds to `r8kt*((hdum(nz)-hdum(nz-1))/dz+1)` in `source`.

As an intermediate step, the depth is updated with:

$$\text{znew} = \text{zold} + \text{prate} * 100 * \text{dt} - \text{r8kt} * ((\text{hdum}(\text{nz}) - \text{hdum}(\text{nz}-1)) / \text{dz} + 1) * \text{dt}$$

where `prate` rainfall has been converted to cm/s. `winflt` = $p - i$ is then computed as:

$$\text{winflt} = (\text{znew} - \text{zold}) / \text{dt} / 100.$$

The surface boundary condition cases are:

- Case 1: Richards equation is solved every `iscale` timesteps. For all other SVE time steps, the infiltration rate is estimated as:

$$\text{r8kt} * ((\text{hdum}(\text{nz}) - \text{hdum}(\text{nz}-1)) / \text{dz} + 1.0) * \text{dt}$$

if the depth is greater than zero. If $\text{depth} = 0$, $q(s)=0$, so that $p = i$ between Richards solver updates. Ponding does not occur until the `potential` subroutine determines that the rainfall exceeds to potential infiltration rate ($p > PI$).

- Case 2: Rain and no ponding : `depth=0` and `prate > 0`.

The potential subroutine is called to estimate a potential infiltration rate PI , defined as the infiltration that would occur with $H = 0$ at the surface (in cm/s).

If the potential infiltration rate is less than the rainfall rate (`(PI .lt. prate*100)`), ponding begins. In this case, the Richards boundary condition is switched to fixed H , and the timestep subroutine is called.

3.6 timestep The Richards equation subroutine

Richards equation is solved by the `timestep` subroutine, which is called from `source`.

timestep inputs and outputs

The inputs are `hnp1m`, `thetan`, which are the initial conditions to the Richards eqn solver (`hdum`, `thetadum` in the source subroutine, which calls `timestep`). The outputs are `hnp1mp1`, `r8thetanp1m`, `r8knp1mp1`, which are H , θ and K at the following timestep (after `dt_r` time elapsed).

When the soil is flagged as saturated, the surface flux is set to K at the surface, which should be very close to saturated (`flux = - r8knp1m(nz)`). This is achieved by adjusting the value of H at node (`nz-1`):

$$\text{hnp1mp1}(\text{nz}-1) = \text{hnp1mp1}(\text{nz}) + \text{dz} + \text{flux} * \text{dz} / \text{r8knp1m}(\text{nz}-1)$$

Note: this approach should be treated with caution. It was designed for the use case of a fixed intensity rain storm, in which a uniform wetting front would arise. It is not meant for variable rainfall cases.

Stop tolerance

`stop_tol0` is the stop tolerance specified in the input `params.dat`, with default value `stop_tol0 = .01`. `stop_tol` is reset to the original `stop_tol0` at the beginning of each Richards solver timestep. The convergence tolerance is relaxed if the Richards solver fails to converge within a specified number of iterations (default = 100).

Potential infiltration

The subroutine `potential` is used to compute the potential infiltration rate PI , defined as the infiltration that would be observed for a surface boundary condition of $H = 0$. Potential infiltration is computed when there is rain but no ponding, and returns the potential infiltration rate, defined as the infiltration rate that would occur with $H = 0$ at the surface.

4 Running the model

The Fortran code can be compiled from the terminal as `gFortran -o ./sw -framework accelerate ./dry.for` and executed as `.sw`, where `sw` is the name of the compiled executable.

`input_coords.py` / `wrap_coords`

`sim_input.py` calls `wrap_coords` for each path / parameter case:

```
wrap_coords(path + '/input', params)
```

which calls the following:

- `write_nodes(path, ncol, nrow)` : writes the cell node indices to `input/nodes.dat`. This function assumes that the grid is rectangular, and would need to be modified for a non-rectangular grid. It saves `nop`, which contains the indices of the nodes surrounding each cell face.
- `codebuild_coords(params)` : constructs the `x,y,z` fields.

4.1 Output files

Most outputs are saved at intervals of `dt_p`, to limit the size of the output files.

- `time.out` : file containing time, and max CFL number at each timestep.
- `h.out` : contains h, U, V ,
- `summary.out` : summarizes ponding time, runtime, final time, and if applicable, the reason for an early exit (i.e. no more water, AMAX too big)
- `hydro.out` : hydrograph at 1 second time resolution. The hydrograph is saved by Fortran with m^3/s units, and normalized by the hillslope dimensions to convert to cm/s in Python. Read by function `new_hydro.out` in `output_dry.py`.
- `dvol.out` : output file to check mass balance. Read by function `get_dvol` in `output_dry.py`. See details below.
- `ptsTheta.out` (optional) : evolution of the soil moisture profiles at two points, one vegetated and one bare.

`h.out`

Columns are `j,k,h,u,v,zinflmap2, xflux0,yflux0,xflux1,yflux1`.

These variables are written to `h.out` for every grid cell (every `j,k`) when `myoutput` is called, every `nprt` timesteps (every `dt_p`).

`zinflmap2` contains the total volume of water infiltrated in cell `j,k`, and is reset to zero when this variable is written to file.

dvol.out

Contains output from Fortran mass balance tracking. Columns are `dvol`, `flux`, `infl`, `zain` from Fortran mass tracking, containing the change in surface volume, horizontal fluxes out of the domain, infiltration and rain volumes, respectively, in units m^3 .

The Python function `get_dvol` in `output_dry.py` reads only the first three columns `dvol`, `flux`, `infl`, and the last is available for mass balance debugging. `get_dvol` normalizes by the domain area ($L_x * L_y$) and converts to cm.

5 The Simulated Domain

To simulate runoff/runon environments on patchily-vegetated hillslopes, several aspects of the BK model have been modified (in addition to coupling to a 1D Richards solver). This section describes the simulated domain, as illustrated in Figure 7.

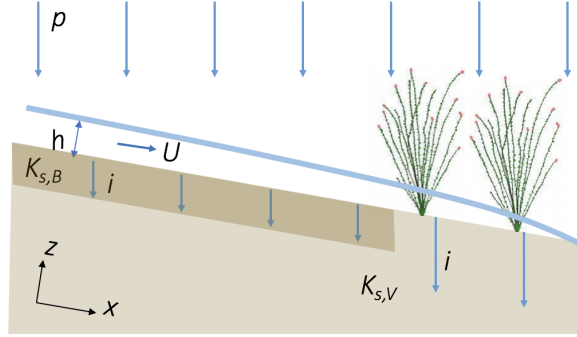


Figure 7: Schematic of the model domain. The darker shading under the non-vegetated regions represents the surface crust. $K_{s,V}$ and $K_{s,B}$ represent the saturated hydraulic conductivities of undisturbed soil and surface crust, respectively.

First, roughness parameters are specified separately for vegetated and bare cells. The parameters for the resistance formula for vegetated cells are specified as `alpha`, `eta` and `m` (α , η and m), and for bare soil cells as `alpha_B`, `eta_B` and `m_B` (α_B , η_B and m_B). The default soil parameters are Manning's equation with `alpha` = 0.1, and `alpha_B` = 0.03.

The model is set up to specify rain-driven overland flow on a planar hillslope, for which the lateral boundaries are closed with the exception of the open boundary at the bottom of the hillslope. Limited support is provided for a fixed flux (subcritical) upslope boundary condition, where the inflow boundary dimensions are equal to the entire upslope area. Code for other boundary conditions is not provided, but can be implemented with modification to `input.coords.py` (or directly to the Fortran input parameter functions). For a fixed flux upslope boundary, `tr` also specifies that the time as which the inflow stops (i.e. the boundary changes from fixed flux to a wall boundary). In the case of rain AND upslope inflow, for example, to simulate rain on a longer hillslope domain, the rain and inflow will end

at the same time. The code would need to be modified with the addition of a new variable (e.g. time of inflow end) to change this, which would be straightforward.

The bare soil areas are represented by a two-layer model, where the upper, less-permeable layer represents a surface crust. With the exception of K_s , the van Genuchten parameters are the same between vegetated and bare cells. In the bare soil areas, K_s of the lower soil layer is equal to that in the vegetated patches (See schematic). The van Genuchten parameters and vertical structure of the soil can be modified in `input_phi.py`.

6 Python wrapper scripts

Example Python scripts are provided to generate the Fortran input files, execute `dry.f` and read the outputs. `call_dry.py` is a control script which executes each of these components, as illustrated in the Figure 8 schematic.

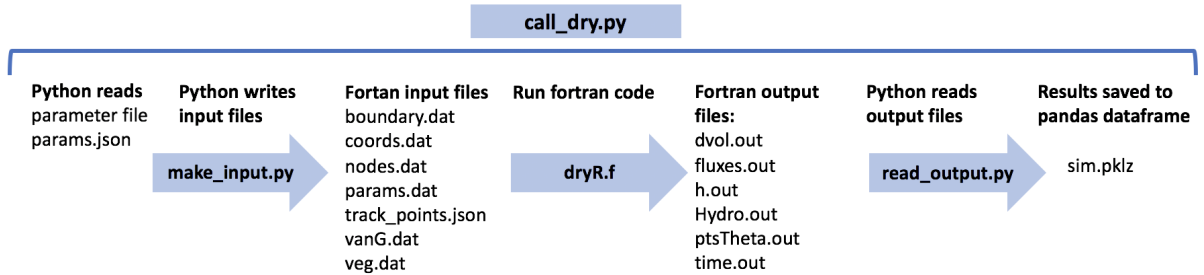


Figure 8: Schematic illustrating the python wrapper script workflow. `call_dry` takes as input `sim_name`, the name of a subfolder containing the parameter file `params.json`.

`output_dry.py` contains three main sub-functions, listed below:

- `call_dry.py`
 - `sim_input.py`
 - `runmodel`
 - `sim_read.py`

`call_dry.py` is a control script which writes the Fortran input files, compiles and executes the Fortran code `dry.for`, and reads the output files.

6.1 `sim_input.py` writes Fortran input files

`sim_input.py` reads and interprets the parameter file `params.json` located in the simulation folder `sim_name`. See Appendix B for examples of the Fortran input files. `sim_input` specifies default pa-

rameters where no parameters are provided in `params.json`. For example, if `ncol` is not provided, an aspect ratio must be included instead (`aspect`).

`sim_input.py` calls additional functions to write the Fortran input files located in `sim_name/input`:

- `write_param`: writes the general parameter file `params.dat`.
- `input_veg.py`: contains functions to write the input vegetation file `veg.dat`.
- `input_coords.py`: contains functions to write the grid and topography files `coords.dat`, `boundary.dat`, `nodes.dat`.
-

6.2 Parameters specified in `params.json`

SVE parameters

- `dx` : grid discretization (default = 1.)
- `nrow` : Number of grid cells in the longitudinal (along-slope) direction (default = 40).
Hillslope length $L_y = \text{nrow} * \text{dx}$
- `ncol` : Number of grid cells in the transverse (across-slope) direction (default = 10).
Hillslope width $L_x = \text{ncol} * \text{dx}$
- `topo` : default = "plane", specifying a planar hillslope, aligned along the y-axis. other option is to specify coordinates.
- `vegtype` : specified how the vegetation field is built
 - `randv` : random vegetation field, defined by σ_V and f_V
 - `image` : read from image file.
- `scheme` : resistance formulation (default in Manning equation)
specifying the scheme is equivalent to specifying eta and m (see source subroutine, where roughness formulation is implemented. Explicitly specifying m and/or eta will overwrite these default parameters.
- `alpha` / α_V : generalized roughness parameters for vegetated / permeable areas. If Manning's equation is used to specify the resistance formulation, then $\alpha_B = n_B$.
- `alpha.B` / α_B : equivalent to `alpha` / α_V for bare soil areas.
- `epsh` / ϵ : depth tolerance for dry cells. The momentum equations are not solved if $h < \text{epsh}$ ($h < \epsilon$).
- `beta` / β : a constant in limiter subroutine if `ilim` = 5, beta family (default = 1).

Infiltration parameters

- **KsV** : infiltration rate in vegetated/permeable areas, in cm/hr.

Note **ksatV**=**KsV**/3600. is the infiltration rate in units of cm/s.

- **KsB** : infiltration rate in bare soil/impermeable areas, in cm/hr.

Note **ksatB**=**KsB**/3600. is the infiltration rate in units of cm/s.

If **KsB** is not specified, then **KsB** is set to **KsV/s_scale**, where **s_scale** is the ratio.

- **s_scale** : the soil infiltration ratio between vegetated and non-vegetated (or permeable/impermeable) areas.
- **stop_tol** : error tolerance for Richards equation convergence, default = 0.01
- **tsat_min**: default = 600. Specified in seconds. Minimum time until the soil is allowed to saturate (*i* is set to K_{sat} , bypassing the Richards equation solver.
- **H_i** : initial *H* at the surface. The soil is initialized to an equilibrium profile (this can be modified in **sim_input.py**.
- **tr** : storm duration (min)
- **p** : rain intensity (cm/hr)
- **rain** : rain intensity (mm/s)

6.3 runmodel: compiles and execute dry.f

The function **runmodel** is located in **call_dry.py**.

6.4 read_sim: reads the Fortran output files:

Output files are read by Python script **batch_read.py**, using functions from **output_dry.py**.

The Python function **get_h** in **output_dry.py** reads **h.out** and returns: primitive variables **h,u,v**, infiltration **inflVmap** and the inter-cell fluxes **xflux0,yflux0,xflux1,yflux1**, all with dimensions: **np.r_ x ncol x nrow**. Units are:

- **h** : m
- **u, v** : m/s
- **inflVmap** : cm m^2 (converted from m^2)
- **xflux0, yflux0, xflux1, yflux1** : cm m^2 (converted from m^2)

fluxes are positive directed out of the domain.

zinflc: infiltration depth in cm

7 Examples

7.1 example_2by2

is provided to illustrate the grid set-up (see Appendix).

7.2 example_inflow

illustrates a subcritical inflow boundary at the top of the hillslope, with $\text{rain} = 0$ and $K_s = 0$.

8 Other iPython notebook files

- GW : stand alone implementations, and code to compare to SVE-R model output.
- Richards equation : stand alon, basic and module versions
- Code to compare simulation output to the kinematic wave approximations $-i$ related to friction sims.
- Code to compare IF predictions by KWA, SVE-R and GW.
- roughness matching

Stand alone implementations

8.1 SVE-R model

Template notebooks to process / view the SVE-R simulation results:

- `illustrate.ipynb`: Illustrate simulation results
- `illustrate_3D`: Illustrate simulation results in 3D. Useful for visualizations.
- `illustrate_soils.ipynb`: Plot soil infiltration profile.
- `mass_balance_check.ipynb`: Check mass balance in the SVE-R model.
- `boundary_flux.ipynb`: Check lateral fluxes in the SVE-R model.

Template notebooks to view / modify how the SVE-R inputs are constructed

- `input_coords.ipynb`: Illustrate how the topography is created. Includes limited 3D plotting functionality. Case: planar...
- `input_veg.ipynb`: Illustrate how the vegetation is created. Case: randv, stripes (random widths and spacing), read image.

- `image_read.ipynb` : Code that reads image file and adjusts to the specified hillslope dimensions. After `wrap_image_veg` function in `input_veg.py`.

8.2 Giraldez and Woolhiser

- `Giraldez_Woolhiser.ipynb` : Basic implementation of the method of characteristics.
- `GW_check.ipynb` : Basic code to compare SVE-R simulations to GW prediction, for a single simulation.
- `mass_balance_with_GW` : Compare GW and SVE-R predictions of IF , rising and recession time (loop over all simulations).

Assumes Manning's roughness

8.3 Roughness

- `scheme_compare.ipynb`
- `scheme_compare.ipynb`

8.4 Richards stand-alone code

1. Richards solver implementation in Python (to assess sensitivity to Richards discretization)

A stand alone Python implementation is available in the jupyter notebook file `Richards.ipynb` and Python module `richards.py`.

2. GW code (to compare SVE-R and GW solutions. note, also includes code to estimate sorptivity from Van Genuchten parameters)

Richards alone ipynb notebooks

- `Richards_basuc.ipynb` : a simple ipynb implementation of the Richards equation solver, following *Celia et al. (1990)*. Translated from Alejandro Flores' Matlab version:

<https://leaf.boisestate.edu/blog/2010/11/07/richards-solver-code/>

- `Richards.ipynb` : uses `Richards.py` from `richards1D.py`. Need to add crust functionality and saturation logic to this module!

-

The stand-alone 1D Richards equation solver is structured as a class `Richards`, which takes as input a parameter dictionary `params`.

Appendix B: A 2×2 grid example

To illustrate the grid set-up, example input files are included here for the 2x2 grid example. Note that while the SVE-R model Fortran code does not assume a rectangular grid, the provided Python code and sample input files do assume a rectangular grid. Figure 9 shows the cell center and node indices for the example grid, and Figure 10 shows the boundaries, which are a fixed-flux subcritical boundary at the top of the hill, closed/wall lateral boundaries, and an open boundary at the bottom of the hill.

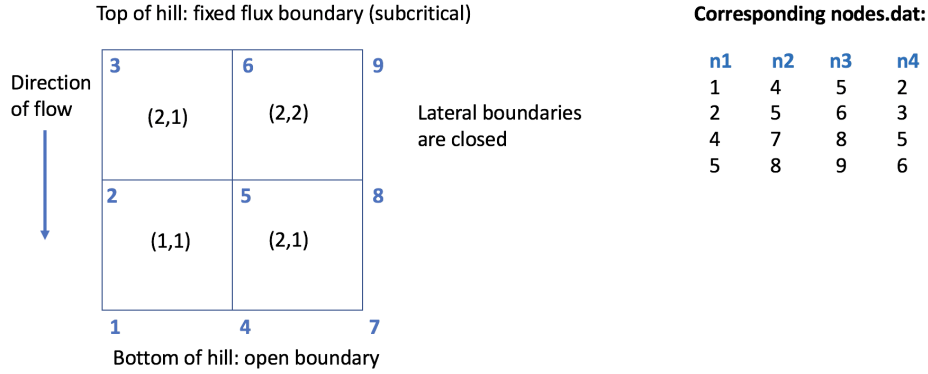


Figure 9: Example 2×2 grid to illustrate how cell centers and nodes are assigned. (j,k) coordinates are shown in the cell centers, and the node indices are at the cell corners.

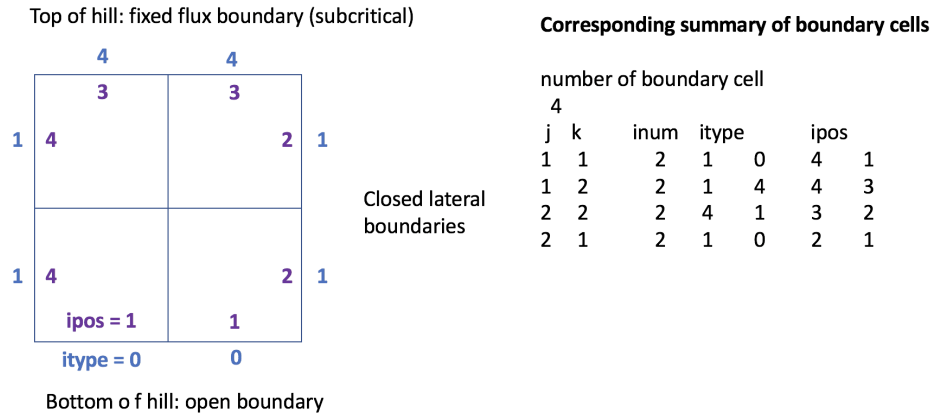


Figure 10: Boundaries for the 2×2 grid in Figure 10. The boundary types are labeled in blue outside the grid, with 0 for the open boundary, 1 for the closed boundaries and 4 for the fixed flux boundary.

8.5 params.dat

Sample `params.dat` files can be found in the `input` directories of the examples. The parameters specified in `params.dat` are:

- `grav` : acceleration due to gravity.
- `dt` : SVE timestep (`dt` is labeled `dt_sw` in the Python code).
- `tmax` : maximum time until the simulation ends.
- `tr` : rain duration in seconds (`tr` is labeled `t_rain` in the Python code).
- `prate` : rain intensity in m/s.
- `nt` : number of time steps (`tmax/dt`)
- `epsh` : depth threshold to solve the SVE momentum equations.
- `beta` : value of beta in `limitr` subroutine.
- `npert` : frequency f which the SVE-R output fields are saved: `npert= dt_p`. Note: the hydrograph is saved every second.
- `iscale` : ratio of SVE to Richards equation timestep durations.
- `stop_tol` : convergence criteria for Richards equation solver (default = 0.01).
- `h0`, `u0`, `v0` : initial depth, u , and v (defaults = 0).
- `r8mV`, `r8etaV`, `r8alphaV` : roughness parameters m , η and α for vegetated cells.
- `r8mB`, `r8etaB`, `r8alphaB` : roughness parameters m , η and α for bare cells.
- `jveg`, `kveg` : j , k indices of a vegetated cell for which the soil profile (H and θ) is saved every `npert` timesteps.
- `jbare`, `kbare` : j , k indices of a bare soil cell for which the soil profile (H and θ) is saved every `npert` timesteps.
- `tsat_min` : Minimum time until the soil is allowed to “saturate” (default =0).

8.6 boundary.dat

Sample `boundary.dat` for the 2x2 grid example, for which every cell is a boundary cell. The array on lines 3-7 show the (j,k) indices for each boundary cell, the number of boundaries associated with that cell, and boundary type of each boundary cell.

`inum` is the number of boundary faces in a boundary cell, and `ipos` is the orientation of boundary face: (bottom:1, right:2, top:3, left:4).

number of boundary cell

4						
j	k	inum	itype		ipos	
1	1	2	1	0	4	1


```

1      2      2      1      4      4      3
2      2      2      4      1      3      2
2      1      2      1      0      2      1
ncol
2
nrow
2
j      kbeg      kend
      1      1      2
      2      1      2
number of fixed bc cells, ndir
2
j      k      fix h      fix u      fix v
1      2      0.0      0.0      -2e-05
2      2      0.0      0.0      -2e-05

```

8.7 coords.dat

In the first row, the first value specifies the number of nodes (**npt**), and the second specifies the number of cells (**ne**). The remaining rows list the x,y,z coordinates at the nodes, in order of node number. Below is an example **coords.dat** file for the 2×2 example leg rid.

```

9      4
0.0      0.0      0.0
0.0      1.0      0.02
0.0      2.0      0.04
1.0      0.0      0.0
1.0      1.0      0.02
1.0      2.0      0.04
2.0      0.0      0.0
2.0      1.0      0.02
2.0      2.0      0.04

```

8.8 vanG.dat

Specifies the soil dimensions and van Genuchten parameters. See example below from The first two rows specify the soil depth (**zmax**) and soil layer thickness (**dz**), and the second two rows specify the number of soil grid points (soil layers + 1). Following this, the van Genuchten parameters for each soil layer in a vegetated cell, in order of decreasing soil depth (the last row is the top of the soil column). Note that all vegetated cells are initialized with the same parameters. From left to right, the columns are: **alpha**, **theta_s**, **theta_r**, **lambda**, **ksat**, **h_init**. This format is repeated for the bare soil cells.

```

dz zmax
1.0      20
nz

```

```

21
L
0.5
  vegetated cells
0.0096 0.472 0.0378 0.47 0.0005555555555556 -320.0
0.0096 0.472 0.0378 0.47 0.0005555555555556 -321.0
....
  bare soil cells
0.0096 0.472 0.0378 0.47 0.0005555555555556 -320.0
0.0096 0.472 0.0378 0.47 5.555555555556e-05 -321.0

```

Appendix C : dry.f pseudocode

8.9 Model overview/summary

```

call init ! grid subroutine called from the input subroutine
call myoutput ! write initial conditions to file

do it = 0,nt-1 ! loop over time steps (it = time iteration number)
  t = t+dt ! increment time (t = current time)

  ! Predictor step
  loop over grid cells:
    call bconds(j, k, h, u, v) ! update h,u,v values in ghost cells
    call predict(j, k) ! get predictor variables hp, up, vp

  ! Corrector step
  ! Corrector part 1: compute the fluxes between cell faces
  loop over grid cells:
    call bconds(j,k,hp,up,vp) ! update hp,up,vp in the ghost cells
    ! compute interfacial fluxes.
    call fluxes(j-1,j,k,k,1) ! vertical faces.
    call fluxes(j,j,k-1,k,2) ! horizontal faces.
  loop over boundaries:
    compute boundary fluxes

  ! Corrector part 2:
  loop over cells:
    call source ! update the source term qs
    compute q from qs and f ! q = (h,u,v)
    ! update h,u,v from q. check
    if q > 0: ! check for negative depth.
      h = q(1) ! if positive depth, update h
    else:
      q = 0 ! if negative depth, reset h and q(1) to zero

```

```

        h = 0

        if h < epsh:      ! neglect momentum in nearly dry cells
            u, v = 0.      ! set u and v to 0 in nearly dry cell
            q(2:3) = 0      ! set momentum fluxes to 0

        elif h >= epsh:    ! store u and v values
            u, v = q(2)/h, q(3)/h

        ! Exit when the ponded water is less than min_vol
        if volume < min_vol
            call gracefulExit

        ! Exit when the CFL number is too big
        if(CFL > 10) then
            call gracefulExit

        ! Exit if time runs out
        if(t > tmax) then
            call gracefulExit

\end{verbatim}

\subsection{Grid pseudocode}
Pseudocode for grid setup:

\begin{lstlisting}
subroutine grid
include "dry.inc" ! include common variables
read np, ne      ! np: number of grid points, ne : number of cells
read x, y, zz    ! from coords.dat, coordinates of the cell nodes
read veg         ! from veg.dat, vegetation at the cell nodes
read nop         ! from nodes.dat, the node numbers surrounding each cell.

! Compute grid metrics.
for each cell (j,k):
    n1, n2, n3, n4 = nop(j,k,1:4) ! get the node numbers
    ! compute xc, yc, zc as the average of x,y, zz at the surrounding nodes
    xc(j,k) = 0.25*(x(n1) + x(n2) + x(n3) + x(n4))

    compute dxi, deta, area ! compute grid metrics
    compute sx, sy, dz, ds, sn, cn

loop over cells: ! Set values in the ghost cells
loop over faces:
    call findbc(i,j,k,jj,kk,j2,k2) ! get ghost cell indices (jj, kk)
    set sx, sy, dxi, deta in ghost cell equal to values in boundary cell (j,k)

```

8.10 Source pseudocode

```
! select the soil profiles from the r8H, r8Theta, r8K
hdum = r8H(j,k,1:nz)
thetadum = r8THETA(j,k,1:nz)
r8kdum = r8K(j,k,1:nz)

! convert the ponded depth to cm to use with as the Richards equation BC
zold = depth*100.d0 ! input depth in cm
znew = zold ! initialize depth after the infiltration step in cm
PI = 0.d0 ! Initialize the potential infiltration as 0.
iskip = 0 ! Initialize the infiltration indicator (iskip = 1 if Richards solver is
! skipped, in which case r8H, r8Theta and r8K matrices are not updated).

! determine whether the cell is vegetated or not, and define the roughness parameters
! accordingly:
if vegetated cell:
    isveg = 1
    fm,falpha,feta = r8mV, r8alphaV, r8etaV
else: ! bare cell
    isveg = 2
    fm,falpha,feta = r8mB, r8alphaB, r8etaB

! only update depth in the corrector step
if lstep = 1 then

! Before the Richards equation solver is called, several cases need to be handled.
! only solve Richards equation every iscale timesteps
if mod(it, iscale) /= 0 then
    if there is ponding, set the the surface flux / infiltration rate using K from the
    previous timestep
! Case 2: rain and no ponding
else (if prate > 0 and zold = 0) then
    call potential(hdum, thetadum, PI) ! determine the potential infiltration rate PI. Note:
    potential is a subroutine that takes hdum and thetadum as inputs and returns PI as
    the output.
! Handle two cases: PI > prate (no ponding) and PI < prate (ponding starts)
if prate > PI then
    record the time of ponding, tp
    isetflux = 0 ! set the Richards boundary type to fixed H (Dirichlet), with htop=0 at
    the surface.
    call Richards equation with the updated boundary conditions.
else ! no ponding
    isetflux = 1
    flux = - prate*100. ! set the infiltration rate to the rainfall rate
    winflt = (znew - zold)/dt/100.d0
! Case 3: no ponding and no rain
else if (zold = 0 and prate = 0) then
    isetflux = 1 ! set the Richards surface boundary condition to fixed flux
```

```

    flux = 0      ! set flux to zero
    call Richards equation solver to update the soil moisture profile
    winflt = 0
! Case 4: ponding (with or without rain)
    else if zold > 0 then
        isetflux = 0 ! set the Richards surface boundary condition to fixed H
        htop = zold   ! set surface BC to ponded depth
        call Richards equation solver with updated boundary conditions.

        update depth (znew) using the infiltration rate from the updated K
        winflt = (znew - zold)/dt/100.d0
! End of Richards equation solver cases.

    if iskip = 0 ! if Richards equation was called by timestep
        ! update r8THETA, r8H, r8K (3D soil matrices) with 1D solver results
        r8THETA(j,k,1:nz) = thetap1dum
        r8H(j,k,1:nz) = hp1dum
        r8K(j,k,1:nz) = r8kp1dum

        compute r8fluxin (surface flux)

        if ipass = 0 ! if Richards equation was not passed (due to saturation flag)
            compute r8fluxout ! drainage flux
        else ! saturation flag went off
            r8fluxout = r8fluxin ! assume soil flux out = flux in
        compute r8newmass ! change in soil moisture since the previous timestep

        ! update oldTHETA (soil moisture content from the previous timestep)
        oldTHETA(j,k,1:nz) = r8THETA(j,k,1:nz)
    else ! in the predictor step, there is no infiltration or precipitation
        winflt = 0.d0

    if (depth > epsh) then ! depth greater than threshold
        ! compute magnitude
        vmag = dsqrt(udum*udum + vdum*vdum)

        if (feta .eq. 0.5) then ! non-laminar schemes

            fricSx = (falpha/depth**fm)**(2.d0)*udum*vmag
            fricSy = (falpha/depth**fm)**(2.d0)*vdum*vmag

        elseif (feta .eq. 1) then ! special case for laminar

            fricSx = falpha*udum/depth**fm
            fricSy = falpha*vdum/depth**fm

            ! include a catch for high Re cases: use DW ff with f = 0.5
            Rel = vmag*depth/1.e-6

```

```

    if (Rel .gt. 500) then
        ffact = 0.5 ! okay for smooth surfaces (following Kirstetter)
        fricSx = ffact*udum*vmag/8./grav/depth
        fricSy = ffact*vdum*vmag/8./grav/depth
    endif

endif

! update the source terms
qs(1) = winflt

qs(2) = 0.5D0*udum*winflt - grav*depth*fricSx -
&      grav*depth*sx(j,k) ! sx(j,k) = x-dir bed slope
qs(3) = 0.5D0*vdum*winflt - grav*depth*fricSy -
&      grav*depth*sy(j,k) ! sy(j,k) = y-dir bed slope
else

qs(1) = winflt
qs(2) = 0.d0
qs(3) = 0.d0

endif

```

8.11 timestep pseudocode

Pseudocode for subroutine timestep:

```

subroutine timestep(hnp1m,thetan,hnp1mp1,r8thetanp1m,r8knp1mp1)
!
!   Input:
!       hnp1m, thetan (real, kind = 8) - initial
!   Output:
!       hnp1mp1,r8thetanp1m,r8knp1mp1 - h, theta and k at time m+1
!
!   Comments: uses common variables nz, stop_tol, htop as surface h
!             modifies common variable ipass
include 'dry.inc'

declare variables hnp1m(nz), thetan(nz), r8cnp1m(nz), r8knp1m(nz) ... ! input and output
arrays, and arrays used by the Richards eqn solver
istop_flag = 0 ! indicator variable switches to 1 when convergence criteria is met
niter = 0 ! number of iterations
stop_tol = stop_tol0 ! reset stop tolerance to input (in case condition was relaxed)
ipass = 0 ! indicator variable, set to 1 if soil is 'saturated'

do while stop_flag = 0:
    ! r8cnp1m,r8knp1m,r8thetanp1m given hnp1m

```

```

call vanGenuchten(k,hnp1m(k),
    r8cnp1m(k),r8knp1m(k),r8thetanp1m(k), isveg)

Do some linear algebra (see Celia et al. (1990)

! Compute deltam, the increment in iteration for iteration m+1
deltam = matmul(Ainv, R_MPFd)

increment niter (number of iterations)
niter = niter + 1

if niter > 100
    stop_tol = stop_tol*10 !relax stop tolerance
    if stop_tol > 10:
        quit and return error

! Handle saturation cases
compute t2b_theta = theta(top) - theta(bottom) ! the soil moisture difference between the
    top and bottom of the soil profile.

! Test whether the soil has saturated
! After time tsat_min, if there's ponded water at the surface:
! If the soil moisture difference between the surface and bottom is very small, then:
if t > tsat_min and depth > 0
    if t2b_theta < 0.005, then
        hnp1mp1, thetanp1m = hnp1m, thetan ! update h and theta to (n+1, m+1)
        flux = - r8knp1m(nz) ! set flux to K(surface)
        ! adjust hnp1mp1(nz-1) to obtain this flux
        hnp1mp1(nz-1) = hnp1mp1(nz) + dz + flux*dz/r8knp1m(nz-1)

        ipass = 1 ! flag the soil as saturated
        istop_flag = 1 ! exit Richards solver - don't wait for convergence
! Give the soil a chance to unsaturate after the rain, and the water has drained
! Test whether a saturated soil has unsaturated
! After the storm, if there's no ponding
if t > tr and depth = 0
    if t2b_theta < 0.005, then !if the soil moisture difference between the surface and
        bottom is very small, then the soil was flagged as saturated.

        hnp1mp1, thetanp1m = hnp1m, thetan ! update h and theta to (n+1, m+1)

        flux = 0 ! set surface flux to zero

        ! set hnp1mp1(nz-1) to achieve zero flux BC at the surface
        hnp1mp1(nz-1) = hnp1mp1(nz) + dz
        ipass = 0 ! unfreeze the soil
        isetflux = 1 ! set a fixed, zero flux boundary condition

if (ipass eq 0) then ! if the soil is not saturated then...

```

```

if max(deltam) < stop_tol then ! convergence criteria has been met
  istop_flag = 1

  hnp1mp1 = hnp1m + deltam ! update h(n+1,m) to (n+1,m+1)

  ! apply surface boundary conditions
  if (isetflux .eq. 0) then ! fixed H
    hnp1mp1(nz) = htop
  elseif (isetflux .eq. 1) then ! fixed flux
    r8gkt = r8knp1m(nz-1)
    hnp1mp1(nz) = hnp1mp1(nz-1) - dz - flux*dz/r8gkt
  endif
  ! apply free drainage BC at the lower boundary
  hnp1mp1(1) = hnp1mp1(2)

  call vanGenuchten

else ! update h and keep iterating

  hnp1mp1 = hnp1m + deltam ! update h(n+1,m) to (n+1,m+1)
  hnp1m = hnp1mp1 ! update the old h(n+1,m)

  ! apply surface and lower boundary conditions

```
