Frehg User Manual

Zhi Li, PhD
Lawrence Berkeley National Laboratory

01/04/2021

Introduction

Fine Resolution Environmental Hydrodynamic and Groundwater model (Frehg)

Existing capabilities:

- Simulate 2D depth-integrated shallow water flow with wetting/drying and complex topography
- Simulate 3D variably-saturated subsurface flow
- Simulate coupled 2D surface and 3D subsurface flow
- Simulate simple advective-diffusive transport of passive scalars

Pre-requisite

- **laspack** for solving linear systems (http://www.mgnet.org/mgnet/Codes/laspack/html/node2.html)
- **gcc** or other C-compiler
- **mpich** for message passing

Start a simulation

In the frehg directory:

Create an input folder to put model input data (e.g. ex1_input)

Create an output folder for saving model outputs (e.g. ex1_output)

Customize user setting file (named **input** by default), where:

- sim_id = Simulation ID. User can define case-specific settings in the source code by referring to certain sim_id
- finput = Directory of input data (e.g. ex1_input/)
- foutput = Directory of output data (e.g. ex1_output/)

Compile the code by typing **make** in terminal

Execute simulation by typing **./frehg** in terminal

If in parallel mode, execute by typing **mpiexec –np N ./frehg** (N = total number of sub-processes)

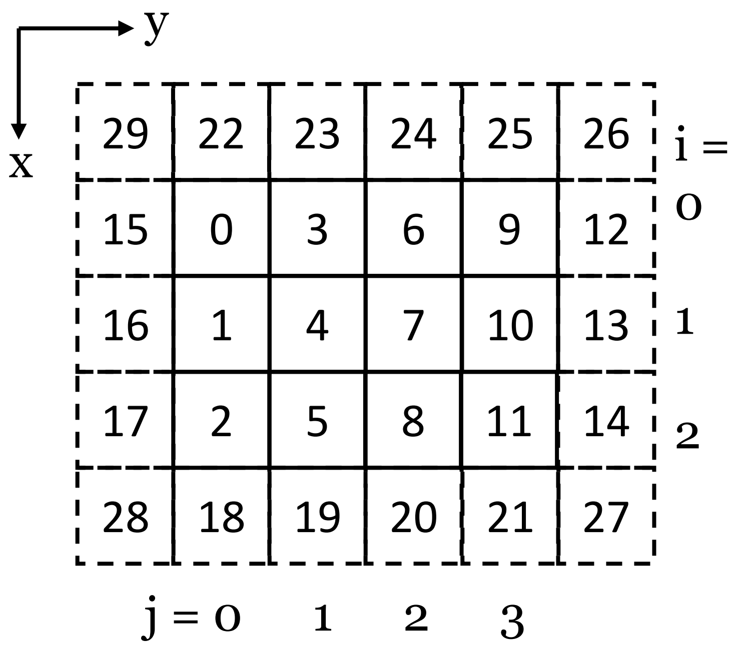
Surface domain

Set sim_shallowwater = 1 to activate surface flow

Set difuwave = 1 to use the diffusive wave approximation

Surface domain is characterized by:

- NX = number of grids in x direction (i-index)
- NY = number of grids in y direction (j-index)
- dx = grid resolution in x direction [m]
- dy = grid resolution in y direction [m]



Use the 2D domain on the right side for illustration:

- NX = 3, NY = 4, ghost cells are added along the boundaries
- Variables are stored in allocated 1D arrays
  - Numbers in the cells represent 1D array indices
  - A map is built for conversion between 1D/2D indices (named **smap** in the source code)
  - The map also stores the connections between grid cells

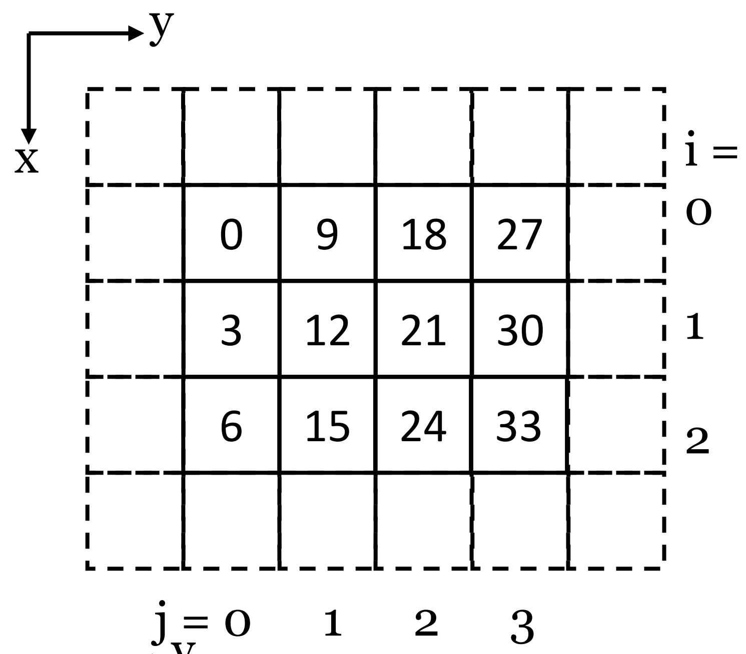
Set bath_file = 1 to read bathymetry from a **bath** file, otherwise bottom elevation is zero all over the domain

Subsurface domain

Set sim_groundwater = 1 to activate subsurface flow

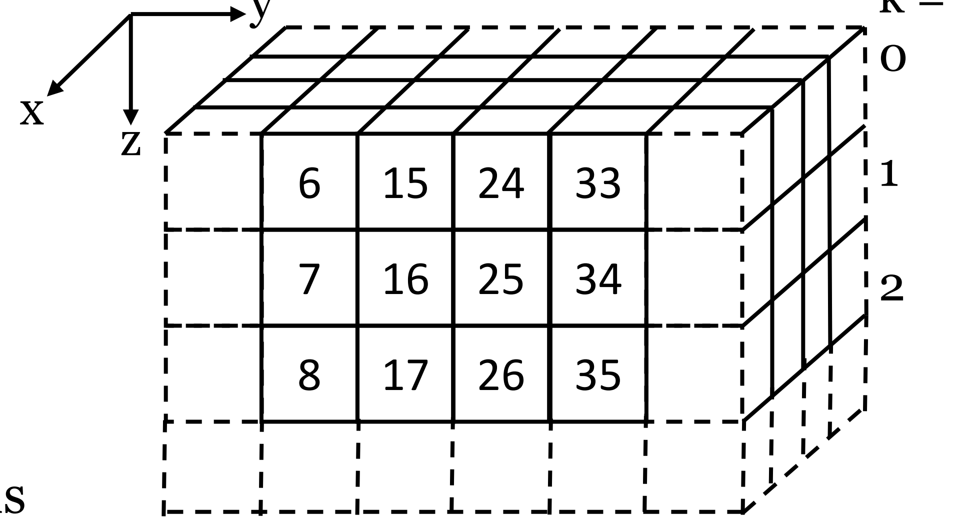
Subsurface domain is characterized by:

- botZ = bottom elevation of the domain [m]
- dz = grid resolution in z direction [m]
- dz_incre = used to create non-uniform z-discretization (not implemented for the current version)



Use the 3D domain on the right side for illustration:

- NX = 3, NY = 4, NZ = 3
- Ghost cells are added (not indexed) along the boundaries
- Variables are stored in allocated 1D arrays
  - Numbers in the cells represent 1D array indices
  - A map is built for conversion between 1D/3D indices (named **gmap** in the source code)
  - The map also stores the connections between grid cells



Simulation control

Parallelization (set use_mpi = 1):

- mpi_nx = number of threads in x direction (must be a factor of NX)
- mpi_ny = number of threads in y direction (must be a factor of NY)

Time control:

- dt = time step size [sec]
- Tend = end time [sec]
- dt_out = output frequency [sec]

If sim_groundwater = 1, variable dt is allowed by setting dt_adjust = 1. Simulation will start from dt and self-adjust between dt_min and dt_max:

- dt_max = maximum dt [sec]
- dt_min = minimum dt [sec]
- Co_max = maximum Courant number for subsurface solver

Then dt is adjusted using the water content + Courant number criteria as described in [3]

Surface domain – IC

Initial conditions for surface flow includes:

- init_eta = initial surface elevation [m]
- init_tide = initial tidal elevation at the tidal boundary [m]. This is used when surface elevation at a boundary is a constant.

Spatially-variable initial conditions can be read from file:

- eta_file = 1 reads file **surf_ic** from finput folder, which contains cell-by-cell initial surface elevation values in the order of the 1D array
- uv_file = 1 reads file **uu_ic** and **vv_ic** from finput folder, which contains cell-by-cell initial velocities in the order of the 1D array
    - If uv_file = 0, initial velocities are zero
- When restarting a simulation, the user should copy simulated surface elevation and velocities from foutput into finput, rename them to **surf_ic**, **uu_ic** and **vv_ic**, then set eta_file and uv_file to 1.

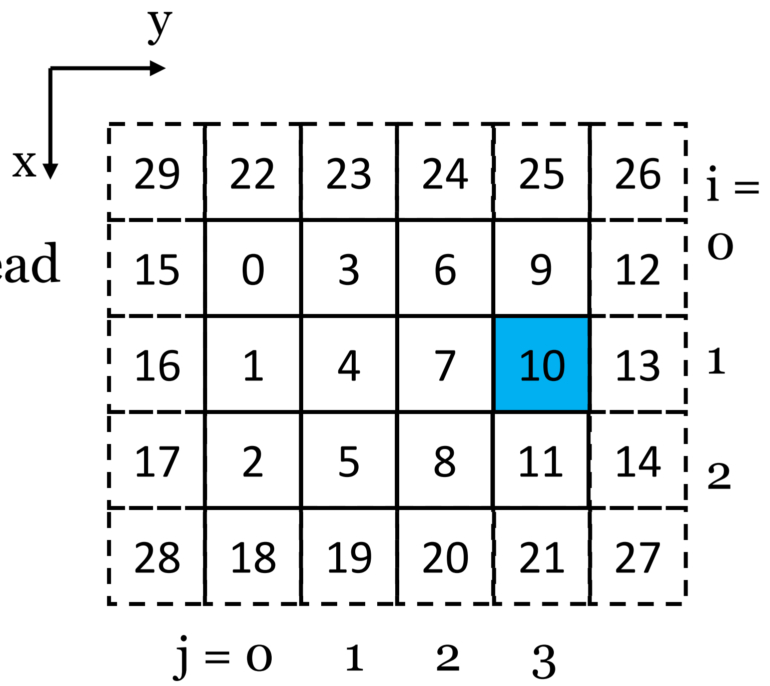
Surface domain – tide

Set tidal boundary condition (or Dirichlet-type BC in general):

- bctype_SW = type of boundary conditions used for x-minus, x-plus, y-minus and y-plus boundaries
  - e.g. bctype_SW = 0,0,0,1 indicates Dirichlet BC for y-plus boundary and no flux BC for other boundaries
- n_tide = total number of tidal boundaries used
- tide_locX = start, end x-indices of each tidal boundary (when simulating in parallel, this is the global indices)
- tide_locY = start, end y-indices of each tidal boundary
- tide_file = 1 reads time-variable tidal elevations from file **tideN** (e.g. **tide1**), otherwise tidal elevation is fixed at init_tide
- tide_dat_len = number of time-variable tide data points to be read

For example, if tidal BC is enforced for cell 10 in the right figure:

- bctype_SW = 0,0,0,1
- tide_locX = 1,1
- tide_locY = 3,3



Surface domain – inflow

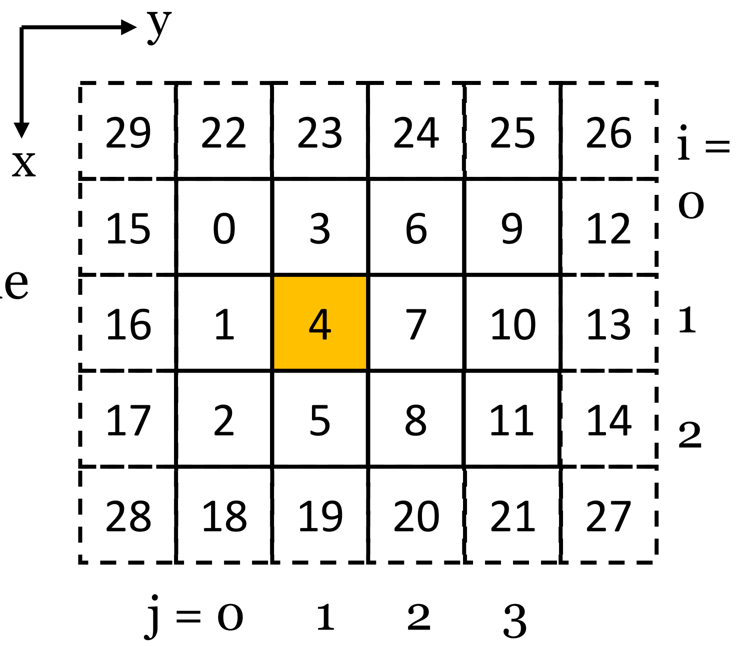
Set inflow boundary condition (or source/sink terms in general):

- inflow_locX/locY = x-start, x-end, y-start, y-end indices where inflow BC is applied, which is similar to how tide location is defined.
- n_inflow = total number of inflow BC to be applied
- inflow_file = 1 if time—variable inflow data is to be read
- init_inflow = constant inflow rate [m^3/s], which is used when inflow_file = 0

For example, if inflow BC is enforced for cell 4 in the right figure:

- inflow_locX = 1,1
- Inflow_locY = 1,1

If inflow_loc contains **N** grid cells, inflow rate for each cell is (assume constant) **init_inflow/N**



Surface domain – rainfall

Set evaporation/rainfall boundary conditions:

- evap_file = 1 if time—variable evaporation data is to be read
- q_evap = constant evaporation rate [m/s], which is used when evap_file = 0
- rain_file = 1 if time—variable rainfall data is to be read
- q_rain = constant rainfall rate [m/s], which is used when rain_file = 0

Evaporation from the subsurface domain:

- evap_model = 1 if aerodynamic evaporation model is applied to estimate moisture-dependent evaporation rate on dry surface. Otherwise evaporation only occurs on wet surface.

In the current version, evaporation/rainfall have to be applied to the entire domain.

Both evaporation and rainfall rates should be positive here.

Surface domain – wind

Set wind stress over wet surface:

- sim_wind = 1 if wind effect is modeled
- wind_file = 1 if time-variable wind speed and direction are read from data files (this function is not implemented yet, so set wind_file = 0 for now)
- init_windspd = wind speed [m/s]
- init_winddir = direction of wind [degrees] measured clockwise from the "north" direction
- north_angle = direction of "north" direction measured clockwise from the –x direction
- Cw = wind drag coefficient
- CwT = decay rate of wind stress within the thin-layer [1]

Surface domain – parameters

Set model parameters for the surface domain:

- grav = gravitational constant [m/s^2]
- viscx, viscy = eddy viscosity [m^2/s] (currently version only supports constant eddy viscosity)
- min_dept = minimum depth below which a grid cell is considered as dry [m]
- manning = Manning's roughness coefficient
- wtfh = minimum depth to trigger the waterfall model [1]. In the current version, the waterfall model is disabled
- hD = depth below which drag coefficient is increased to avoid instability [1]
- rhoa = air density [kg/m^3]
- rhow = water density [kg/m^3]

Subsurface domain – IC

Initial conditions for subsurface flow includes:

- init_wc = initial water content. If init_wc is between saturated and residual water content, the domain will be initialized with constant init_wc
- init_h = initial pressure head [m]. If init_wc is out of range and init_h is not positive, the domain will be initialized with constant init_h
- init_wt_rel = initial water table relative to the topography [m] (must be positive). If neither init_wc nor init_h is used, a water table init_wt_rel m below topography will be set. Regions below the water table is fully saturated.
- init_wt_abs = initial water table [m]. If none of the above 3 settings is invoked, initial water table is located at init_wt_abs, which means a non-terrain-following water table is used.
- h_file = 1 if element-by-element initial head values is to be read from a file named **head_ic**.
- wc_file = 1 if element-by-element initial water content values is to be read from a file named **moisture_ic**.

Subsurface domain – BC

Boundary conditions for subsurface flow is set by:

- bctype_GW = type of boundary conditions used for x-minus, x-plus, y-minus, y-plus, z-minus(bottom), z-plus(top) boundaries, possible values are:
  - 0 = no flow
  - 1 = constant head (Dirichlet type)
  - 2 = constant flux (Neumann type)
  - 3 = free drainage (only used for bottom boundary)
  - When simulating surface-subsurface exchange, the top boundary cells with positive surface depth will automatically switch to Dirichlet condition.
- qtop = flux at top boundary [m/s]
- qbot = flux at bottom boundary [m/s]
- htop = head at top boundary [m]
- hbot = head at bottom boundary [m]

Note that qtop and qbot are positive if upward, negative otherwise. If bctype_GW[5] = 2 and net infiltration exists (evaporation+rainfall is downward), infiltration becomes surface ponding and BC is Dirichlet type. If net exfiltration exists, exfiltration rate is used as qtop.

Subsurface settings

- use_corrector = 1 activates the explicit corrector step [3], otherwise only the head form of the Richards equation is solved, which is non-conservative in the unsaturated zone
- post_allocate = 1 activates the post-allocation step [3]
- use_full3d = 1 activates 3D post-allocation, otherwise post-allocation is only 1D in the vertical direction. For large domain where horizontal mesh size >> vertical mesh size, set use_full3d = 0 is sufficient.
- use_mvg = 1 activates the modified Mualem-van Genutchen model [3], which allows larger dt than the original van Genutchen model
    - aev = air entry value when using the MVG model [m]

Subsurface parameters

- Ksx, Ksy, Ksz = saturated hydraulic conductivities in x, y, z directions [m/s]
- Ss = specific storage [1/m]
- soil_a = alpha parameter in the van Genutchen model
- soil_n = n parameter in the van Genutchen model
- wcs = saturated water content
- wcr = residual water content

Scalar transport

The current version of frehg only simulates advective-diffusive transport of passive scalars.

- n_scalar = total number of scalars to be modeled
- difux/difuy/difuz = molecular diffusivities of the scalars.
- disp_lon/disp_lat = longitudinal/transverse dispersivity in the subsurface domain
- scalar_surf_file = 1 if initial surface scalar concentration is to be read from a file named **scalar_surf_icN**, where N represents the index of the scalar.
- init_s_surf = initial scalar concentration for the surface domain, which is used if scalar_surf_file = 0
- scalar_subs_file = 1 if initial subsurface scalar concentration is to be read from a file named **scalar_subs_icN**, where N represents the index of the scalar.
- init_s_subs = initial scalar concentration for the subsurface domain, which is used if scalar_subs_file = 0

Scalar transport

The current version of frehg only simulates advective-diffusive transport of passive scalars.

- scalar_tide_file = 1 if time-variable scalar concentration for tide is to be read from a file named **scalarN_tideM**, where N is the index of scalar and M is the index of the tide.
- scalar_tide_datlen = number of time-variable scalar data points to be read from the **scalarN_tideM** file.
- s_tide = constant scalar concentration for tide, which is used when scalar_tide_file = 0
- scalar_inflow_file = 1 if time-variable scalar concentration for inflow is to be read from a file named **scalarN_inflowM**, where N is the index of scalar and M is the index of the tide.
- scalar_inflow_datlen = number of time-variable scalar data points to be read from the **scalarN_inflowM** file.
- s_inflow = constant scalar concentration for tide, which is used when scalar_inflow_file = 0

References

[1] **Li, Z.** and Hodges, B.R., 2019, Model instability and channel connectivity for 2D coastal marsh simulations, *Environ Fluid Mech, (19): 1309*, https://doi.org/10.1007/s10652-018-9623-7

[2] **Li, Z**. and Hodges, B.R., 2019, Modeling subgrid-scale topographic effects on shallow marsh hydrodynamics and salinity transport, Advances in Water Resources, (129) 1-15, https://doi.org/10.1016/j.advwatres.2019.05.004

[3] **Li, Z.,** Ozgen, I. and Maina, F.Z., 2020, A mass-conservative predictor-corrector solution to the 1D Richards equation with adaptive time control, Journal of Hydrology, (592)125809, https://doi.org/10.1016/j.jhydrol.2020.125809