Manual of the SubFREHD-C Configurations (version 3.3)

Zhi Li

2018-02-05

1 Grid settings

- dx: Grid size in x direction, which corresponds to index "ii" in the solver code.
- dy: Grid size in y direction, which corresponds to index "jj" in the solver code.
- NX: Dimension of computation domain in x direction in [m].
- NY: Dimension of computation domain in y direction in [m].
- nx: Dimension of computation domain in x direction for the current rank. Since only 1D subdomain partitioning was performed, nx = NX.
- ny: Dimension of computation domain in y direction for the current rank. ny = NY / np, where np is the total number of ranks used for parallelism.
- N2CI: Total number of internal grid cells, N2CI = NX*NY.
- N2CT: Total number of grid cells including ghost cells, N2CT = (NX+2)*(NY+2).
- N2ci: Total number of internal grid cells for one rank, N2ci = nx*ny.
- N2ct: Total number of grid cells including ghost cells for one rank, N2ct = (nx+2)*(ny+2).

2 Operation settings

- dt: Length of time step in [s].
- Nt: Total number of steps to be modeled.
- OutItvl: Date output interval in [steps].
- tStart: Start time of simulation in "YYYY-MM-DD".
- tNStart: Start time in the form of C date number.
- tNEnd: End time in the form of C date number, tNEnd = tNStart + dt*Nt.
- saveFolder: Directory where output data will be saved.
- inputFolder: Directory where input data will be read.
- savesurface: Whether or not save free surface elevation as output files.
- saveuu: Whether or not save the velocity in x direction as output files.
- savevv: Whether or not save the velocity in y direction as output files.
- savedepth: Whether or not save the cell depth as output files.
- savescalar: Whether or not save the scalar concentration as output files.
- savesub: Whether or not save the subgrid variables.
- saveCD: Whether or not save bottom drag coefficients.
- useCellEdge: Use the elevated cell edge model, which requires input file "edgeX.dat" and "edgeY.dat".
- isRestart: Select 1 to make a restart run.
- ttRestart: The time step at which model restarts.
- restartFile: Directory where restart file is saved.

3 Physical properties

- g: Gravitational constant, g = 9.8066.

- NUx: Eddy viscosity in x direction, [m^2/s].
- NUy: Eddy viscosity in y direction, [m^2/s].
- CDnotN: Whether or not using constant bottom drag coefficient. CDnotN = 1 means yes. CDnotN = 0 means using depth dependent drag coefficient.
- CDx: Bottom drag coefficient in x direction. This parameter and the next one are only effective when CDnotN = 1.
- CDy: Bottom drag coefficient in y direction.
- manningN: Manning's n for the bottom. This parameter is only effective when CDnotN = 0.
- z0: Thickness of the bottom buffer layer in [m].
- rhoa: density of air in [kg/m^3].
- Cw: Wind drag coefficient.

4 Boundary conditions

- bcType: Define the types of boundary conditions. Currently only 3 types are defined. bcType = 1 is tidal-inflow BC with tide added on the positive y boundary. bcType = 2 is tidal-inflow BC with tide added on the negative y boundary. bcType = 3 is tidal-tidal BC.
- useWind: Whether or not to activate the wind force model.
- northAngle: The angle of the real north direction clockwise from the modeled north (the negative x direction). This parameter is only effective when useWind = 1.
- tideNP: The number of time-value pairs to be read from the tidal data file for the positive y boundary. This parameter is only effective when tide on positive y boundary exists. Similar condition applies for the next four parameters.
- tideNM: The number of time-value pairs to be read from the tidal data file for the negative y boundary.
- inflowN: The number of time-value pairs to be read from the inflow data file.
- windspdN: The number of time-value pairs to be read from the wind data file for wind speed.
- winddirN: The number of time-value pairs to be read from the wind data file for wind direction.
- tideLocLengthP: The number of grid cells along the positive y boundary where tidal data will be added.
- tideLocLengthM: The number of grid cells along the negative y boundary where tidal data will be added.
- inflowLocLength: The number of grid cells where inflow data will be added.
- tideLocP: The 1D map indices for tidal boundary locations on positive y boundary.
- tideLocM: The 1D map indices for tidal boundary locations on negative y boundary.
- inflowLoc: The 1D map indices for inflow locations.
- useEvap: Whether or not to activate the evaporation model (it is not implemented in the current version).
- evapN: The number of time-value pairs to be read from the evaporation data file.
- useRain: Whether or not to activate the rain model (it is not implemented in the current version).
- rainN: The number of time-value pairs to be read from the rain data file.

5 Initial conditions

- initU: Initial velocity in x direction, [m/s].
- initV: Initial velocity in y direction, [m/s].
- initSurf: Initial surface elevation in [m]. These initial values are all uniformly distributed for the entire domain.

6 Scalar settings

- useScalar: Whether or not activate the scalar model.
- useConstInitS: Whether or not use uniform initial condition for scalar. If not, the initial scalar concentration will be read from an input file.
- useConstInflowS: Whether or not use constant inflow scalar concentration.
- useConstTidePS: Whether or not use constant scalar concentration on the positive y boundary.
- useConstTideMS: Whether or not use constant scalar concentration on the negative y boundary.
- tidalPSN: The number of time-value pairs to be read from the positive y tidal boundary scalar file. This option is only effective when useConstTidePS = 0.
- tidalMSN: The number of time-value pairs to be read from the negative y tidal boundary scalar file. This option is only effective when useConstTideMS = 0.
- initS: The initial scalar concentration. This option is only effective when useConstInitS = 1.
- tidePS: The scalar at the positive y tidal boundary. This option is only effective when useConstTidePS = 1.
- tideMS: The scalar at the negative y tidal boundary. This option is only effective when useConstTideMS = 1.
- inflowS: The scalar for inflow. This option is only effective when useConstInflowS = 1.
- Kx: Eddy diffusivity in x direction, [m^2/s].
- Ky: Eddy diffusivity in y direction, [m^2/s].

7 Subgrid settings

- useSubgrid: Whether or not use the subgrid area and volume model.
- useSubDrag: Whether or not use the subgrid drag coefficient model. Set to 0 for no drag model. Set to 1 for subgrid curvature model. Set to 2 for the drag model developed by [Volp 2013].
- useCorrector: Whether or not use the predictor-corrector model for updating subgrid variables.
- subgridFolder: The directory where the subgrid variables are saved.
- dxf: Size of the fine resolution grid in x direction [m].
- dyf: Size of the fine resolution grid in y direction [m].
- dA: Area of a fine resolution grid in [m^2].
- subR: Grid coarsening ratio of the subgrid model.
- surfmax: Maximum possible surface elevation stored in the subgrid variable look-up table, [m].
- surfmin: Minimum possible surface elevation stored in the subgrid variable look-up table, [m].
- dsurf: Interval between two possible surface elevations in [m].

8 Parallel settings

- useMPI: Whether or not use message passing.
- np: Total number of ranks used. If useMPI = 0, then must set np = 1.

9 Other settings

- minDepth: Minimum allowable cell depth, below which the cell is assessed to be dry, [m].
- eps: Residual to stop the iterative matrix solver.
- maxIter: Maximum number of iterations of the iterative matrix solver.
- useThinLayer: Whether or not use the buffer layer drag model for shallow water.
- CDmax: Maximum bottom drag coefficient for the buffer layer model.
- CwT: Decay rate of wind stress inside the buffer layer.
- hD: Depth of buffer layer in [m], hD = z0.
- wtfh: Minimum depth below which the waterfall model is prohibited in [m].
- CFLl: Lower limit of the CFL number below which CFL limiter is not applied on nonlinear term.
- CFLh: Upper limit of the CFL number above which the nonlinear term is forced to zero.