iRIC and Nays2D manual for modelling shallow flow in physical scale experiments

Steven A.H. Weisscher*[1], Marcio Boechat Albernaz[1], Jasper R.F.W. Leuven[1], Wout M. Van Dijk[1], Yasuyuki Shimizu[2], and Maarten G. Kleinhans[1]

[1]Faculty of Geosciences, Utrecht University, Princetonlaan 8A, 3584 CB, Utrecht, The Netherlands
[2]Faculty of Engineering, Hokkaido University, North 13, West 8, Kitaku, Sapporo, Hokkaido, 080-8628, Japan

February 13, 2020

This appendix describes how to install and use the iRIC user interface to run the hydrodynamics in Nays2D over DEMs of fluvial and tidal flume experiments. The DEMs used in this study are appended as CSV files and may be used to remodel the findings of this study.

Installation iRIC

The installer .exe file for the iRIC user interface can be downloaded from https://i-ric.org/en/download/ under "Version3.X". To enable downloading the files, the user needs to sign up for free on this website. Run the .exe file once it has been downloaded. During installation, the folder *iRIC 2.3(x64)* is created in *Program Files*. After installation, go the subfolder *solvers*. Here you will see the default set of solvers that are provided in iRIC. See https://i-ric.org/en/solvers/ for a more in-depth explanation on the default solvers. The default solvers can be used in the iRIC user interface. For flume experiments with unidirectional flow, you can use the solver nays2dh_v1 or its predecessor nays2d if available.

For simulating tilting flume experiments, paste the entire solver folder *nays2d_Metronome003* in the folder *solvers*. In order for this new solver to work, an update of the redistributables is required. This can be found via the link: https://software.intel.com. We used the Intel Parallel Studio XE 2015 Update 7 for Fortran on Windows. After installation the following error message should not be given when you try to run the tilting flume solver (Figure 1).

Figure 1: Likely error message upon running the solver nays2d_Metronome003 for tilting flume experiments in the iRIC user interface. This is solved by updating the redistributables, as explained in the text. This issue should not arise for the default solvers, which includes the nays2d(h) solver for rivers.

*Corresponding author
Email address: s.a.h.weisscher@uu.nl
Tel. 0031 30 253 7641

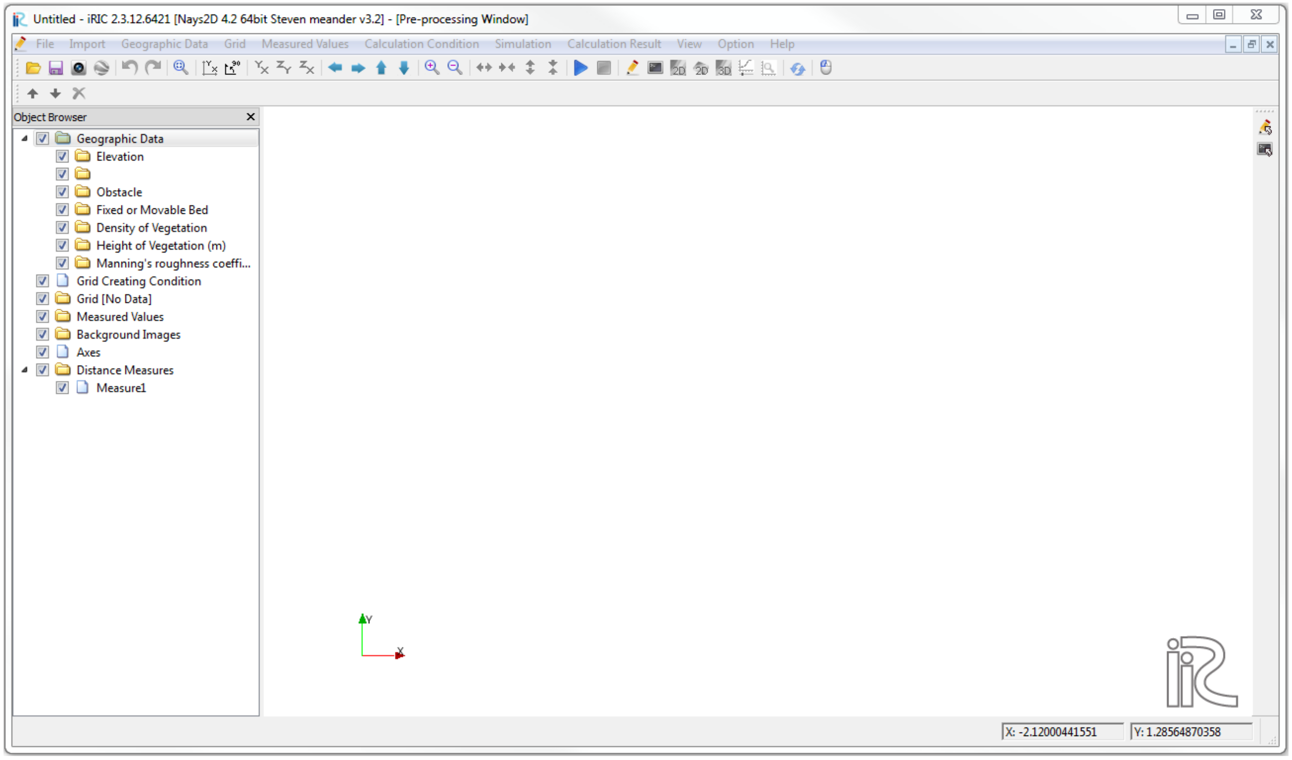

Figure 2: iRIC user interface.

How to use iRIC

After installation, open iRIC. This takes you to the iRIC user interface (Figure 2) in which all computations will take place. Below, a step-by-step plan is given on how to model hydrodynamics over a DEM of a physical scale experiment.

Import a DEM

DEMs are imported via the tab Grid → import. Move to the correct folder and import the CSV file that contains the DEM.

The CSV file requires a header of three lines, followed by the data. An example is given below:

IMAX,JMAX,KMAX
200,20,1
I,J,K,X,Y,Z
0,0,0,-0.01744,2.9346,0.071
1,0,0,-0.0072018,2.9346,0.071
2,0,0,0.0030368,2.9346,0.071
...
...

where IMAX and JMAX are the number of nodes in the x and y direction, and KMAX is the number of layers. In this case, the grid is 200 by 20 nodes large and only contains 1 layer (elevation), so KMAX equals 1. The third line is the header for the actual data. I, J, and K are the nodes in the x y and z direction, given in integers, starting at zero; since only 1 layer is loaded, K equals zero. X, Y and Z are then the actual coordinates in decimals.

After loading the DEM, the folder *Grid* in the *Object Browser* now contains the elevation data. Go to *Grid*, tick and open *Node attributes* and tick *Elevation* to check if your elevation data was loaded correctly; if it isn't, the data in the CSV is structured incorrectly.

Next, assign the Manning roughness coefficient. In the *Object Browser*, go to *Grid*, tick and open *Cell attributes* and tick *Manning's roughness coefficient*. Next, select all cells in the grid and assign a roughness coefficient. In default, go for a spatially constant roughness of 0.02 s.m$^{1/6}$.

Calculation conditions

Go to the tab Calculation Condition and select Setting.... A new window appears in which all parameters that may be changed are given, in various Groups. For in-depth information on all parameters, as well as Groups with a '+' (Figure 3), please consult the iRIC user Manual that is downloadable from https://i-ric.org/en/download/. Parameter settings are explained for unidirectional flow and exceptions for reversing flow are explained in bold.

1. Solver Type

The first Group of parameters is Solver Type (Figure 3). In case of modelling only the hydrodynamics, set the parameters to the values as shown in Figure 3. This way, morphological change is not calculated, because bed deformation, bank erosion and slope collapse are disabled. The turbulence is set to the simplest equation, as explained in the main paper, but may be changed to a constant eddy viscosity or to a K-epsilon equation. See the iRIC User Manual that is downloadable from https://i-ric.org/en/download/ for more information on the computation of turbulence.

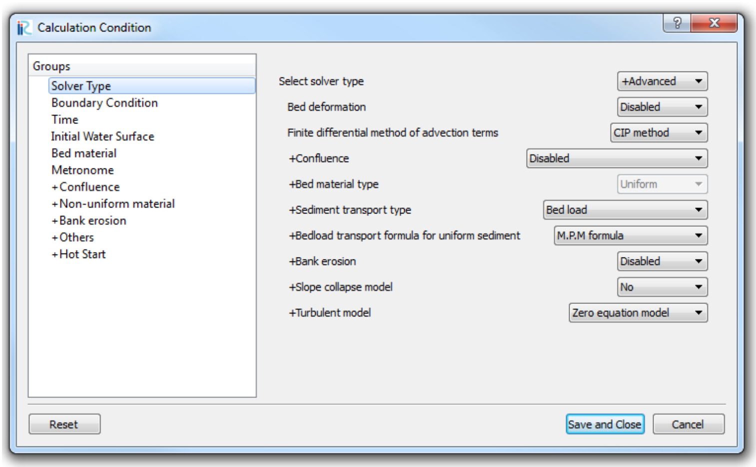

Figure 3: Calculation conditions concerning solver type and whether or not to include morphological change.

2. Boundary Condition

The second Group of parameters is Boundary Condition (Figure 4). This set of parameters describes the inflow and outflow conditions of the model. The periodic boundary condition is disabled for physical scale experiments. The water surface at the downstream end of the flume is determined from a uniform flow condition, a fixed value or is given as a user-defined time-series. Velocity at the upstream boundary may be computed from a uniform flow condition or is calculated from the upstream depth. Finally, the time-series of inflowing discharge must be given, even when it is constant. Click Edit and fill in a discharge curve. The column Water level is ignored when all values are 0.

For reversing flow: the water surface at the downstream end must be set at a constant value, and flow velocity at the upstream end of the flume must be calculated from the upstream water depth.

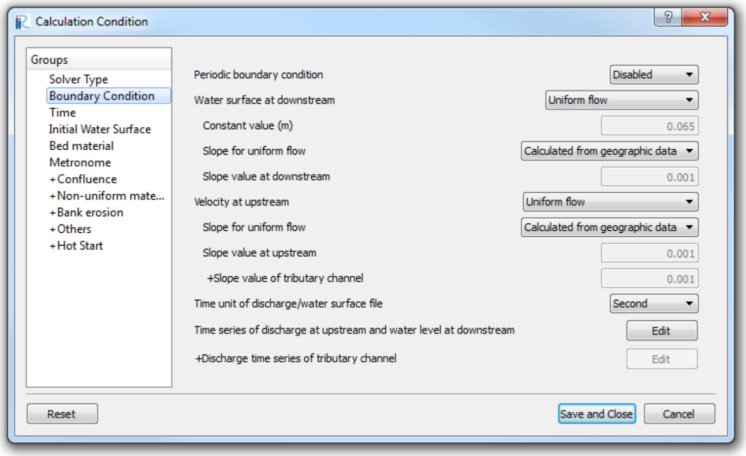

Figure 4: Calculation conditions concerning the boundary conditions at the upstream and downstream ends of the flume.

3. Time

The third Group of parameters is Time (Figure 5). It asks for the output time interval, the calculation time step and start time of output. For physical scale experiments that require a long time to stabilise, in particular setups with reversing flows, it is advised to set the time of output (much) larger than zero so as to limit file size. This group also contains stability parameters concerning water level surface calculations.

4. Initial Water Surface

The fourth Group of parameters is Initial Water Surface (Figure 6). Initial water surface can be set to a constant slope, with the slope being equal the valley slope of the flume setup. It is important that the water at the upstream and downstream boundaries are connected. Other options for an

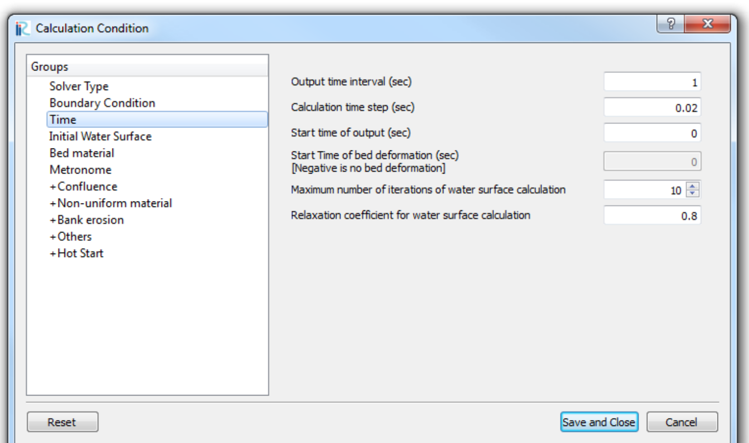

Figure 5: Calculation conditions concerning time step and when output files should be generated.

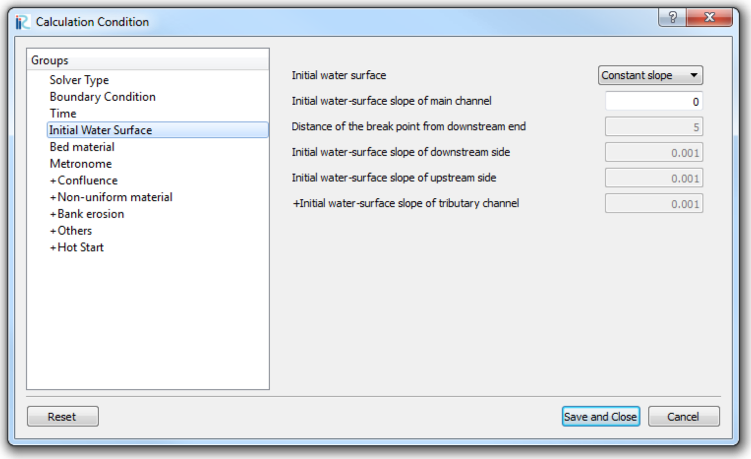

Figure 6: Calculation conditions concerning the initial water surface level.

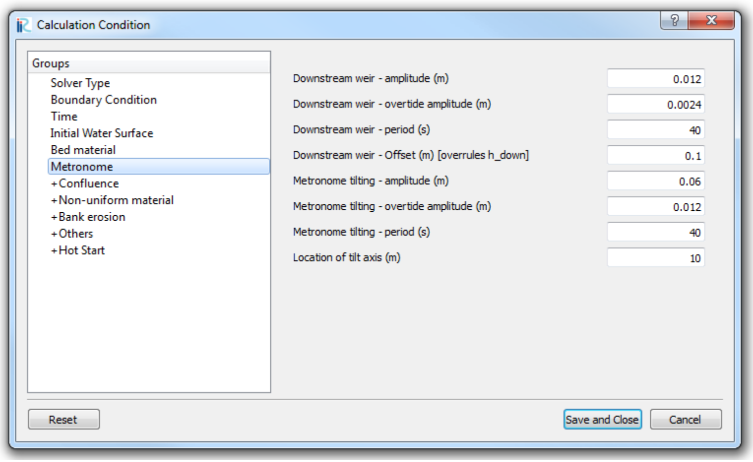

Figure 7: Calculation conditions for sinusoidal tilt of the DEM so as to mimic the tilting flume the Metronome to steer reversing flow.

initial water surface include: a user-defined line, based on a uniform flow condition and based on a nonuniform flow condition.

5. Bed material

This Group contains one parameter which is grain size.

6. Metronome

The sixth Group of parameters is Metronome (Figure 7) and contains all parameters necessary for sinusoidal tilting a DEM over a central tilting axis. For both the flume and the downstream weir, the period, amplitude and its overtide are needed. Overtides in this model have a fixed period that is two times smaller than the user-defined period and have a phase difference of 90°. The location of the tilt axis should be in the middle of the flume (in case of the Metronome, this is 10 m).

7. +Others

The last Group of parameters discussed in this manual is +Others (Figure 8). The most important parameters in this group are those of the eddy viscosity coefficient. For unidirectional flow, the user can define the values for a and b in

$$\mathrm{v_t} = \frac{\kappa}{6}\mathrm{a u_* h + b} \tag{1}$$

where κ is the von Karman constant (-), u_* is the shear velocity (m/s), h is the water depth (-), and a and b are user-defined constants set to the default values of 1 and 0, respectively. u_* is given as

$$\mathrm{u_*} = \sqrt{\theta \mathrm{d_{50}} \frac{\rho_s - \rho}{\rho} \mathrm{g}} \tag{2}$$

in which θ is the nondimensional bed shear stress (-), d_{50} is the median grain size (m), ρ_s and ρ are the density of sand and water (kg/m^3), and g is the gravitational acceleration (m/s^2).

For reversing flow: the parameter b is split into the default value (0) for shallow regions and a user-defined value (in this study: 0.02) for deep regions. The deep regions correspond to the sea in the estuary experiments and require a higher value for numerical stability. The boundary between shallow and deep is requested by the model as the water depth threshold (b_snu_thres) and is set to 0.09 in this study.

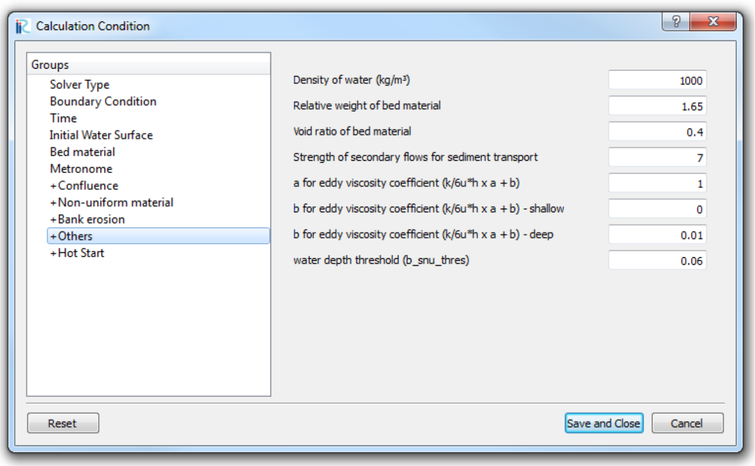

Figure 8: Calculation conditions for viscosity parameter settings, secondary flow and sediment characteristics.

Run the model

After all parameters have been set, go to the tab Simulation and hit Run. This will open a new window in which progress of the computation is tracked. For every output interval, a new line is added; the first column is the time in seconds, the second column is the inflow discharge in m^3/s and the third column is the water level at the downstream boundary in m. If the model is also generating output (as explained under 'Time'), 'out' is added to the row.

To visualise results, even while the model is running, click on the coloured palette with 2D on the top row of symbols. A new window will open and all output parameters are available under the folder *Scalar*. Simply tick the field that needs to be shown on the screen. You may change between windows using the panel at the right-hand side. For further explanation on visualisation, the user is kindly requested to consult the iRIC User manual available on https://i-ric.org/en/download/.