The ComMIT Interface

Tsunami models (both deep-water propagation and inundation) require information on: (1) bottom topography; (2) initial and boundary conditions; and, (3) model run specific information such as time-step and length of model run. The aim of ComMIT is to provide an interface which allows for the selection of model input data (initial condition, bathymetry grids, etc.) as well as a platform to display model output through a graphical user interface. ComMIT has been written in the Java programming language (http://www.java.com) and uses the netCDF file format for model input and output thus making ComMIT platform-independent (i.e. it may be run on different platforms such as WINDOWS, Mac or UNIX). ComMIT may be used with any computational model with the only requirement that the models are able to input and output data in the specified format (netCDF).

The interface consists of a "Start Model" button, Progress Bar, and Model Run selector at the top, and 5 tabs: "Model Setup", "Initial Condition", "Grid Bathymetry", "Results Animation", and "Results Extrema" (Figure 1).

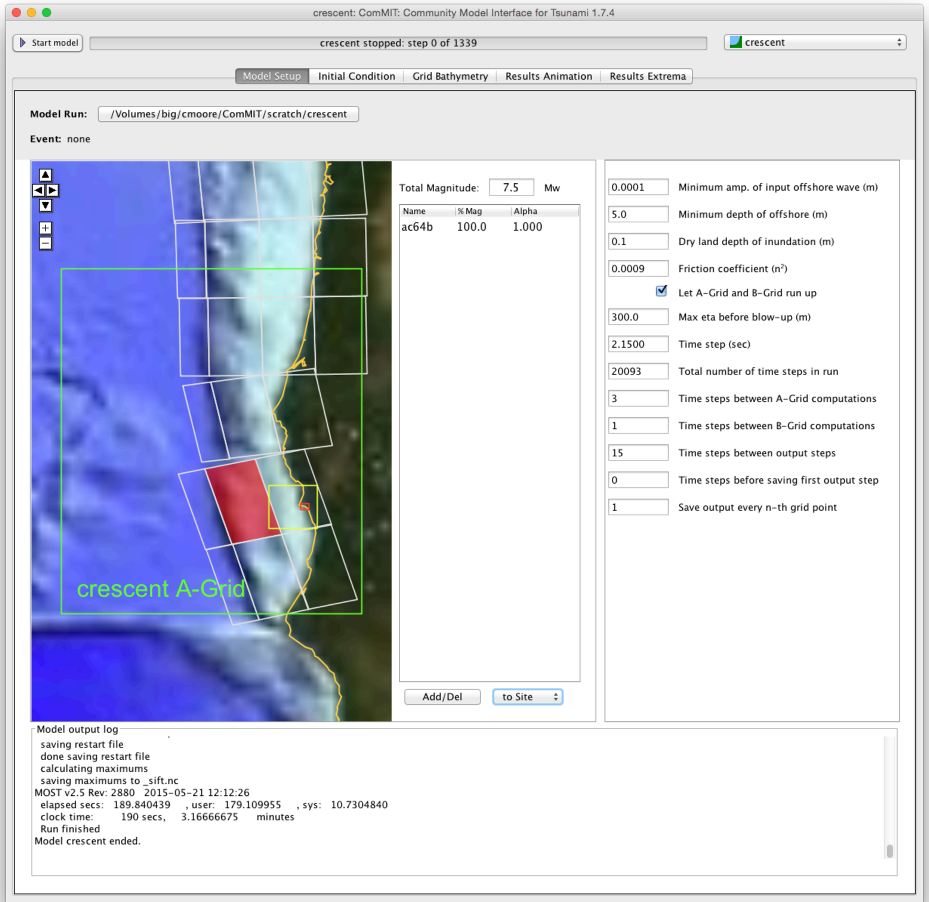

Figure 1: ComMIT Interface

Model Setup tab

The first tab, "Model Setup", shows the location of the Model Run that is currently open ("crescent" in this case). The Model Run consists of three nested grids: an outer, low resolution A-grid (in green), a middle, medium resolution B-grid (in yellow) and an inner, high-resolution C-grid (in red). The Model Run consists of these three bathymetry files, and a file containing the Model Parameters, all in a folder that can be opened by clicking on the button with the Model Run name just above the map.

Unit Source Map

The map also shows white rectangles defining a seismic fault plane. These fault planes, know as Unit Sources, can be used to force the Model Run. The user can select one or more of these Sources, and set their "weight" by adjusting the Alpha coefficient in the table to the right of the map(Figure 2):

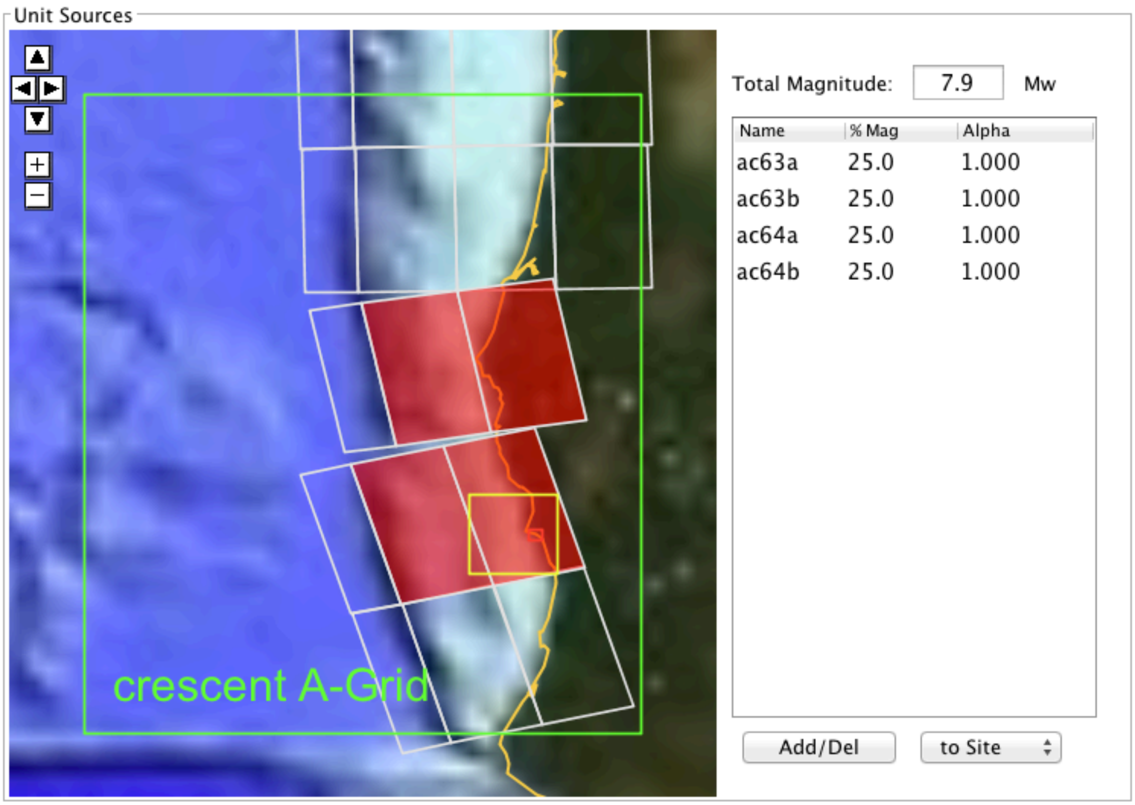

Figure 2: Four Unit Sources selected, each with an Alpha coefficient of 1.0, total moment magnitude Mw=7.9

The total moment magnitude (Mw) can be adjusted by changing the Alpha coefficient, by adding more Unit Sources, or by editing the magnitude directly.

The user can pan the map by clicking and dragging the map, and can zoom by mouse wheel or using the +/- buttons. The drop-down list under the table can be used to jump directly to a region on the map and is helpful to jump between distance sources ("to Selected") and back to the Model Run location ("to Site"). The Add/Del button opens a dialog that allows selecting Unit Sources by their name. Unit Source names include a two-letter code defining their subduction zone (e.g. "ac" for Aleutian-Cascadia subduction zone), and a letter indicating their position on- or off-shore, and a number indicating their position along the fault. The "ki1a" fault plane is the first fault plane at along the Kamchatkai-Yap subduction zone.

Set Model Run Parameters

The MOST model parameters for the run are set manually in the upper-right frame of the Model Setup tab (Figure 3). The following is brief description of the model parameters. Details are in the MOST manual.

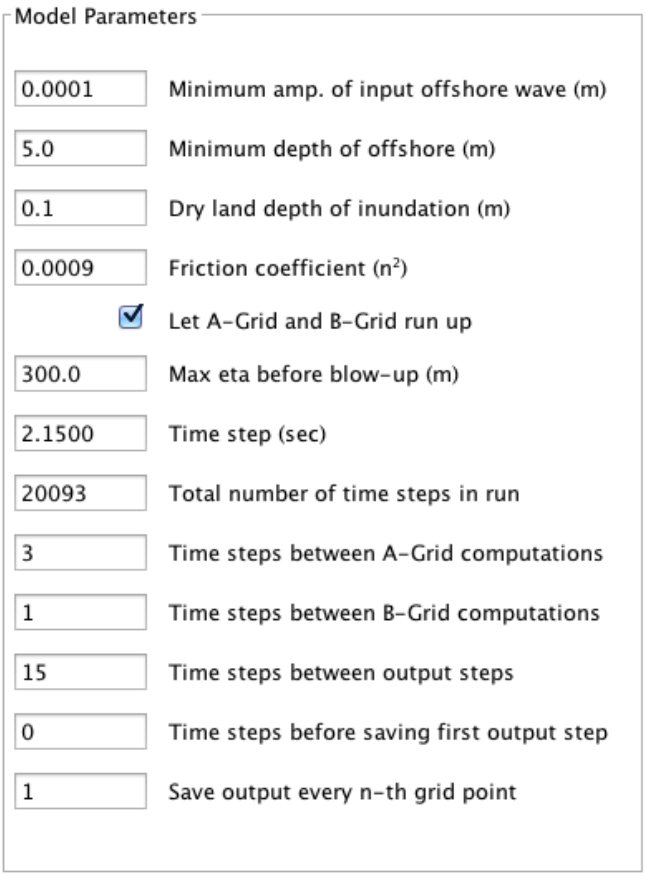

Figure 3: MOST Model parameters

Minimum amplitude of input offshore wave *(m)* – smaller values of input amplitudes are ignored. Input through grid A boundary starts only when absolute amplitudes exceed this threshold. This is usually a small number well above machine zero (0.001 is typical value)

Minimum depth of offshore *(m)* – specifies depth for grid A and B where reflection boundary conditions will be performed. Typical values are 5 – 10 m.

Dry land depth of inundation *(m)* – specifies the minimum depth of the water column that model performs computations on. If the water level is lower than this value, the cell is considered dry and no computation is performed. This effectively sets a moving boundary condition threshold. Typical values are 0.1-0.3 m.

Friction coefficient *(n**2)* – specifies Manning friction coefficient (n^2)

Let A-grid and B-grid runup – if checked, the runup computation is performed for grid A and B (with the dry land parameter specified in *Minimum depth of offshore*). Otherwise, fixed reflective boundary is set at the same depth. Default is "checked".

Max eta before blow-up *(m)* – computation is terminated if computed amplitude is exceeded this value. 30-100m is recommended.
Time step (sec) – specifies time step for grid C computation. Grid C is computed every time step.

Total number of time steps in run – number of time steps

Time steps between A-grid computations – A-grid wave dynamics may be computed less frequently than the C-grid. Since A-grid is coarser than C, the CFL condition may be less demanding and a larger time step can be used for stable computations. Number of skipped steps multiplied by the time step defines the effective time step for A-grid.

Time steps between B-grid computations – same as above for B-grid

Time steps between output steps – specifies the frequency of amplitude and velocity field saves

Time steps before saving first output step – output can saved from specified time step to avoid saving time steps with zero amplitudes

Save output every n-th grid point – specifies the size of saved files by sub-sampling the output fields. The parameter is the same for all grids