

Community Model Interface for Tsunami (ComMIT)

Getting Started

If you're running ComMIT for the first time, you may need a bit of advice getting started... or perhaps you're a know-it-all and are just ITCHING to get started. For you, I just say "File->Create new site", and have a great time!

For the rest of you, just a little help:

1. Put the ComMIT.jar file wherever you'd like to Install ComMIT (the Desktop is fine). Double-clicking the ComMIT.jar will launch ComMIT. ComMIT requires Java version 1.5 or higher. If you need to upgrade your Java Installation, go to <http://java.sun.com/javase/downloads> and look for the Java Runtime Environment.
2. A subdirectory named "ComMIT" will be created wherever the ComMIT.jar file was put. This directory will contain all the files ComMIT installs.
3. The first time it loads takes a minute downloading test grids (it'll run faster next time).
4. The Edit->Preferences lets you choose where you work, where your grids are, where your source files are, and where your MOST executable is.
5. If you don't have a MOST executable, you will be asked if you'd like to download one. ComMIT will attempt to choose the proper executable for your platform. An Internet connection is required.
6. Uninstalling is just a matter of deleting the "ComMIT" subdirectory tree created in the directory you installed the ComMIT jar file.
7. If you don't have any Initial Condition (source) files installed, an Internet connection is required. These large files are usually kept on a server (see <http://sift.pmel.noaa.gov/data/d5>, for example). ComMIT accesses subsets and linear combinations of these files to create a proper Initial- and Boundary-condition for the model.

As a quick test of the system, choose File->Create new site, and choose hiloGridA, hiloGridB, hiloGridC, since this grid is optimized to run quickly. Name it anything (Hilo), and click "Launch". You should see an initial condition file created, and an Initial Condition image. The model results can be animated as the model runs by clicking the "MOST Results – Wave" tab.

A more thorough discussion of the Community Model Interface follows.

User Manual

Interface overview

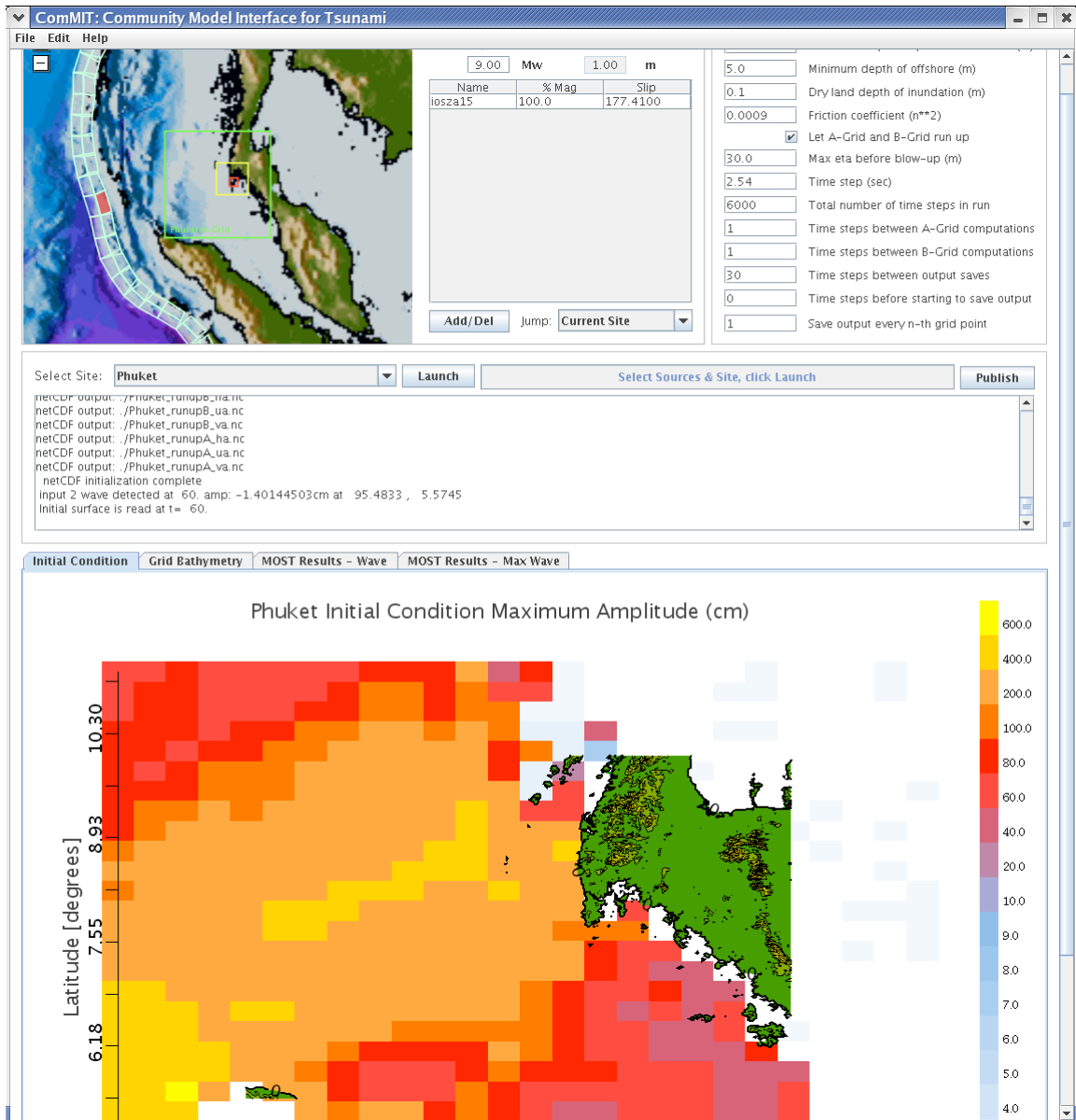
Tsunami models (both deepwater propagation and inundation) require information on: (1) bottom topography; (2) initial and boundary conditions; and, (3) model run specific information such as time-step and length of model run. The aim of ComMIT is to provide an interface which allows for the selection of model input data (initial condition, bathymetry grids etc) as well as a platform to display model output through a graphical user interface (GUI). ComMIT has been written in the Java programming language (<http://java.sun.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 MS WINDOWS, MAC OS or UNIX). ComMIT may be used with any computational model (or a combination of models) with the only requirement that the models are able to input and output data in the specified format (NetCDF).

The interface window consists of three main frames:

Upper frame is the interactive area for choosing initial conditions and parameters for the model run. The left panel is graphical geographic location and source information window; the right panel is text input of model parameters.

Middle frame is text output window showing the model progress

Lower frame is the graphical output window showing graphical representation of model input and results.



Menu

Menu provides ability to add/delete new model sites (or new model runs for the same site) and set up preferences for the interface.

File -> Create new site

Provides a dialog to input the files with bathymetric grids that specify the particular geographic site for inundation modeling.

Input the path to files with preformatted Digital Elevation Model (DEM) data (gridded bathymetry/topography data in flat files). See details of file formats and requirements in MOST manual.

File -> Delete site

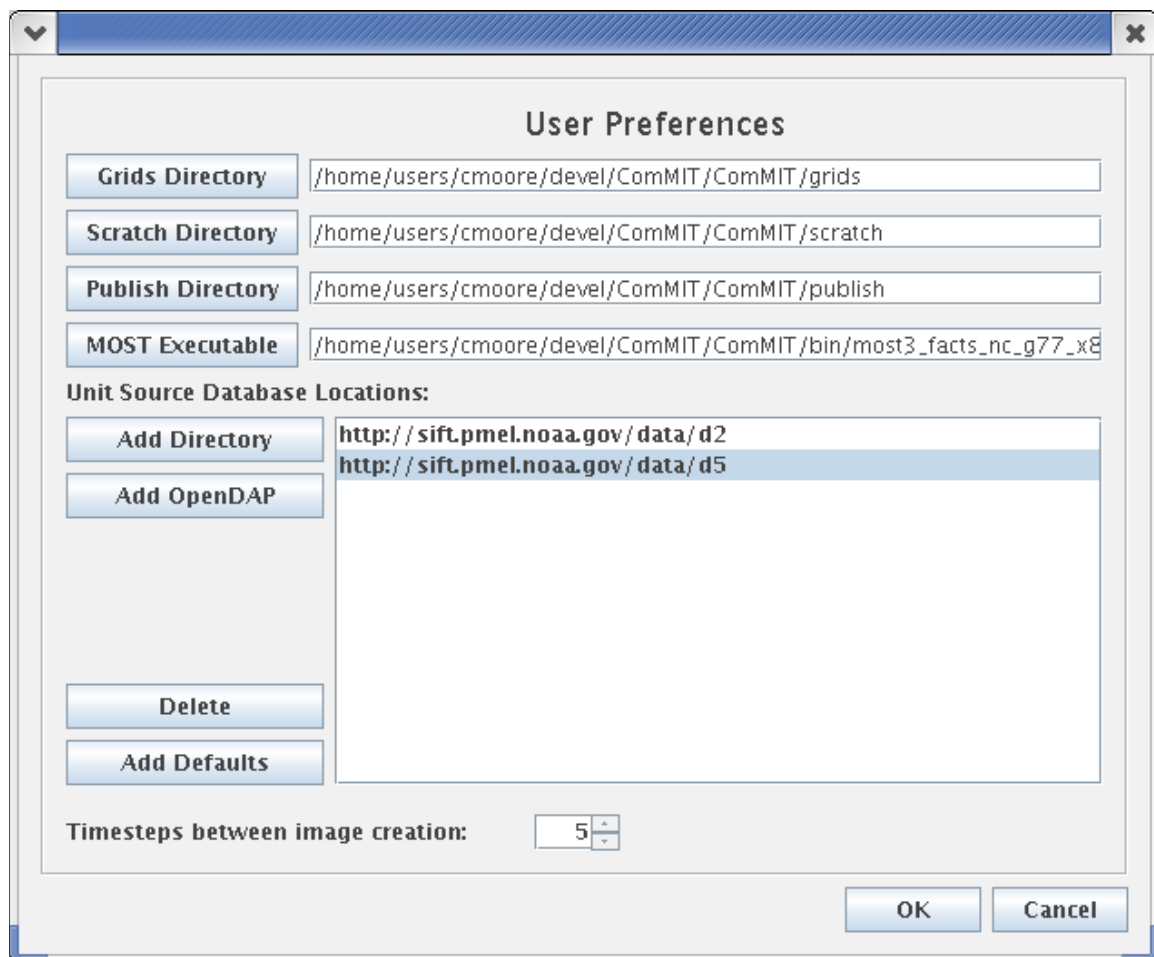
Removes selected sites. This removes all the files associated with the selected model run, including input and output data.

File -> Exit

Exits the program. All results for sites that have not been deleted will be saved and available in the next session.

Edit -> Preferences

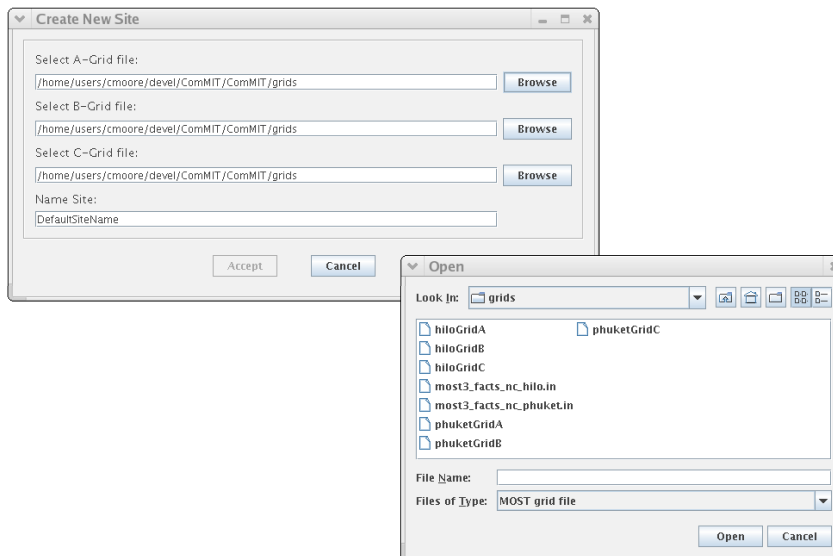
Allows the User to change the directories ComMIT uses for grid locations (Grids Directory), model output (Scratch Directory), published model output (Publish Directory). It also allows the User to set the MOST executable, and to add/delete Initial Condition (Unit Source) files.



Set bathymetry files

Bathymetry for the inundation model runs is provided in three separate ascii files. The files should contain DEMs of three telescoping grids focusing on the location of interest; grid A is spatially largest DEM with coarsest resolution, grid B is the intermediate DEM, grid C is the finest resolution DEM with smallest coverage. The consecutive grids should be completely inside each other. The details of required resolution and file format constrains are described in the MOST manual.

The files are specified in the *File -> Create New Site* dialog:



When correctly formatted DEMs are accepted, the text output window prints the maximum time step that can be used for each grid. The time step in the model parameter window cannot exceed this value, since it will lead to an unstable model run. The geographic location window will show the location of the accepted grids. The plots of the accepted DEMs are available at the *Grid Bathymetry* tab of the graphics output frame.

Set model run parameters

Model parameters for the run are set manually in upper-right frame of the interface. The following is brief description of the model parameters. Details are in the MOST manual.

<input type="text" value="0.0010"/>	Minimum amp. of input offshore wave (m)
<input type="text" value="5.0"/>	Minimum depth of offshore (m)
<input type="text" value="0.1"/>	Dry land depth of inundation (m)
<input type="text" value="0.0009"/>	Friction coefficient (n^{**2})
<input checked="" type="checkbox"/>	Let A-Grid and B-Grid run up
<input type="text" value="30.0"/>	Max eta before blow-up (m)
<input type="text" value="2.54"/>	Time step (sec)
<input type="text" value="6000"/>	Total number of time steps in run
<input type="text" value="1"/>	Time steps between A-Grid computations
<input type="text" value="1"/>	Time steps between B-Grid computations
<input type="text" value="30"/>	Time steps between output saves
<input type="text" value="0"/>	Time steps before starting to save output
<input type="text" value="1"/>	Save output every n-th grid point

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 runoff – if checked, the runoff 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. Runup enable is recommended.

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

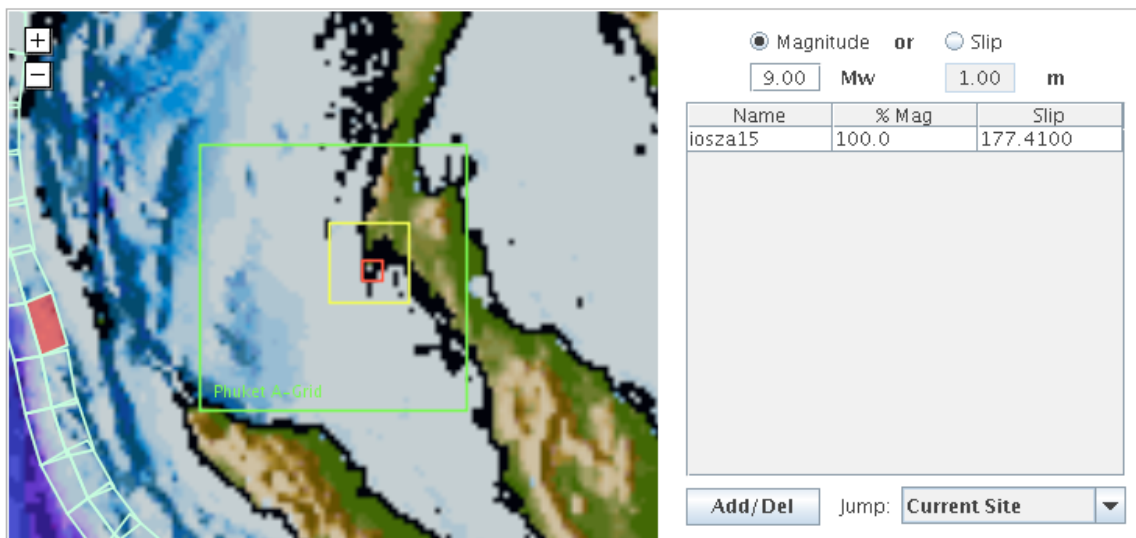
Time steps between output saves – specifies the frequency of amplitude and velocity field saves

Time steps before starting to save output – output can be 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

Set up tsunami initial conditions

The initial conditions for the run are specified from the previously computed tsunami propagation runs. ComMIT provides access to pre-computed propagation model databases from NOAA and BMRC. The user can specify additional precomputed runs or reference to additional database of runs from the *Edit -> Preferences* menu. The user can choose a single propagation run or a linear combination of several propagation runs that specifies a larger combination source.



● Magnitude or ○ Slip

9.00 Mw 1.00 m

Name	% Mag	Slip
iosza15	100.0	177.4100

Add/Del Jump: Current Site ▼

The interactive geographic information window is used to choose (by mouse click) the source fault planes that correspond to associated propagation run. If several faults are selected, a linear combination of those faults is assumed. The magnitude of the earthquake can be adjusted manually in the source specification frame. If adjusted, the whole propagation solution is scaled accordingly. Either magnitude or slip can be specified. For a combination source, the slip or percentage of magnitude can be specified for individual sub-faults.

Run MOST model

When all input data and parameters are specified, the inundation model for the selected site is run by clicking *Launch* button.

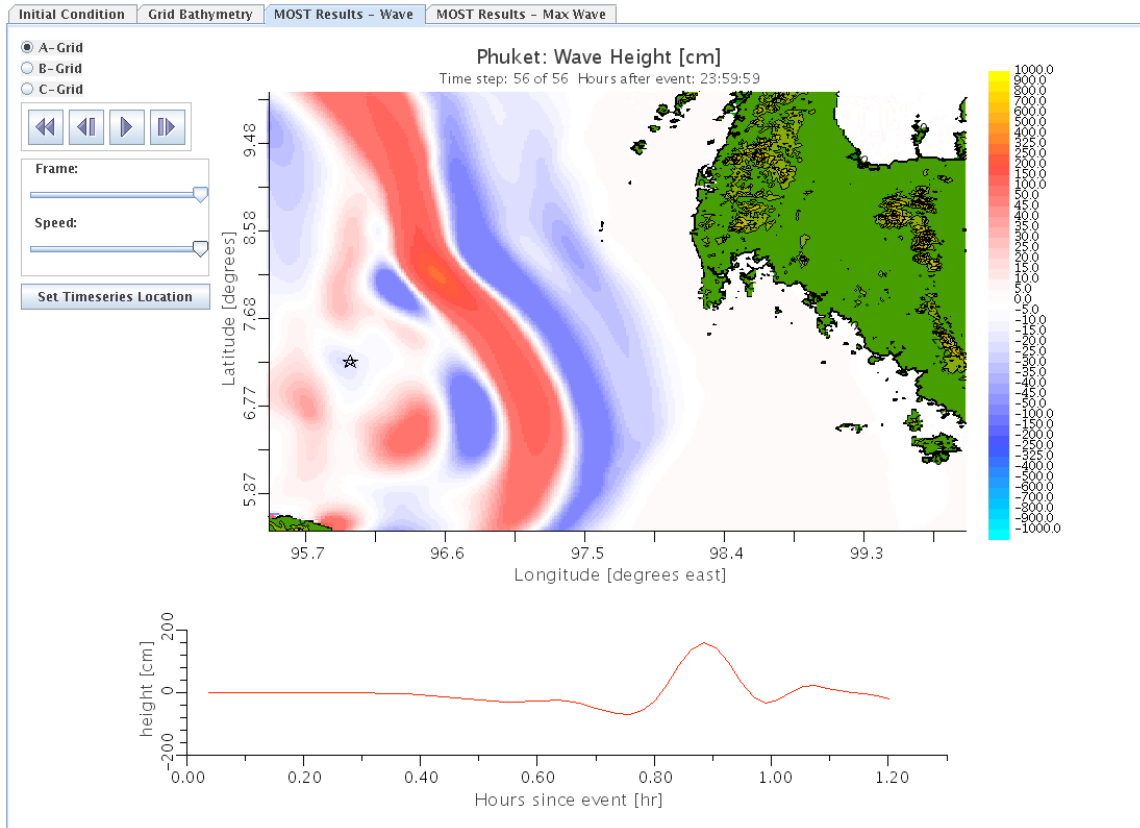
The progress of the inundation run is monitored by the progress bar, the text output window and the graphics output window.

The progress bar shows the time steps computed.

Text output window shows the standard text output of the model (see MOST manual for details).



Graphics output window show amplitude plots for saved time frames. The output is saved as consecutive images and can be viewed as animation at *MOST Results – Wave* tab. The time series of a specific location is also plotted. The location of the time series data can be chosen from a graphical window by clicking *Set Timeseries Location* button.



View results

When the model run is finished the maximum computed inundation is plotted and can be viewed at *MOST Results – Max Wave* tab. All results are saved in scratch directory (specified in the *Edit -> Preferences* menu). The results are standard NetCDF files and can be viewed with many other 3rd party software tools.

Share results with the community

The results of a model run can be shared with the community by publishing on the Internet. Clicking *Publish* button will put the results in the Internet-enabled directory (specified in the *Edit -> Preferences* menu).