

ComMIT Manual

Community Model Interface for Tsunami

Brought to you by the NOAA Center for Tsunami Research
(<http://nctr.pmel.noaa.gov>)

Table of Contents

Intro	2
Installation	2
For PC	2
For Mac	2
For linux	2
Installation basics	2
Quick Test	3
The ComMIT Interface	4
Model Setup tab	5
Unit Source Map	5
Set Model Run Parameters.....	7
Initial Condition tab	9
Grid Bathymetry tab	10
Results Animation tab	10
Results Extrema tab	13
ComMIT Menu items	14
The Model Menu item	14
Create Composite Wave File	15
Tsunami Events and Tweb	16
Custom Propagation Run.....	19
Using Custom Sources	21
The Edit Menu item	22
The View Menu item	22
Creating New Model Runs	24
Global bathymetry data	24
Using the New Model Run Wizard	24
More Information	29

Intro

ComMIT is an interface around NOAA's Method Of Splitting Tsunamis (MOST) tsunami model. ComMIT is designed for ease of use, and allows the dissemination of results to the community while addressing concerns associated with proprietary issues of bathymetry and topography. It uses initial conditions from a pre-computed propagation database, has an easy-to-interpret graphical interface, and requires only portable hardware. ComMIT was initially developed for Indian Ocean countries with support from the United Nations Educational, Scientific, and Cultural Organization (UNESCO), the United States Agency for International Development (USAID), and the National Oceanic and Atmospheric Administration (NOAA).

As of 2015, more than 325 scientists from 57 countries around the world have been trained and are using it in operational inundation mapping.

Installation

For PC

Put the ComMIT_<ver>.exe file wherever you'd like (the Desktop is fine). Double-clicking the ComMIT_<ver>.exe will launch ComMIT.

For Mac

Download the ComMIT_<ver>.dmg file and double-click to launch the ComMIT installer. It may complain that ComMIT is an unsigned app: in this case go to System Preferences, open Security and Privacy and Allow apps downloaded from Anywhere. Drag the ComMIT app to the /Applications folder. Close the installer. Double-click ComMIT from the Applications folder.

For linux

Put the ComMIT_<ver>.jar file wherever is convenient (the Desktop is fine). Double-clicking the ComMIT.jar will launch ComMIT. The user can see debug info by running on the command-line with "java -jar ComMIT_<ver>.jar -d".

Installation basics

ComMIT requires Java, and will tell you which version it needs. If you need to upgrade your Java Installation, go to <http://www.java.com>.

A folder named "ComMIT" will be created in your home folder. This directory will contain all the files ComMIT installs. The first time it loads takes a minute downloading test grids into this folder. If you wish to move this folder, ComMIT will ask you to locate it the next time ComMIT is run.

The ComMIT->Preferences lets you choose where you work, where your grids are, where your source files are, and where your MOST executable is.

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.

Quick Test

As a quick test of the system, click the Menu "Model->Open Model Run", and choose "Crescent". The map on the "Model Setup" tab should show "crescent A-Grid" in green, outlining Crescent City, California. Click any white rectangle "source" on this map – it should turn red. Click "Start Model" at the top. That's it!

You should see the Model output log indicate the source is downloading, and the red rectangle eventually turn green. You can see an initial condition file created, and see it in the Initial Condition tab. The progress bar at the top of ComMIT should start to indicate that the model is running, and time steps should update. The model results can be animated as the model runs by clicking the "Results Animation" tab.

A more thorough discussion of the Community Model Interface follows.

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).

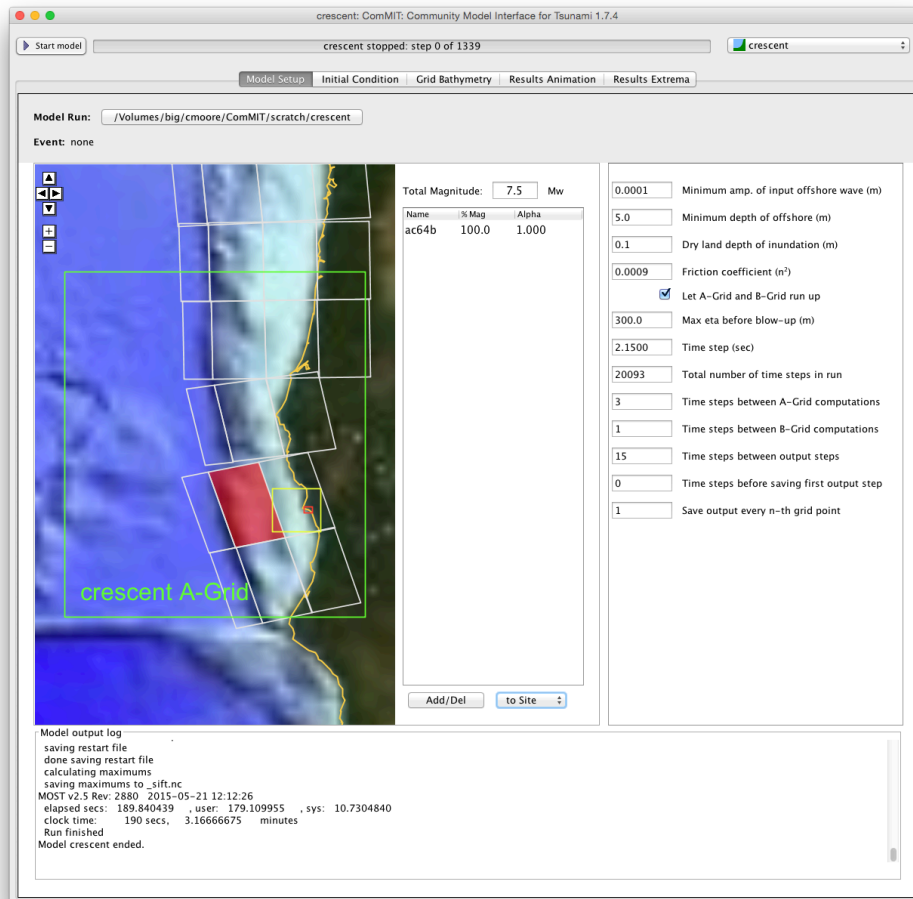


Figure 1: ComMIT Interface

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):

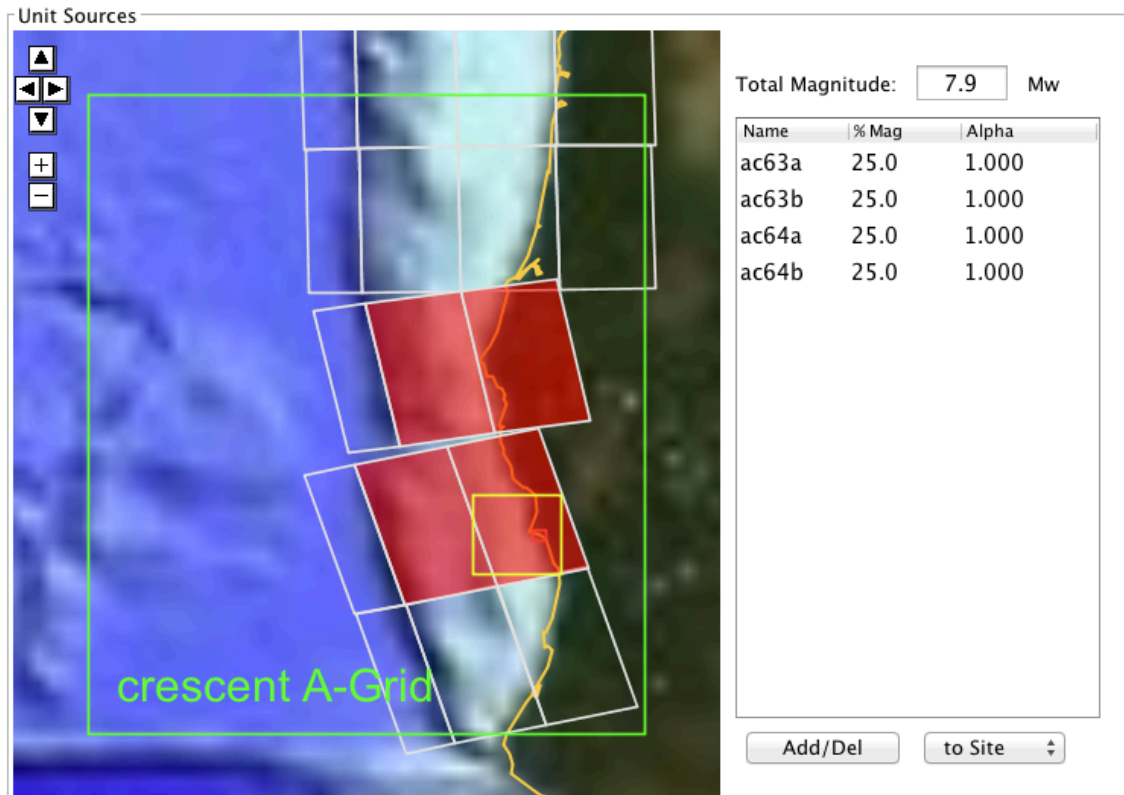


Figure 2: Four Unit Sources selected, each with an Alpha coefficient of 1.0, total moment magnitude Mw=7.9

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.

Model Parameters	
<input type="text" value="0.0001"/>	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="300.0"/>	Max eta before blow-up (m)
<input type="text" value="2.1500"/>	Time step (sec)
<input type="text" value="20093"/>	Total number of time steps in run
<input type="text" value="3"/>	Time steps between A-Grid computations
<input type="text" value="1"/>	Time steps between B-Grid computations
<input type="text" value="15"/>	Time steps between output steps
<input type="text" value="0"/>	Time steps before saving first output step
<input type="text" value="1"/>	Save output every n-th grid point

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

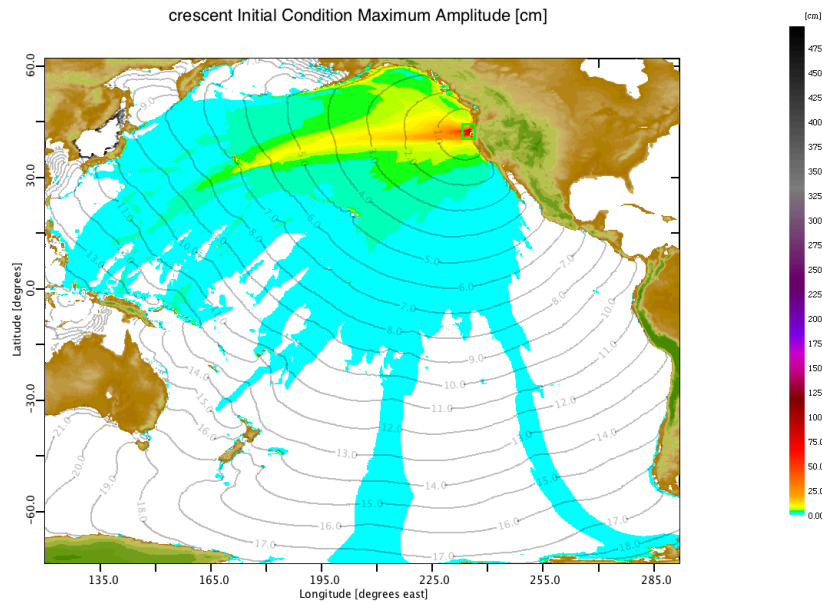
Time steps between output steps – specifies the frequency of amplitude and velocity field saves

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

Initial Condition tab

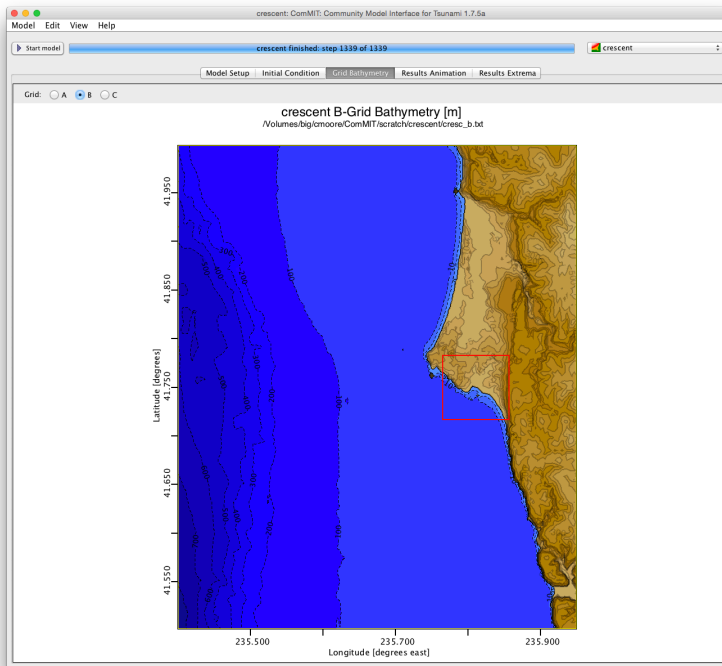
After loading a Model Run, and selecting Unit Sources to force the model, the user can initiate a run by pressing the “Start model” button. The selected Unit Sources are downloaded from the ComMITServer and combined according to the Alpha coefficients selected in the Unit Source table. This combination of sources can be viewed in the Initial Condition tab, which shows the maximum wave amplitude and arrival time (in hours) for the source selected:



The color represents the time-maximum wave amplitude, and shows how the energy of the tsunami tends to move away from the subduction zone in “fingers” or “lobes” extending perpendicular to the coastline.

Grid Bathymetry tab


The Grid Bathymetry tab shows a color contour plot of the grid bathy, along with an outline of the telescoping grids with the same color code for the grids: green for A-grid, yellow for B-grid, and red for C-grid:



The bathymetry grid file name is displayed just beneath the title of the grid. Opening the Model Run folder (from the Model Setup tab) will display this file. The user can zoom this image, and select among the three grids using the “Grid:” buttons.

Results Animation tab

The Results Animation tab shows the full time-dependent results for the model output, including wave amplitude, velocity and time series (Figure 4). The results of any of the grids can be viewed by selecting with grid with the Grid: A-B-C button selector on the left. The main panel shows a color contour plot of Wave

Amplitude over dashed contours of bottom bathymetry, and topography in shades of tan and brown. The tab is split vertically, and the lower window contains a time series. The time series location is shown on the upper Wave Amplitude plot by a green star, and the user can change its location by double-clicking on the Wave Amplitude plot. The latitude and longitude of the time series location star is shown at the top, and editing these values is another way of moving the time series location. The results can be animated by using the arrow buttons:  The center button is “Play”, the button just to the left and right step one time step back and forward, respectively. To jump all the way to the beginning of the model run, click the double-left arrow, and to jump all the way to the end, click the double-right arrow. Clicking on the time series plot jumps to that time step in the animation.

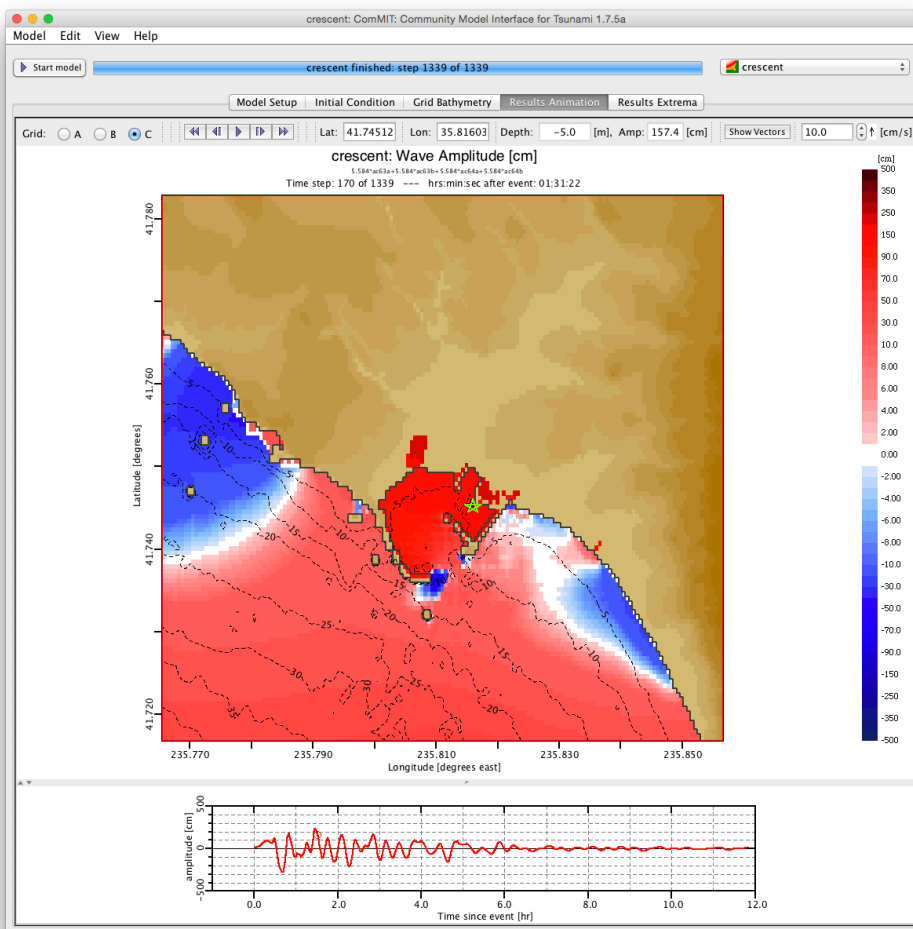


Figure 4: The Results Animation tab shows the full time-dependant model output.

The depth and wave amplitude at the time series location are displayed at the top, and is a good way to determine wave amplitude at a given location and time step.

Velocity vectors can be seen by clicking on the Show Vectors button. The length of the vector is proportional to the velocity at the vector's position, and the scale is shown at the top next to the "Show Vectors" button. This can be scaled up and down to best show the current velocities in the plot. Pressing the animation button allows the vectors to animate along with the wave amplitude field.

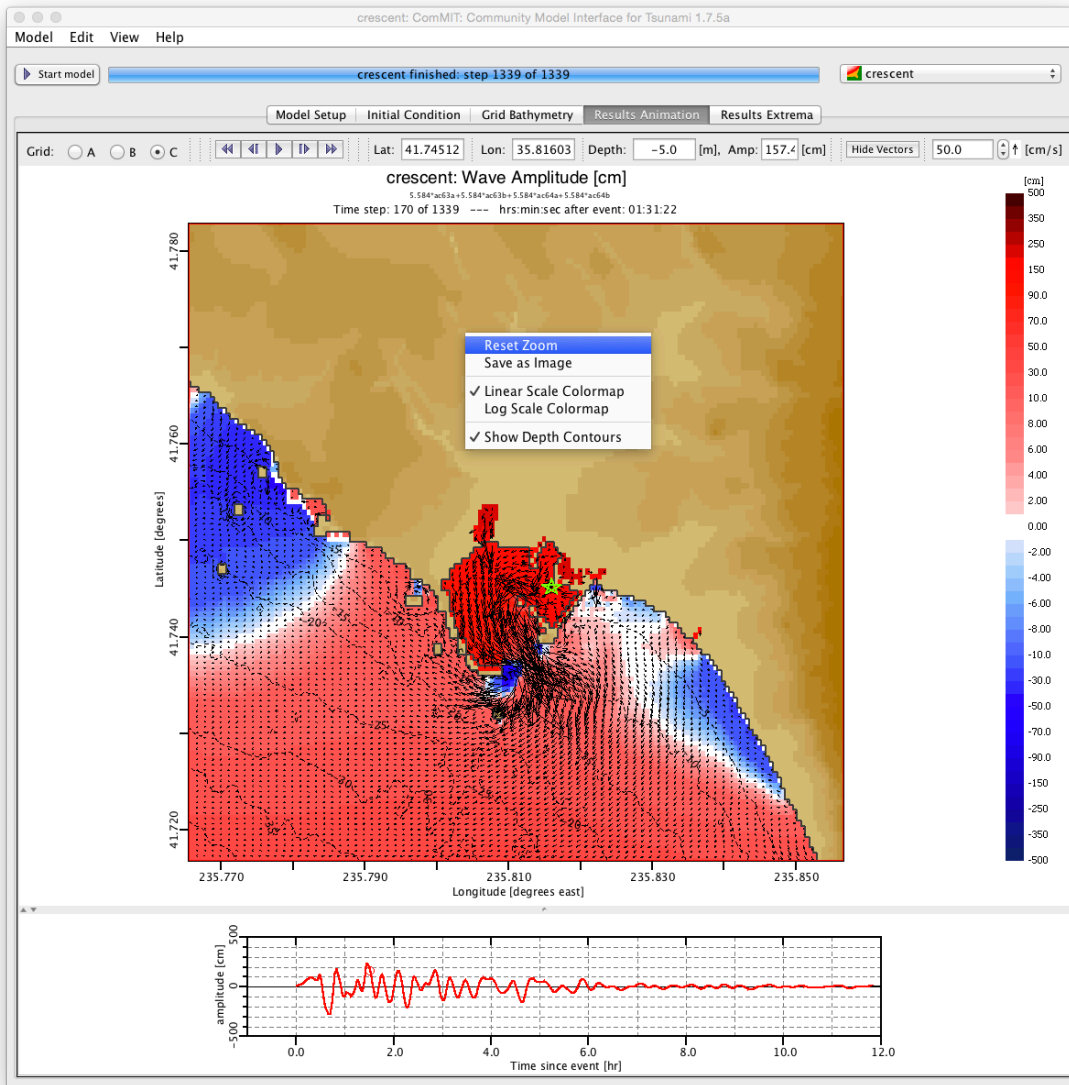


Figure 5: Wave Amplitude with velocity vectors

Zooming the Wave Amplitude plot can be done by clicking and dragging in the plot, and right-clicking gives a menu that allows the user to Reset Zoom (zoom all the way out). The user can also select a pseudo-log scale color map (useful for very small amplitude tsunami simulations), and the plot can be saved as an image.

Results Extrema tab

The Results Extrema tab shows the model results as either the Maximum Wave Amplitude over time, the Flow Depth, the Minimum Wave Amplitude, or the Maximum Wave Speed (Figure 6). The user can choose the preferred view by selecting “Max Amp”, “Flow Depth”, “Min Amp” or “Max Speed”, respectively. By moving the mouse over the plot, the Longitude, Latitude, Depth, and Amplitude are output at the top, and the Show Vectors shows the velocity vectors as in the Results Animation tab (previous section). Note that the velocity vectors represent the speed of the maximum current and the direction of the current at the time of its maximum.

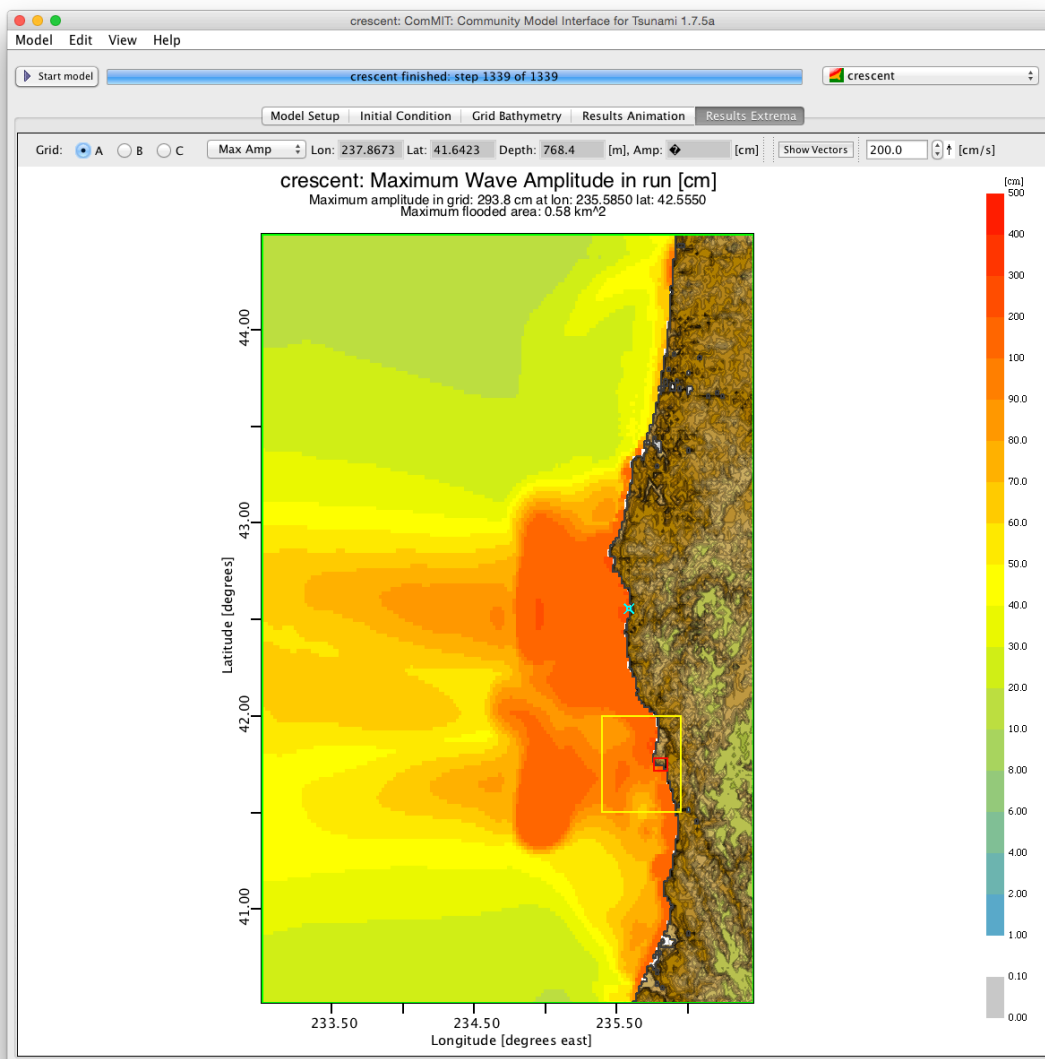


Figure 6: Maximum wave amplitude shown in the A-grid for the crescent Model Run.

ComMIT Menu items

The desktop version of ComMIT currently supports running one Model Run at a time, so the “Model” menu items typically apply to the Model Run that is loaded and showing in the pull-down Model Run selector at the top of ComMIT. A Model Run consists of the 3 bathymetry grids (A-grid, B-grid and C-grid) and the MOST model parameters file. These four items are in a folder named with the Model Run name, hence a Model Run is a folder that can be opened by clicking on the Model Run button in the Model Setup tab.

The Model Menu item

The Model menu item allows the user to Open and Close a Model Run. Results are saved even when a Model Run is closed. Model Run results are only cleared when

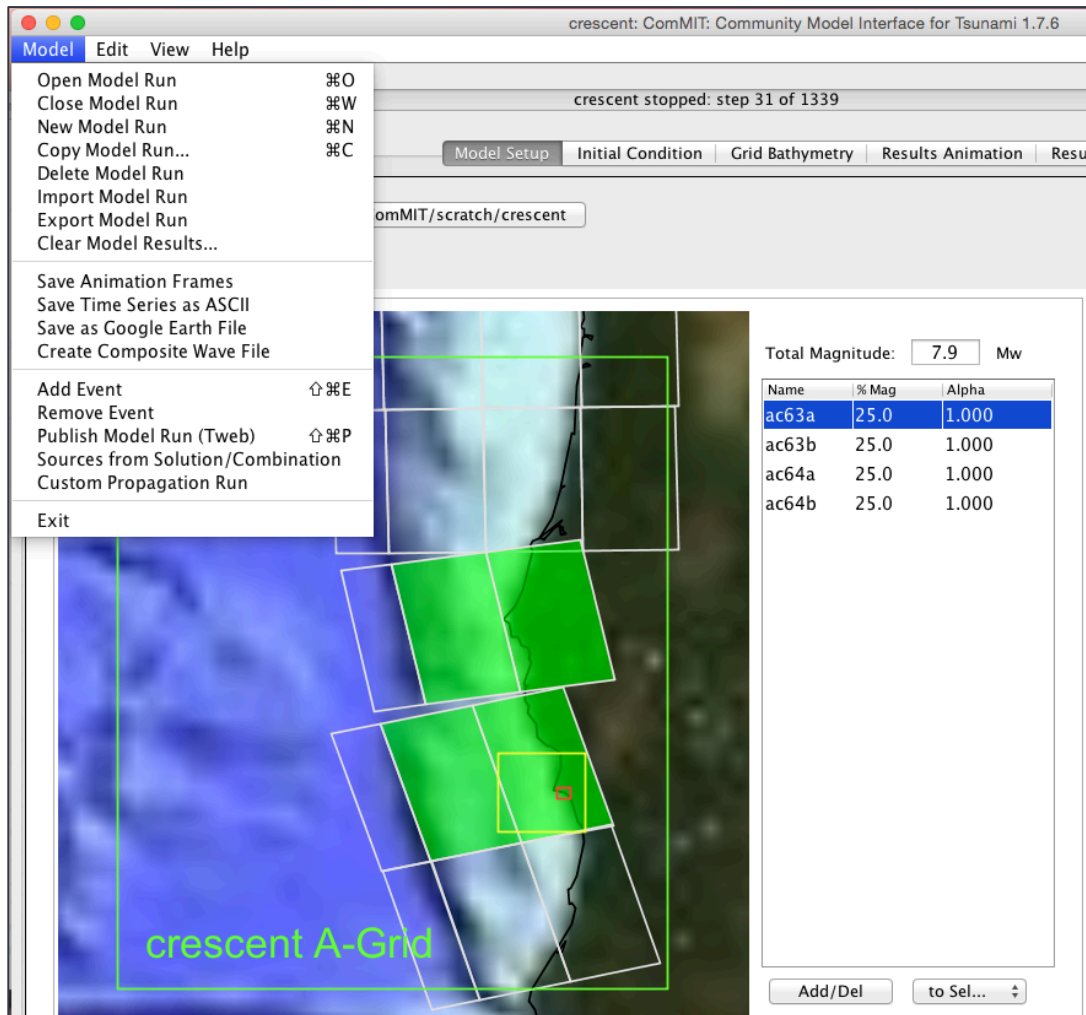


Figure 7: The Model menu item.

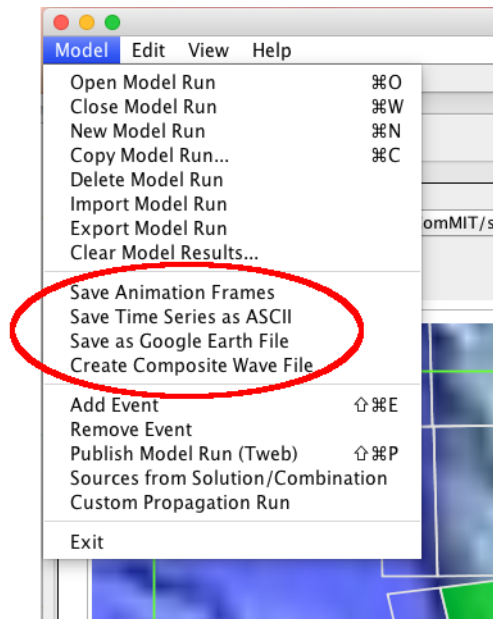
the user chooses new Sources, or when the user chooses the “Clear Model Results” menu item (Figure 7). A “New Model Run” can be created by selecting this option. A dialog helps the user identify three bathymetry grids by either opening files the user supplies or requesting files from the ComMIT Server. More details on this can be found in the following section.

Deleting a Model Run requires the user to be cautious because it is the only function that actually deletes bathymetry files. The user is given a warning about this, but it is wise to keep in mind that *model results are easy to create, while bathymetry is not!* The “Delete Model Run” menu item deletes the entire Model Run folder: model results, grid files, and MOST parameter file. The “Clear Model Results” menu item only clears the model results, which can, of course, be re-created by simply pressing “Start model” and re-running.

The “Import Model Run” and “Export Model Run” simply creates a zip file of the Model Run folder with just the bathymetry and model parameter files. This makes it easy to attach to an email if the user would like help from the modeling Community.

The next group of Model menu items simply allow the user to Save the results in different file formats. The Animation Frames saves each frame of the Results Animation as a sequence of numbered image files in PNG format for creating a movie (the user is tasked with combining these files into an animation). The “Save Time Series as ASCII” saves the time series in the Results Animation tab to a file. The “Save as Google Earth file” does just that, with options to save just maximums or an animation. If a specific tsunami event is chosen to model, times in the Google Earth file are UTC.

The Create Composite Wave File option is described in the next section.



Create Composite Wave File

A common use of ComMIT is to create a hazard assessment or evacuation map for a given location. The technique is covered in the ComMIT training workshops and involves running the same Model Run bathymetry using many different Unit Source combinations. If the user saves all the Model Runs for a hazard assessment, a composite maximum wave amplitude file can be created with this menu item. The user is given a list of Model Runs in their ComMIT folder and

asked which should be included in the composite (Figure 8). Since all the Model Runs must use the same bathymetry in a hazard assessment, using the “Copy Model Run” feature simplifies this task.

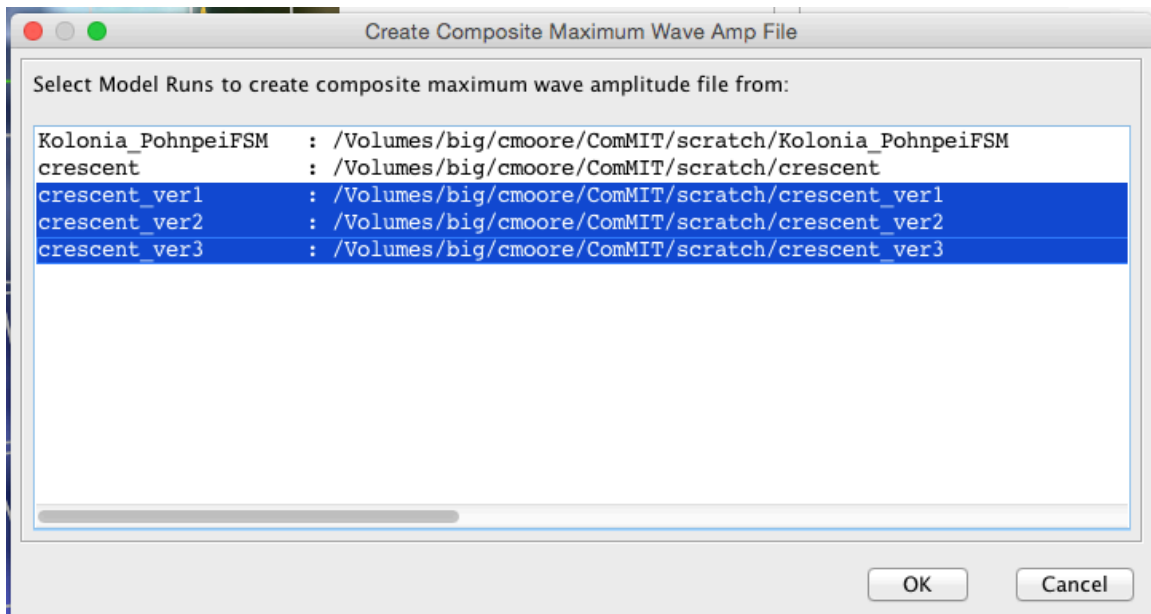
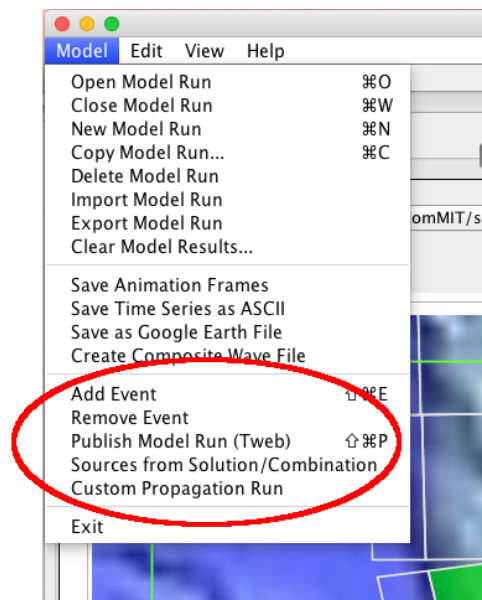


Figure 8: Creating a Composite Wave File. Three Model Runs have been selected by the user.

Tsunami Events and Tweb

The last set of menu items have to deal with getting Unit Sources appropriate for historical tsunami events. The “Add Event” feature opens a dialog showing event information from the archives of real tsunami events. The archives are only a subset of all tsunami events, but each event for which a Unit Source solution has been obtained through the NOAA tsunami forecast system is listed. The information shown includes the date and time (in UTC), the moment magnitude (Mw), the epicenter location (degrees latitude and longitude), and a short descriptor identifying the event.



The dialog allows the user to choose the event from the list of events, giving ComMIT the “authoritative” Unit Sources for the event (Figure 9). As a new event occurs, this authoritative solution is updated as more data comes in from the DART buoy array and new inversions are performed.

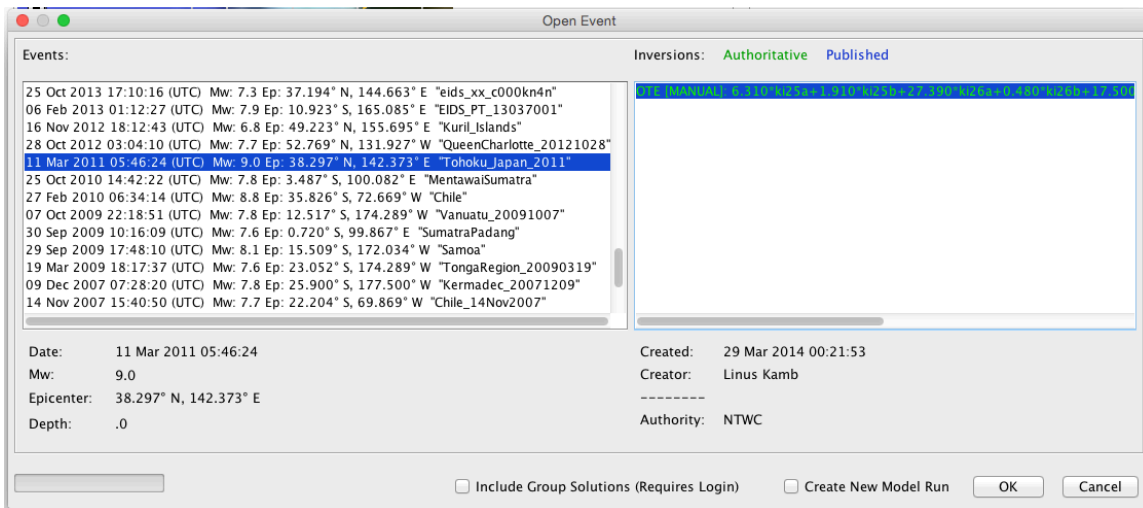
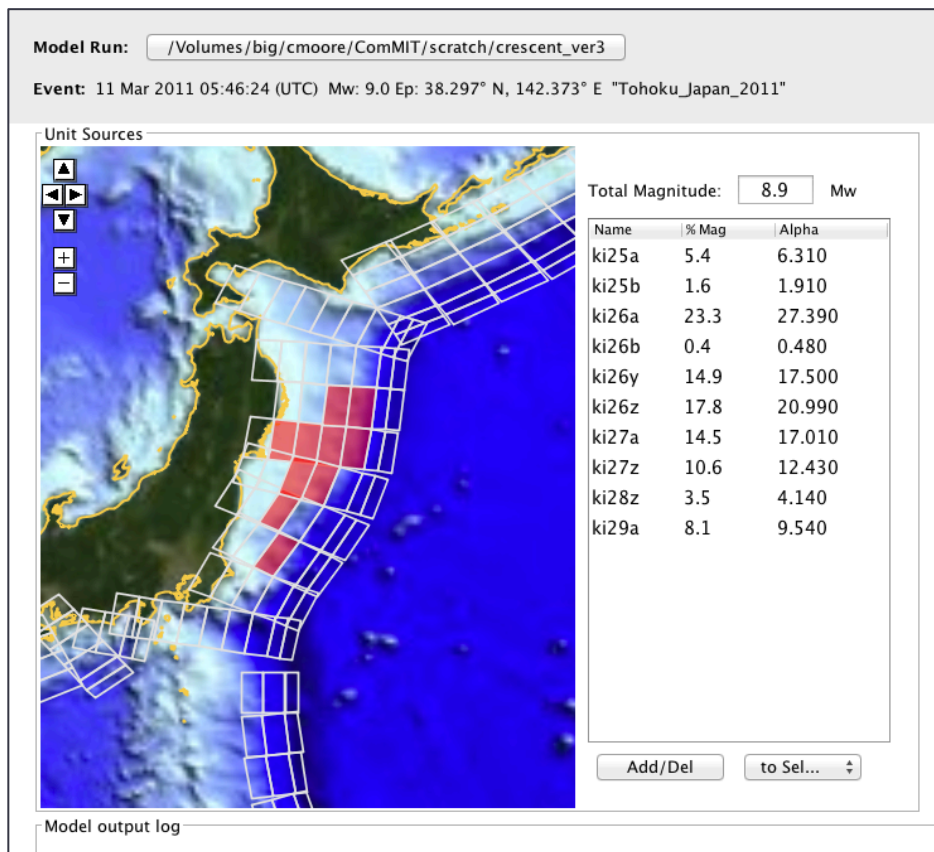


Figure 9: List of tsunami events in the "Add Event" dialog

When a user selects a given event, the Model Run will update the Unit Sources selected in the map on the Model Setup tab, and shows the "Event: ..." information:



The other change is that on the Results Animation tab, if there is a tide gauge in the C-grid, it will be shown as an inverted black triangle (▼) and the tide gauge data for the time period for the event will be plotted in the time series window with the gauge data in black and the model output in red (Figure 10). The user can double-click on the wave amplitude plot to set the time series location (green star) near the black triangle representing the gauge position to compare the model results to the tide gauge results. Keep in mind that the gauge locations reported in the IOC tide gauge web service that ComMIT relies upon are often in error, sometimes by as much as a kilometer or more! Looking at Google Earth or other aerial photography source usually allows the user to identify which pier the tide gauge is deployed on.

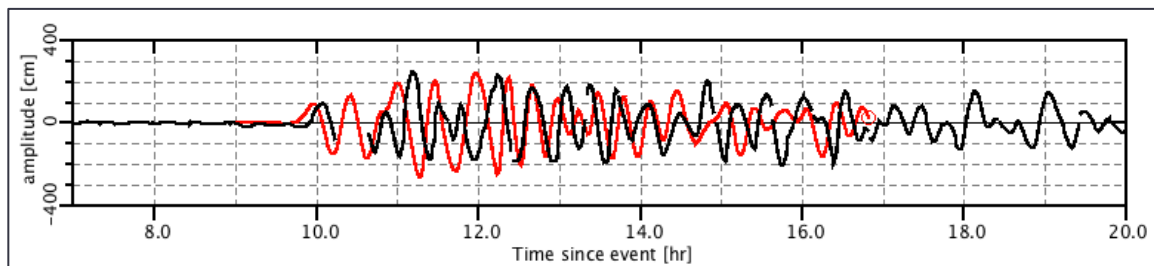


Figure 10: Tide gauge comparison at Crescent City for the Tohoku, Japan tsunami of 11 March 2011

Removing an event from a Model Run simply removes the tide gauge data, and the Event label from the Model Setup tab.

For users with Tweb accounts with “forecaster” status, Publish Model Run allows uploading the model results for a given event to Tweb, the on-line interface to the operational tsunami forecast system, SIFT (Short-term Inundation Forecasting for Tsunamis).

The “Sources from Solution/Combination” allows selecting Unit Sources by entering the Alphas and Unit Source names manually. This is helpful when obtaining a solution via email, or copy-and-paste from another application, such as Tweb.

Custom Propagation Run

The Custom Propagation Run menu item allows the user to add custom seismic Propagation Database files to ComMIT using the MOST model, defining the seismic parameters of an earthquake using the standard for fault plane description as described in the MOST manual (e.g., specifying epicenter, length, width, depth, slip, dip rake and strike). Selecting this item opens a new window for the creation of this new propagation run:

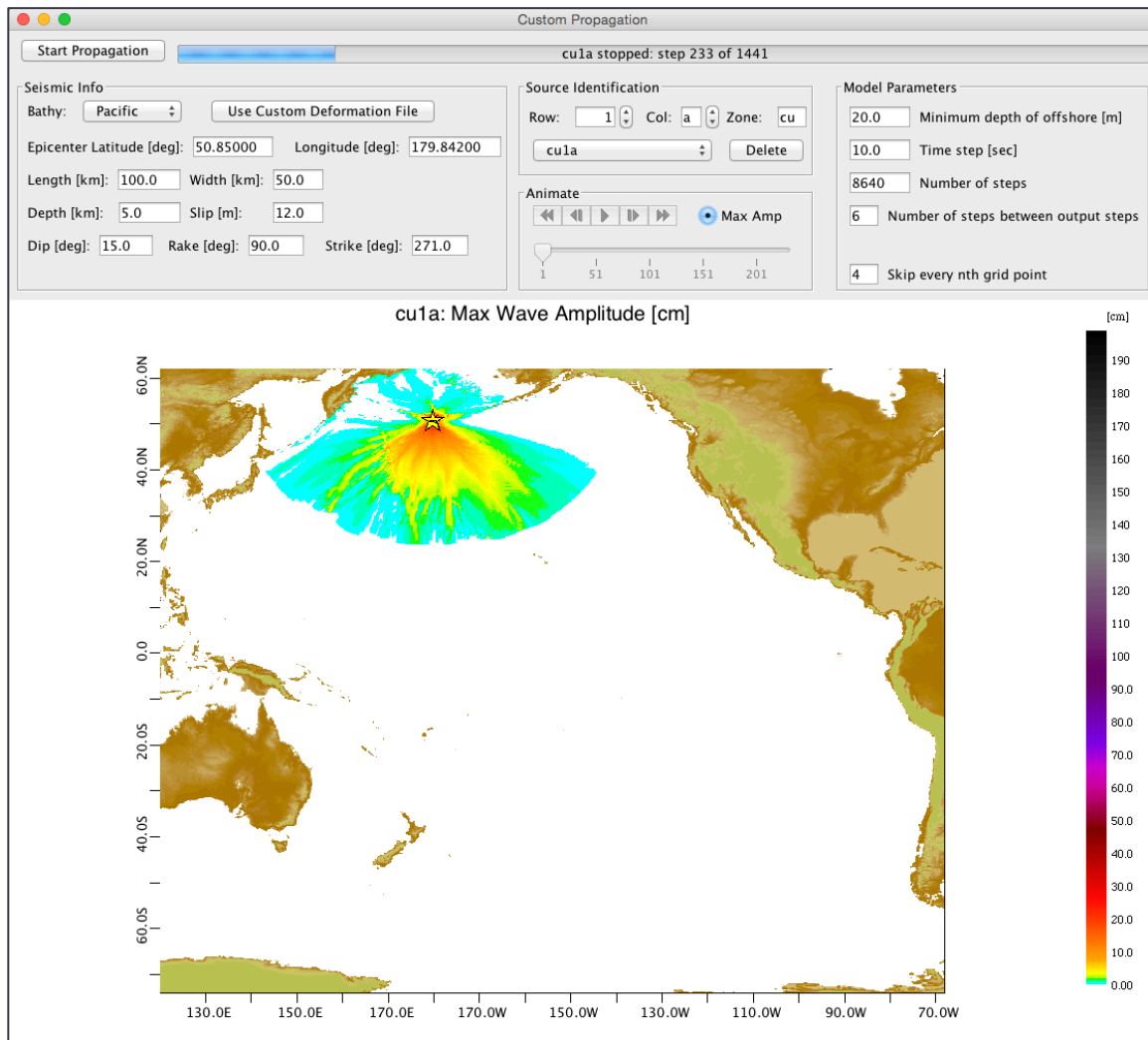


Figure 11: The Custom Propagation window showing maximum wave amplitude as the model runs.

This interface is very similar to the main ComMIT window, in that the propagation model is run with a “Start Propagation” button at the top, with a progress bar showing model progress, and output is animated with the animate buttons or wave maximum amplitude is shown. Full propagation runs can take several hours to run as the grid sizes are considerable. The parameter selection at the top allows the user to set Seismic Info, Source Identification information, or Model Parameters.

Seismic Info

The seismic information is entered at the top left by selecting a bathymetry file (defaults to a Pacific Ocean basin bathymetry), and entering the basic fault plane parameters. The user may double-click on the map to set the epicenter latitude and longitude, as indicated by the yellow star with fault plane outline. When the user changes parameters, the fault plane outline adjusts according to the geometry requested:

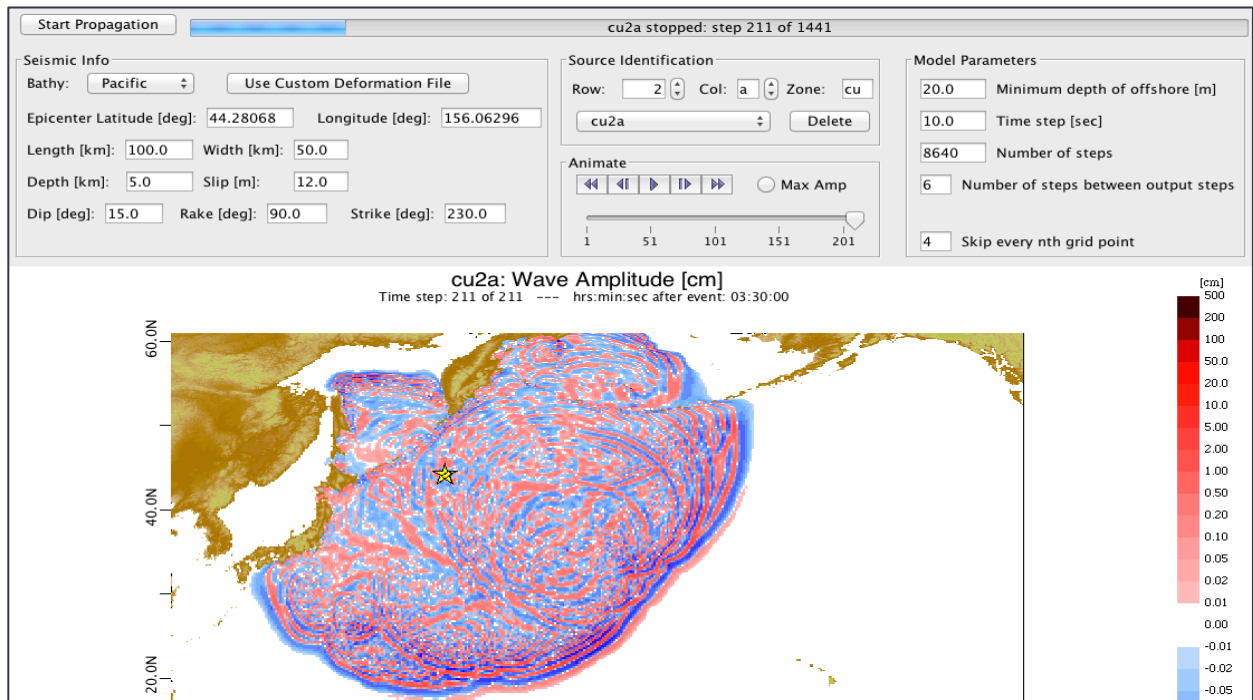


Figure 12: Configuring a custom Propagation run with a source name of cu2a and 12 meter slip.

Source Identification

The source is given an identification to fit the NOAA Propagation Database naming convention, using the “cu” identifier to mark it as “custom”. The row number and column letter can be incremented, and many custom runs can be added and selected in the drop-down selector marked “cu2a” in Figure 12.

Animation

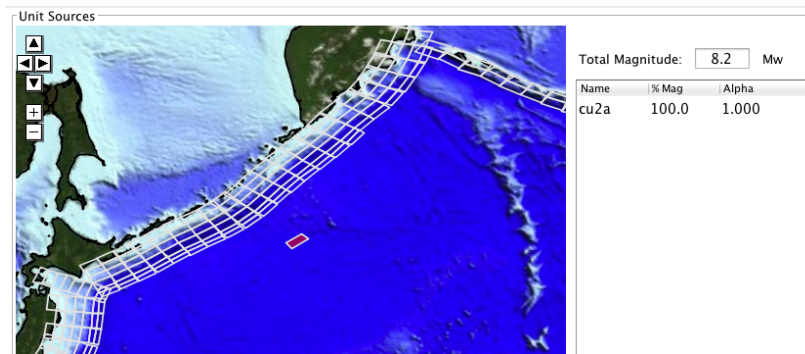
The animation buttons work just as in the main “inundation” window in ComMIT, allowing the user to “play” the animation by clicking the right-facing triangle button, to single-step through the animation, or, by using the slider bar below the animation buttons, to select a given time step. Selecting the “Max Amp” button disables the animation and instead shows the wave maximum amplitude over time as in Figure 11.

Model Parameters

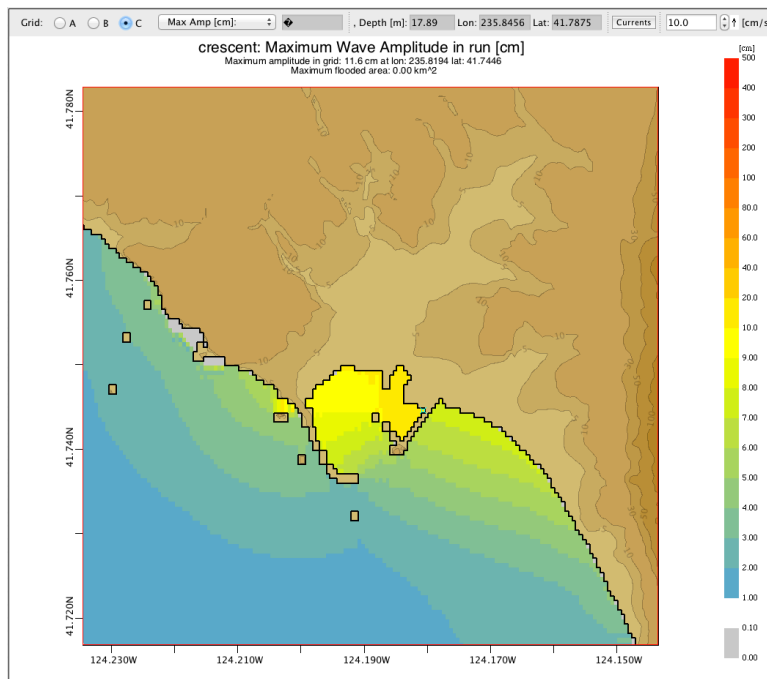
The model parameters are exactly analogous to the main ComMIT inundation model parameters, with the exception that there are no inundation algorithm settings. See the Set Model Run Parameters section for a description.

Using Custom Sources

The resulting sources are automatically added to the Propagation Database in the main ComMIT window, under the Model Setup tab. Notice in this case that the user has chosen an epicenter well offshore of the normal Unit Sources. Using this source is as easy as clicking on it, choosing a Model Run location (e.g. “crescent” for Crescent City), and clicking “Start model”.



In this example, the inundation due to a custom source “cu2a” offshore of the Kuril island chain is seen in the maximum amplitude plot for Crescent City:



The Edit Menu item

The Edit Menu item allows the user to edit either ComMIT preferences or the Bathymetry files. The ComMIT preferences consist of setting the directories (folders) where ComMIT files are, including default locations for bathymetry grid files (Grids Directory), the directory Model Runs are in (Scratch Directory), the MOST model executable (usually downloaded from the ComMIT Server), and the ComMIT Server itself (Figure 13). The Unit Sources in ComMIT are provided through the ComMIT Server, but the Propagation Database tab allows adding a local repository of propagation model results to use as sources. The Network tab allows setting proxy information if you are running ComMIT behind a proxy firewall.

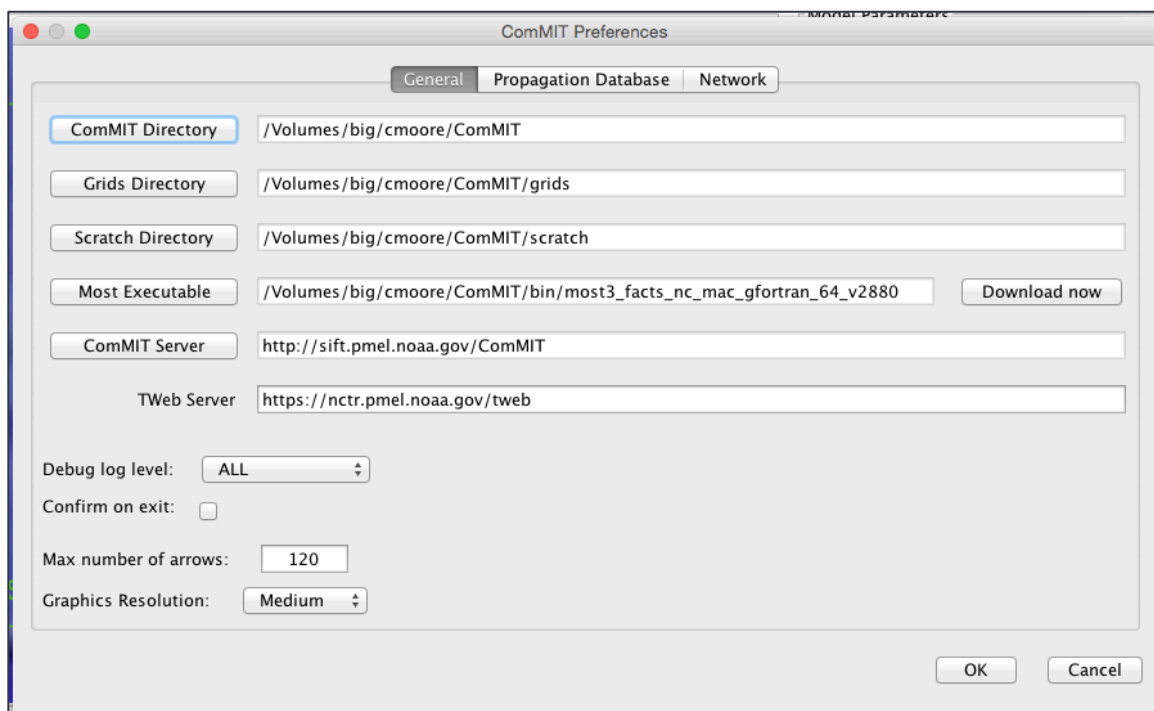


Figure 13: The ComMIT Preferences dialog.

The View Menu item

The View menu item allows the user to see information about the bathymetry grids, or to see the extents of the Unit Source propagation files (grouped into Pacific, Indian and Atlantic Ocean basins).

The View Grid Information (CFL) menu item shows a dialog of statistics about each of the three bathymetric grids for the open Model Run (Figure 14). The information includes filename, dimensions, extents (in latitude and longitude), the resolution or “spacing”, the maximum height and depth, and the CFL criterion for each grid.

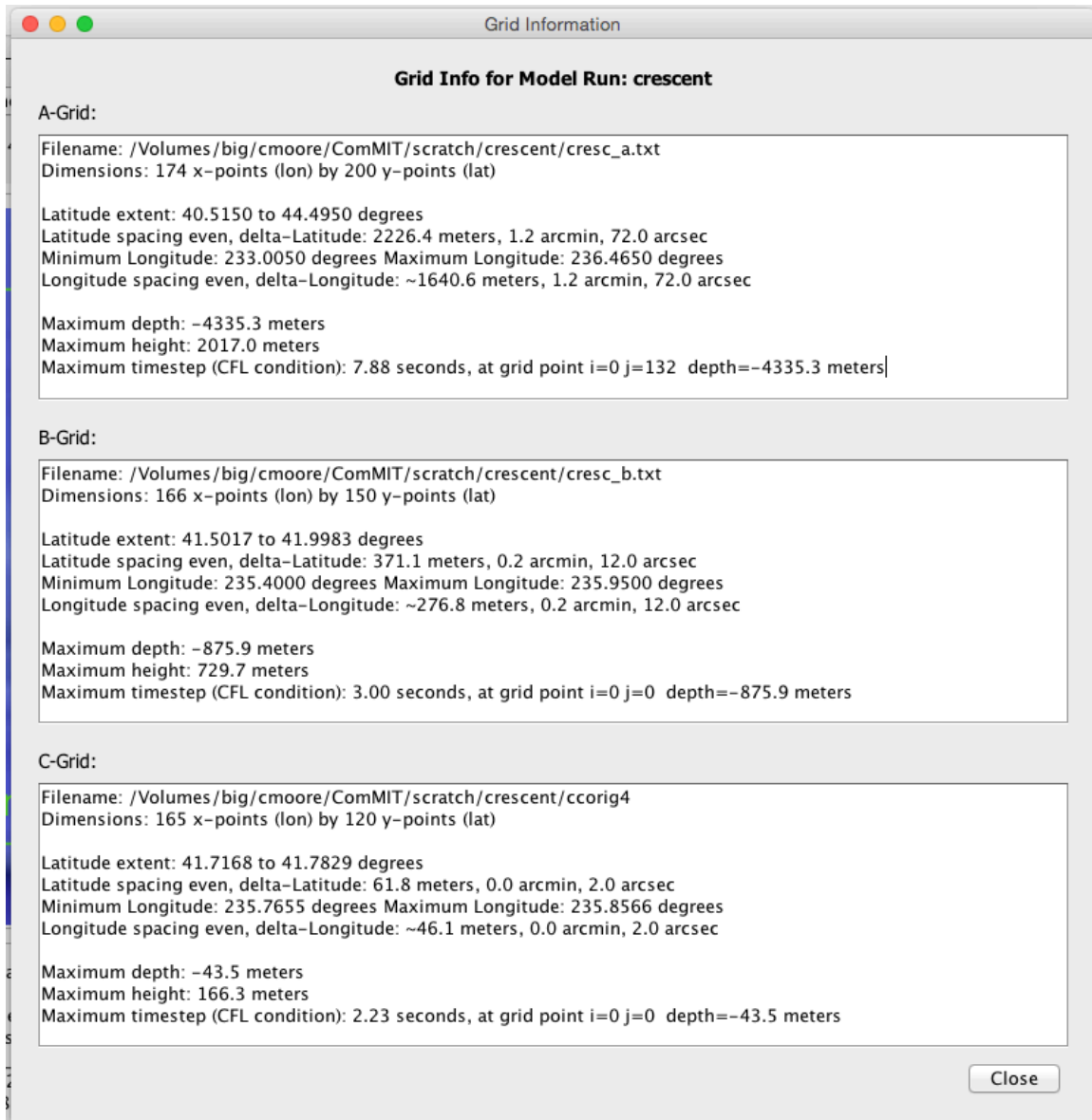


Figure 14: The Edit -> View Grid information (CFL) dialog for Crescent City.

Creating New Model Runs

Global bathymetry data

New Model Runs are created by using the Model -> New Model Run menu item. Users can use this feature to either load their own bathymetry grids, or download bathymetry grids from the ComMITServer.

The ComMITServer provides combined bathymetric/topographic gridded digital elevation models for tsunami modeling.

While the dataset is useful for training and demonstration purposes, it was automatically generated primarily from low-resolution bathymetric data sources, and has received limited manual review. It should not be used for hazard mapping or similar purposes.

The ComMITServer bathymetry dataset is derived from two sources:

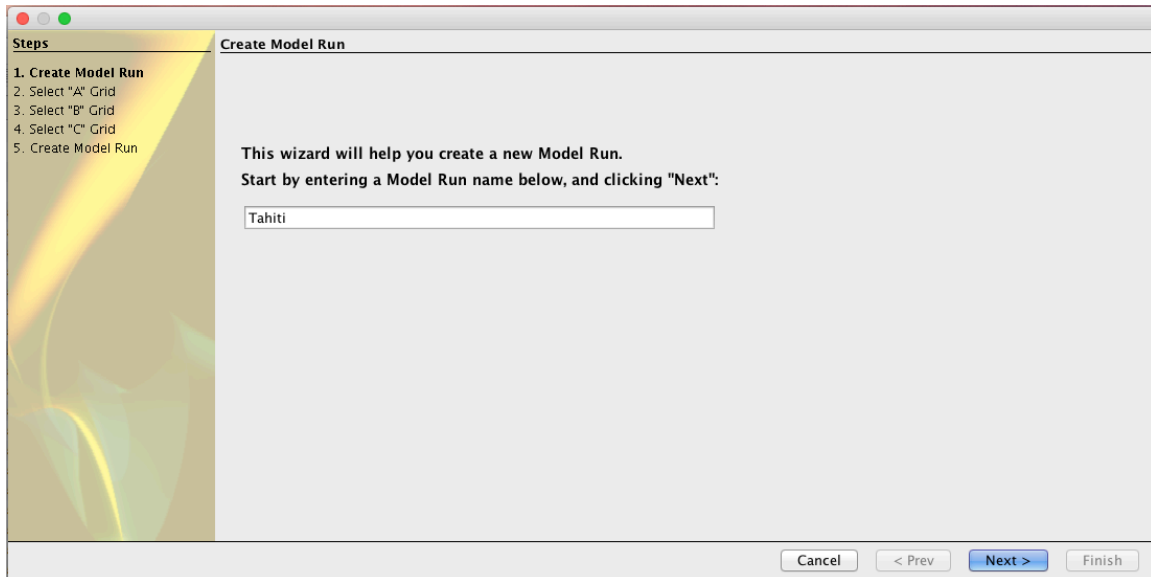
- Bathymetry is from the [ETOPO1 1 arc-minute gridded global relief model](#) produced by the NOAA National Geophysical Data Center. The ETOPO1 dataset is compiled from many different global and regional data sets, evaluated and edited, and shifted to reference mean sea level vertically, and WGS84 horizontally. This data has been INTERPOLATED FROM 60 ARC-SECONDS TO 3 ARC-SECONDS to match the topography data set. This makes it inappropriate for any use except training.
- Topography is from the [CGIAR SRTM 90m](#) version 4 digital elevation model produced by the CGIAR Consortium for Spatial Information. This 3 arc-second (~90 meters at the equator) dataset is a processed version of the Shuttle Radar Topography Mission (SRTM) data created by NASA.

The Etopo1 data was interpolated from 60 arc-seconds onto the 3 arc-second SRTM grid after setting Etopo1 land values to -1.5 meters. Land values are then replaced with the SRTM data. This interpolation produces a dataset is relatively inaccurate in bathymetry, but tends to be stable for most sources using the NCTR propagation database, and is therefore useful for training purposes. NCTR recommends it be used *only* for this purpose.

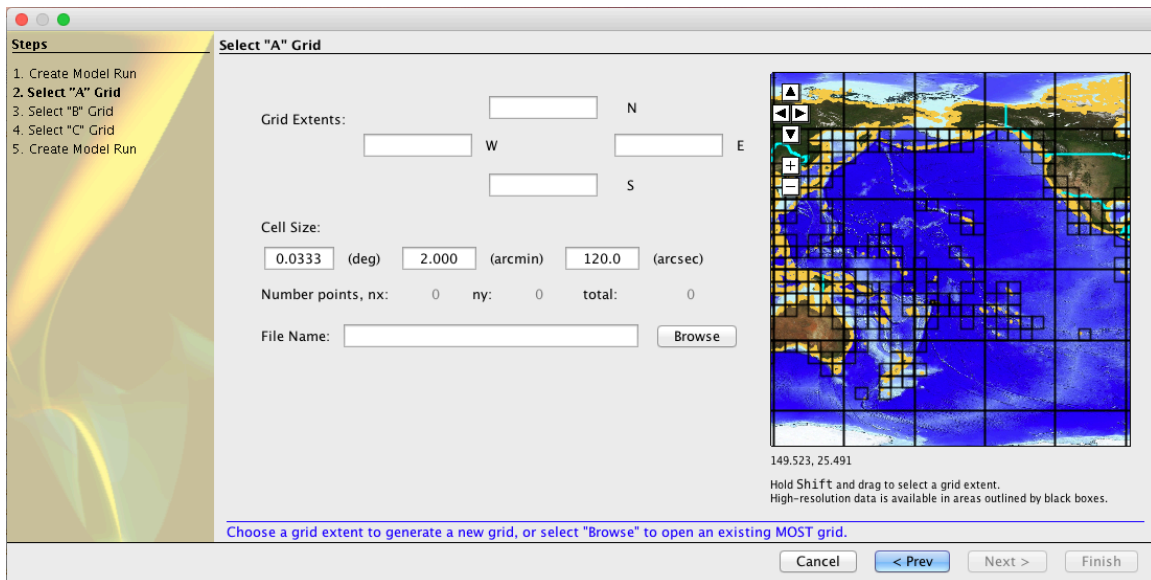
Using the New Model Run Wizard

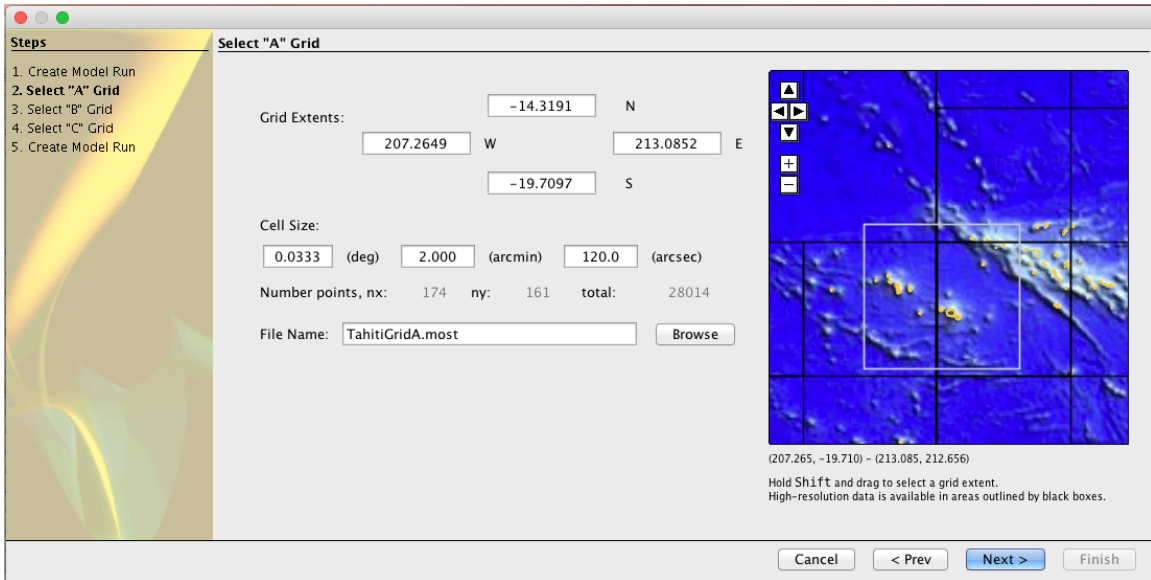
To use this data, user clicks on the Model -> New Model Run menu item. This opens a Wizard that helps the user select (or request) bathymetry grids, and creates a default Model Parameters file. The Wizard walks the user through a

series of steps, the first of which simply asks the user to name the Model Run and click “Next”:



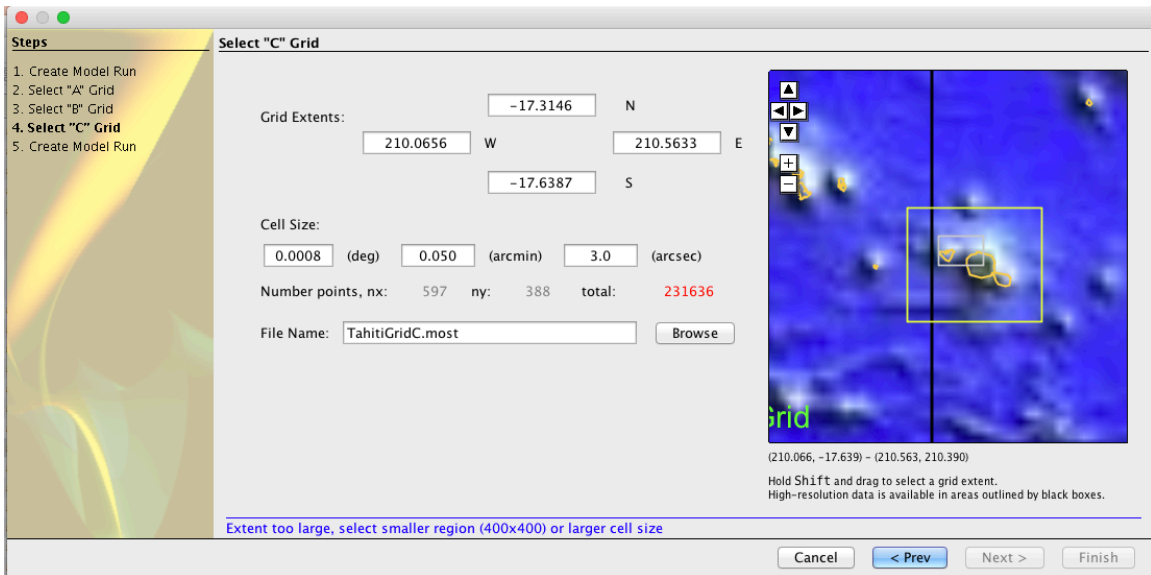
In steps 2-4, the user selects the three bathymetric grids. The user can either browse to a grid file by clicking the “Browse” button, or zoom the map to a region and “rubber-band” an extent by holding down the shift button and dragging the mouse:



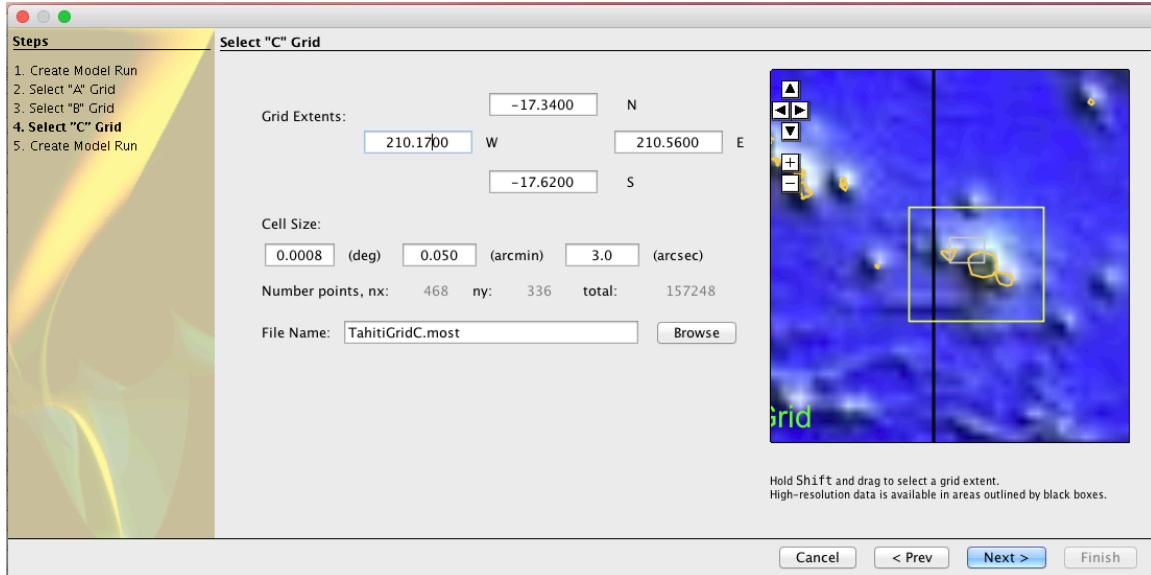


After a region is selected by holding the shift key and dragging the mouse, the Grid Extents are shown, and a grey rectangle of the region is plotted. For the A-grid, the region should be at least as large as the 5 degree by 5 degree black squares on the plot. The default cell size is 120 arc seconds. In this case the dimensions of the A-grid will be 174 by 161. There is a limit on the size of the grid requested from the ComMIT Server of 160,000 points (e.g., 400 by 400). The Wizard does not allow requesting files larger than this.

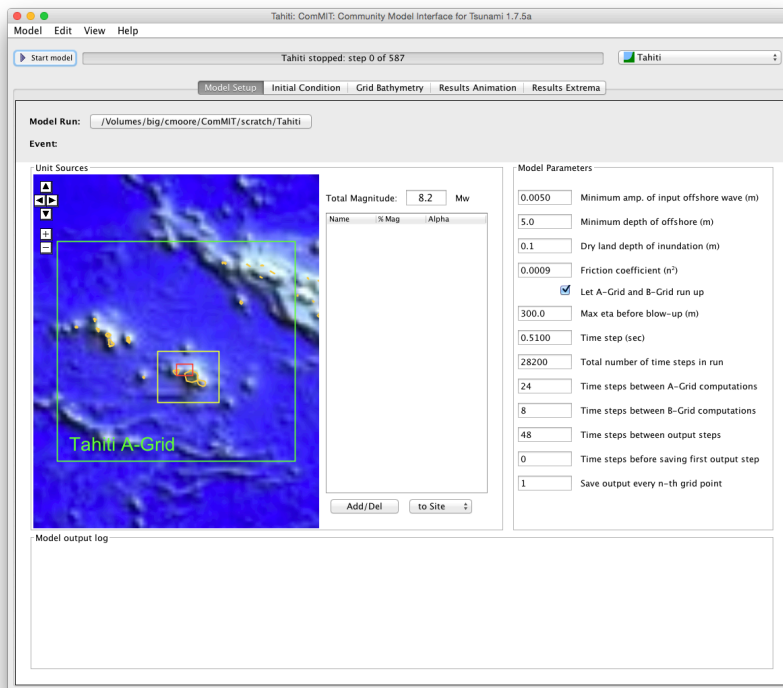
After selecting the region, the user can click “Next” and select the B-grid and C-grid in the same way. As the resolution of the background map is fairly crude, it often helps to have a reference map, such as Google Earth, open to pinpoint the C-grid area of interest. It is helpful to “rubber-band” a region, and then edit the Grid Extents manually to get exactly the extents you want:



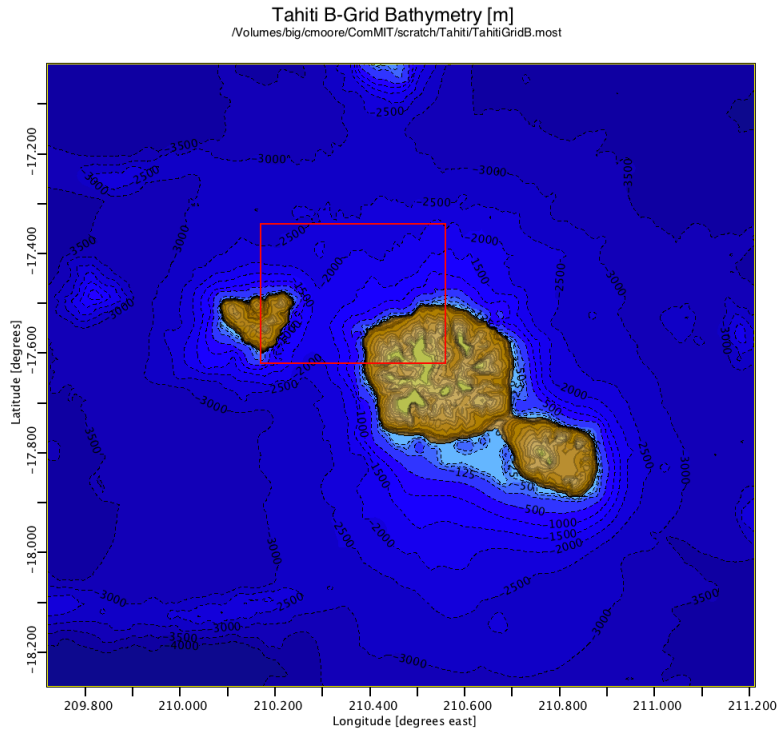
Adjusted manually to the exact extents to lower the total number of points:



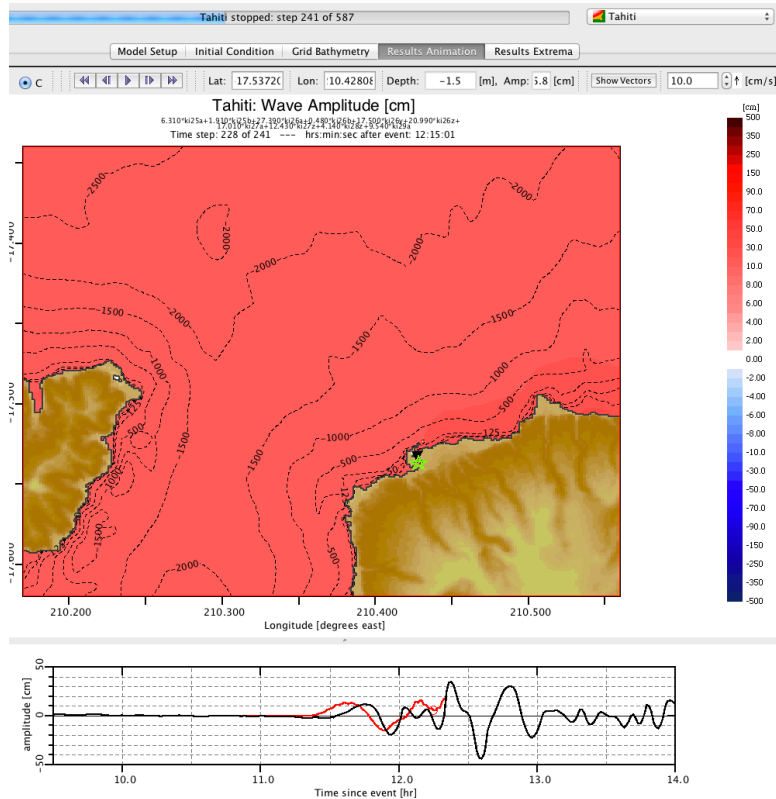
The last step requires the user to click “Finish” to send the request to the ComMIT Server. Requesting bathymetry grids can take a while: ComMIT gives a progress bar in the lower left part of the Wizard while it downloads grids. When it is done, the user clicks “Close” to close the Wizard, and the new Model Run is loaded:



The user can view the grid files under the Grid Bathymetry tab:



And run the model to see inundation results:



Note that bathymetry grids from the ComMITServer are saved under the ComMIT/grids folder, and then copied to the Model Run folder. If a user-supplied grid is used (by clicking Browse in the Wizard), it, too, is copied to the Model Run folder so as to preserve the original file. Bathymetry cropping and smoothing only affect the version in the Model Run folder, not the original file.

More Information

For more information about ComMIT, tsunami forecasting, and hazard assessment, please visit the NOAA Center for Tsunami Research at:

<http://nctr.pmel.noaa.gov>