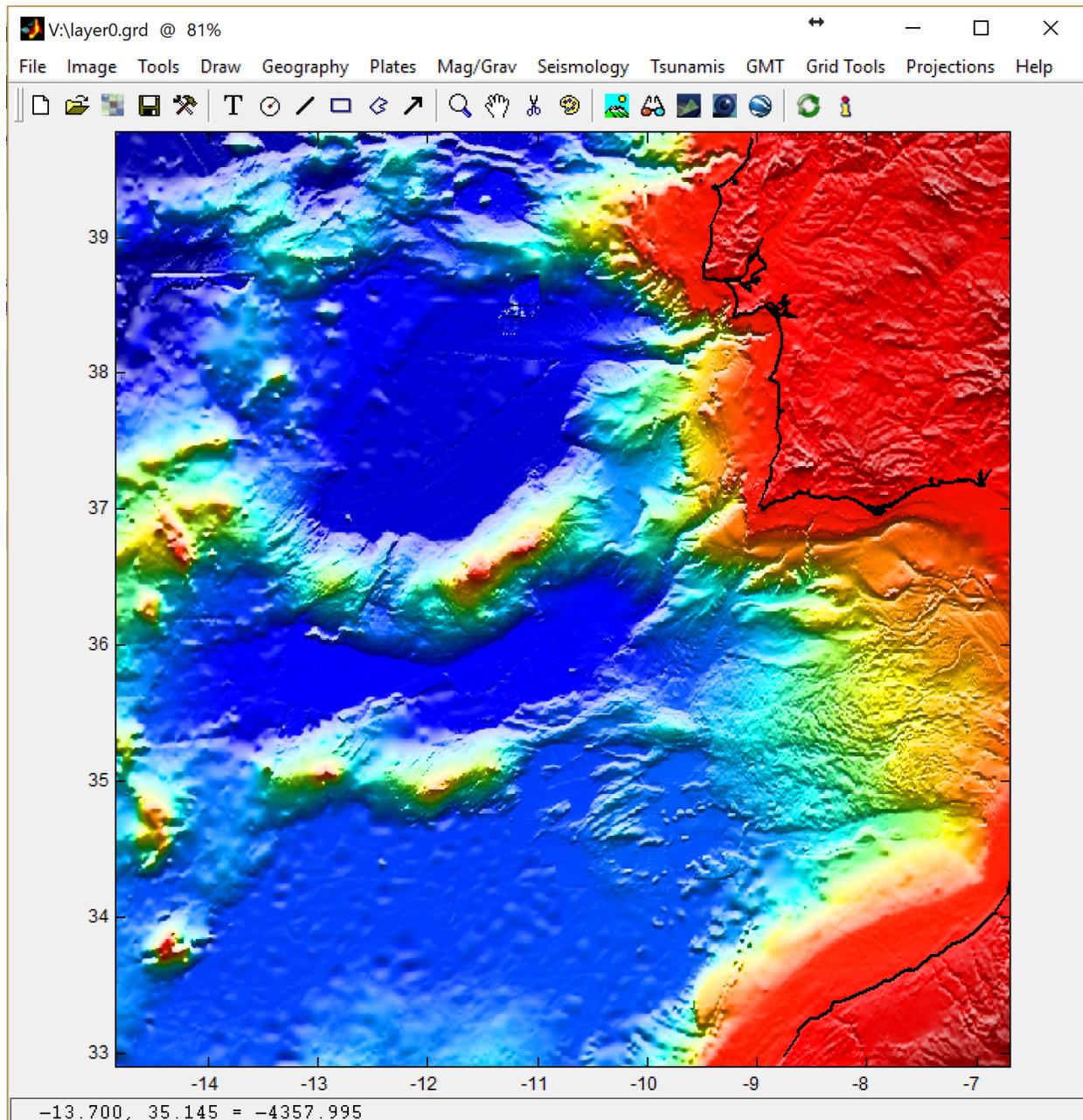


A simple tsunami propagation example

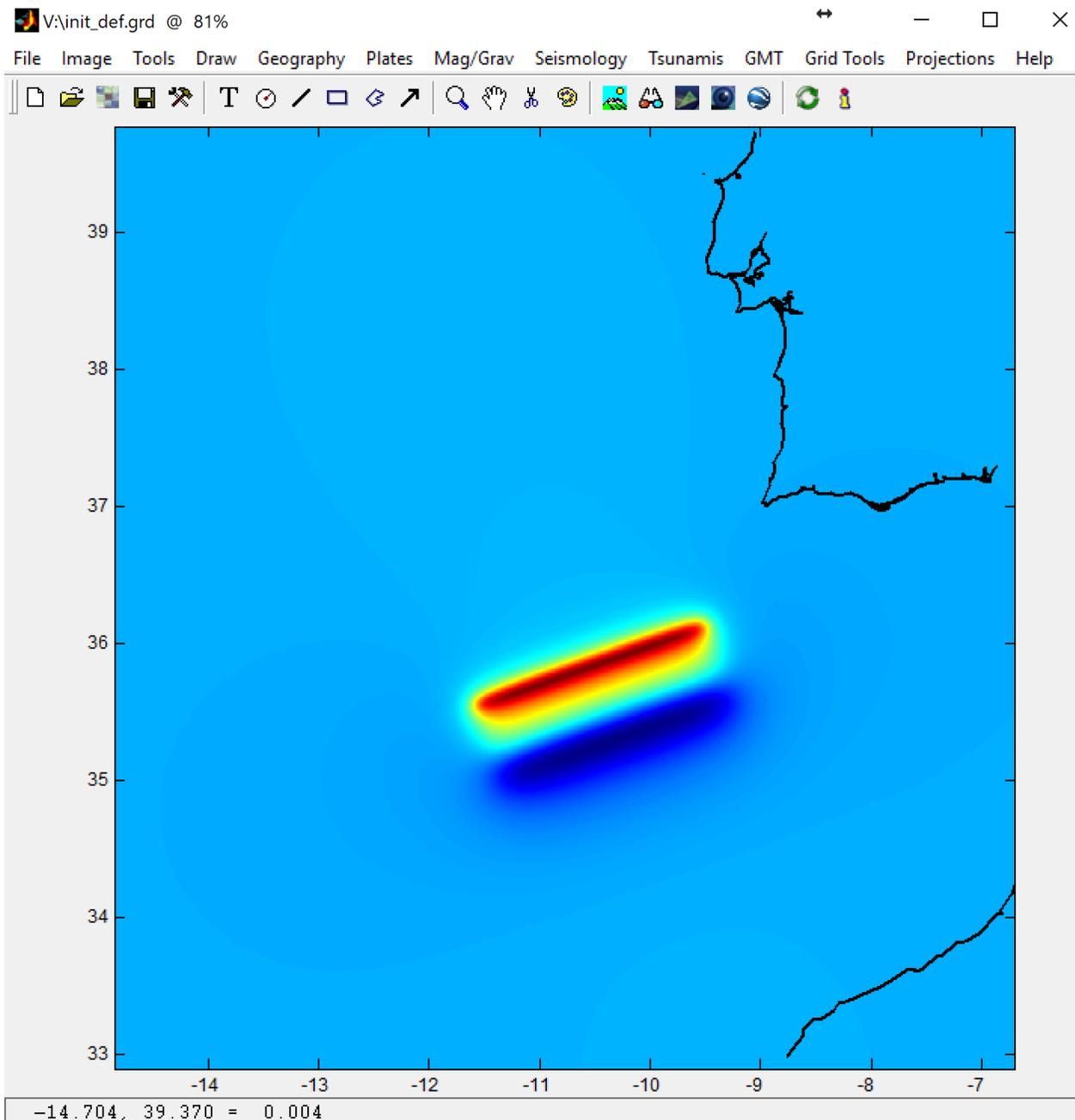
For modeling tsunami wave the propagation we only need one bathymetry grid and the initial condition of water displacement. This initial condition is normally calculated after the seismic parameters of a certain seismic event. Please refer to the tutorials on "[How to compute seismic elastic deformation](#)" or "[Compute seismic deformation and interferogram of a multi-patch solution](#)"

Open the grid with the bathymetry



Not strictly necessary but to help visualize we added the coastlines and shaded illuminated the grid. Note also that the bathymetry must be in meters with the z axis positive up (i.e. depths are negative).

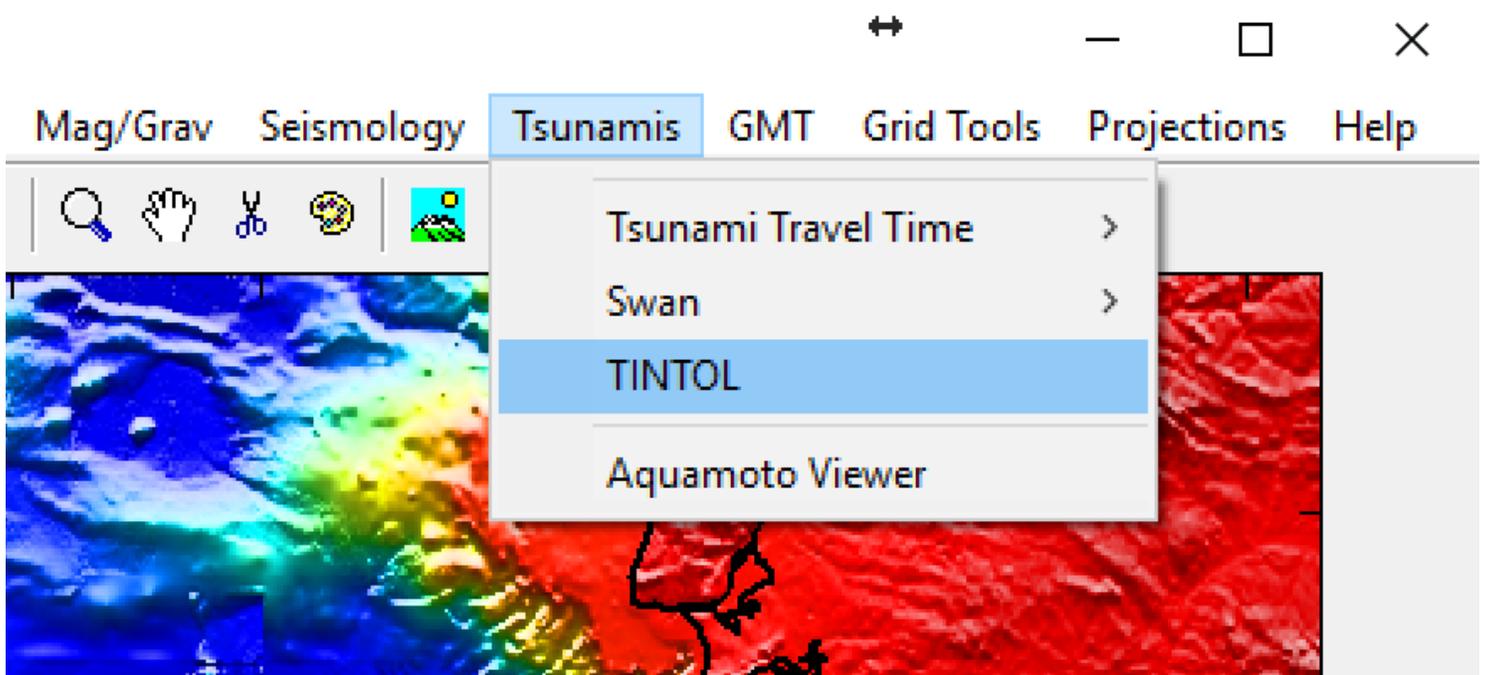
To confirm that you have the things you need, open also the initial deformation grid



This is an example of an initial deformation computed as described in the links above. It is absolutely crucial that both the bathymetry and deformation grids have exactly the same grid steps and cover exactly the same region.

Next we are going to transform the Mirone standard figure into the TINTOL mode.

Move to the TINTOL mode



Just do like in the above and land ...

The TINTOL environment

V:\layer0.grd @ 81%

File Draw Geography Tsunamis Projections Help

The screenshot displays the TINTOL software interface. The main window shows a bathymetric map of the Mediterranean region, with a color scale ranging from blue (deep) to red (shallow). The map is overlaid with a grid and a black outline of the coastline. The x-axis (longitude) ranges from -14 to -7, and the y-axis (latitude) ranges from 33 to 39. The left panel contains the following configuration options:

- NSWING
- Source: V:\init_def.grd
- Nest: [empty]
- 0 -- level ready to use
- Bordering
- Output grids: Output grids, ANUGA .sww, MOST .nc
- Name: V:\tsu_time_
- Surface level: Surface level, Total water, Max water
- 3D file: 3D file, Velocity, Momentum
- Maregraphs: Maregraphs, Saving step (time = Time step * this) 1
- In file: [empty]
- Out file: [empty]
- Nº of cycles: 2000, Jump initial: 0
- Time step (sec): 1.5, Saving step (cycle units): 20
- RUN

The Parameters section

The screenshot shows a software interface for setting simulation parameters. The parameters are organized into several sections:

- Source and Nest:** Two text input fields. The first is labeled "Source" and contains "V:\init_def.grd" (marked with a circled 1). The second is labeled "Nest" and is empty (marked with a circled 2). Both have folder icons to their right.
- Level Selection:** A dropdown menu showing "0 -- level ready to use".
- Bordering:** A text input field containing "Bordering" (marked with a circled 2).
- Output Format:** Three radio buttons: "Output grids" (selected, marked with a circled 2), "ANUGA .swm", and "MOST .nc".
- Name:** A text input field containing "V:\tsu_time_".
- Simulation Options:** A grid of options:
 - Surface level (selected, marked with a circled 3)
 - Total water
 - Max water
 - 3D file (checked, marked with a circled 4)
 - Velocity
 - Momentum
- Maregraphs:** A checkbox (unchecked) and a text input field "Saving step (time = Time step * this)" containing "1".
- In file:** A text input field (empty) with a folder icon.
- Out file:** A text input field (empty) with a folder icon.
- Simulation Control:** A grid of input fields:
 - Nº of cycles: "2000" (marked with a circled 5)
 - Jump initial: "0"
 - Time step (sec): "1.5" (marked with a circled 6)
 - Saving step (cycle units): "20" (marked with a circled 7)
- Run Button:** A button labeled "RUN" at the bottom right.

Here is where the user must take some decisions (make choices). In (1) we gave the file name of the initial deformation grid. (2) tells the name of the output file. In (3) we selected to compute the water surface level and in (4) we said that we want a single 3D netCDF grid with all the steps. In (5) we decided how many run steps this simulation will run. (6) indicates the time step (in seconds) increment for this simulation. The program proposes one based on the so called CFL condition but user should round it to a more convenient figure (but NOT very different from what was proposed). Steps (5) and (6) determine to total time of the simulation. In this case $2000 * 1.5 = 3000$ seconds. Finally (7) tells the program what is the saving step of the disk file. 20 means saving a step every 30 s of simulation ($20 * 1.5 = 30$).

And that's it, just hit RUN, wait a little (nothing special, it took about one minute and 15 seconds to run in my 2014 laptop) and when it finishes open the output file in Mirone that will take you to the Aquamoto tool where you can slide trough all saved time steps, as in ...

A view of the tsunami simulation for the time 1530 sec

