SAND2016-9626 TR

SNL-Delft3D-CEC

3D/2D modeling suite for integral water solutions

Building an Advanced Model

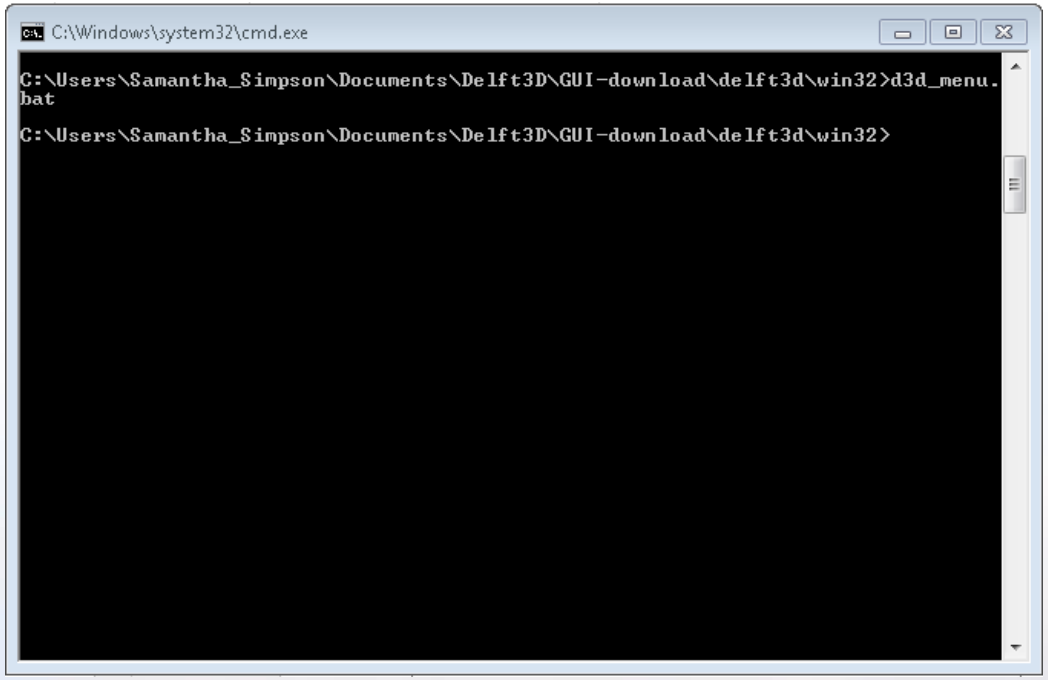
Sandia National Laboratories is a multi-mission laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

Overview

This tutorial will aim to:

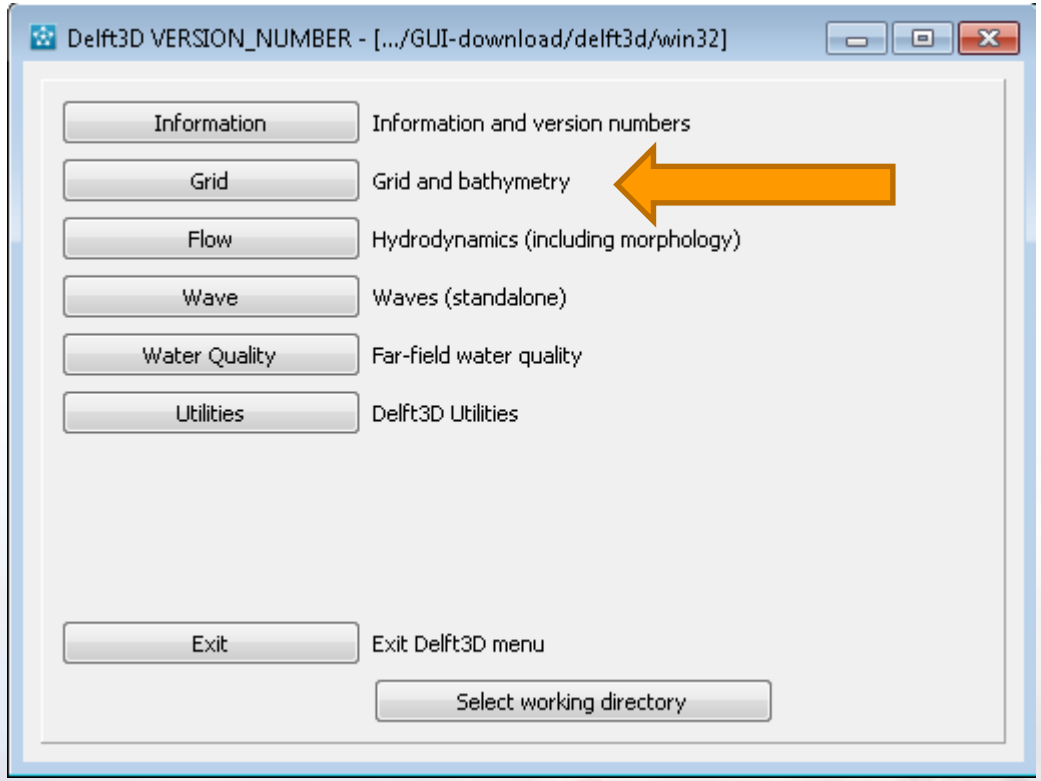
- **Edit grids with attribute files (polygons)**
- **Create depth files through interpolation and extrapolation**
- **Manually add boundaries to geographical grid**
- **Run a simulation**

Delft 3D initialization



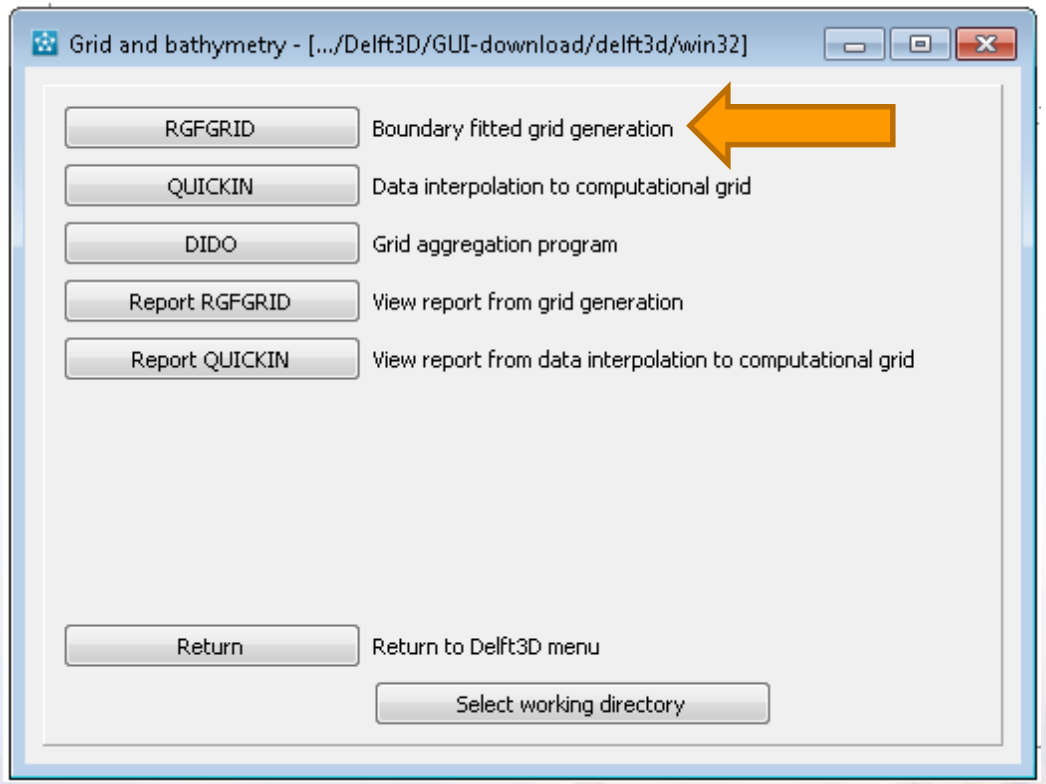
- Start the Delft 3D program.
- From the Windows command line, run d3d_menu.bat

Grid



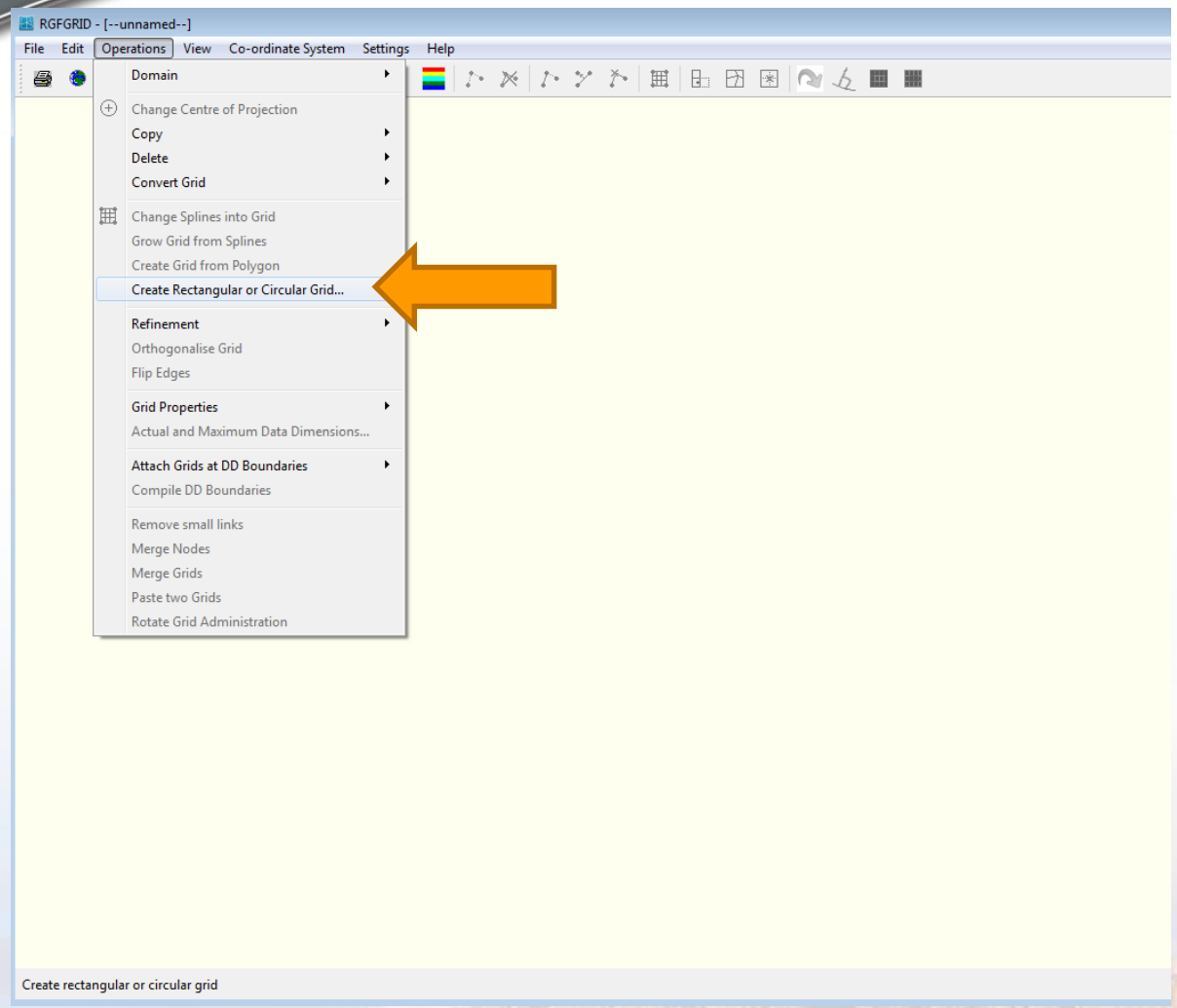
- From the main menu, select Grid

RGFGRID



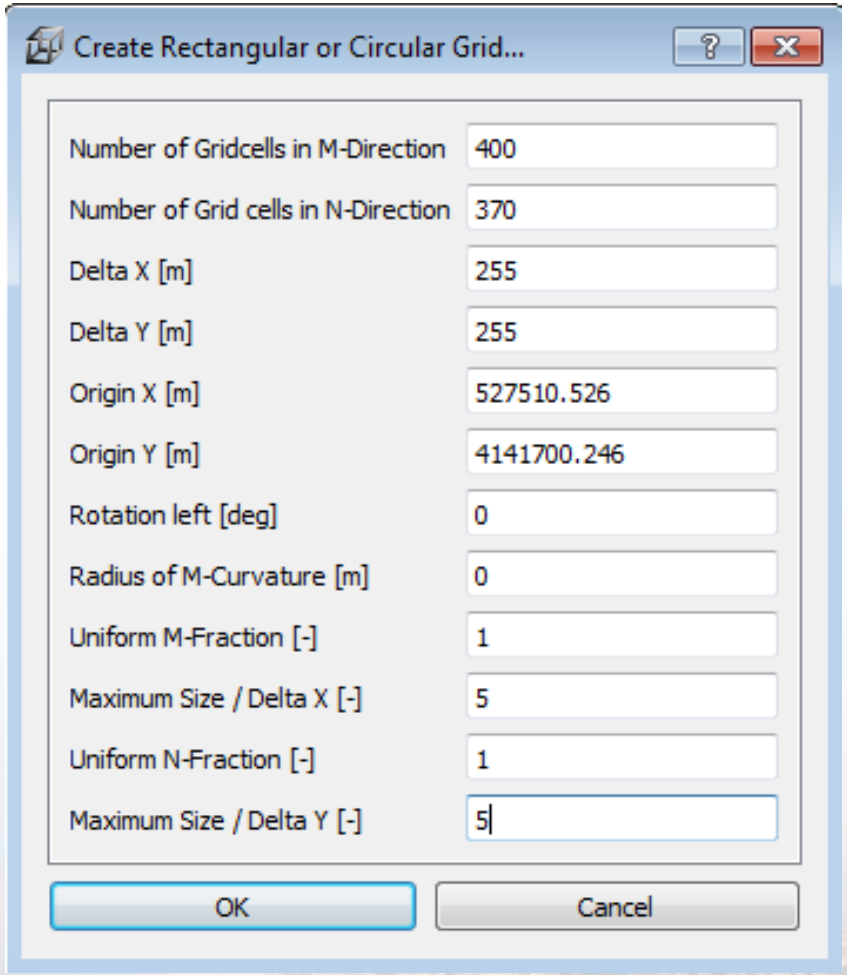
- From the Grid and Bathymetry menu, select RGFGRID

RGFGRID



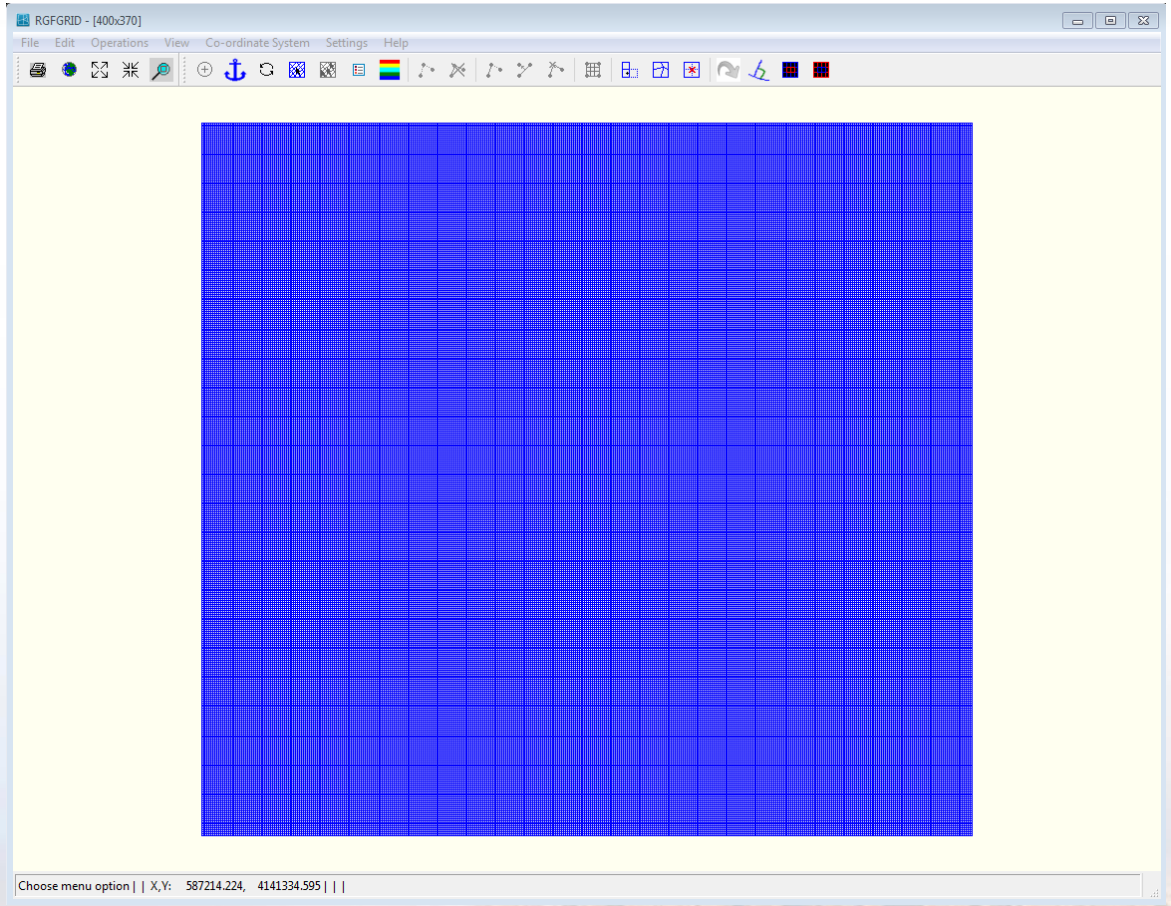
- In this tutorial, we will create our own 400x370 grid to be edited into the shape of SF Bay.
- Go to Operations → Create Rectangular or Circular Grid

Create your own Grid



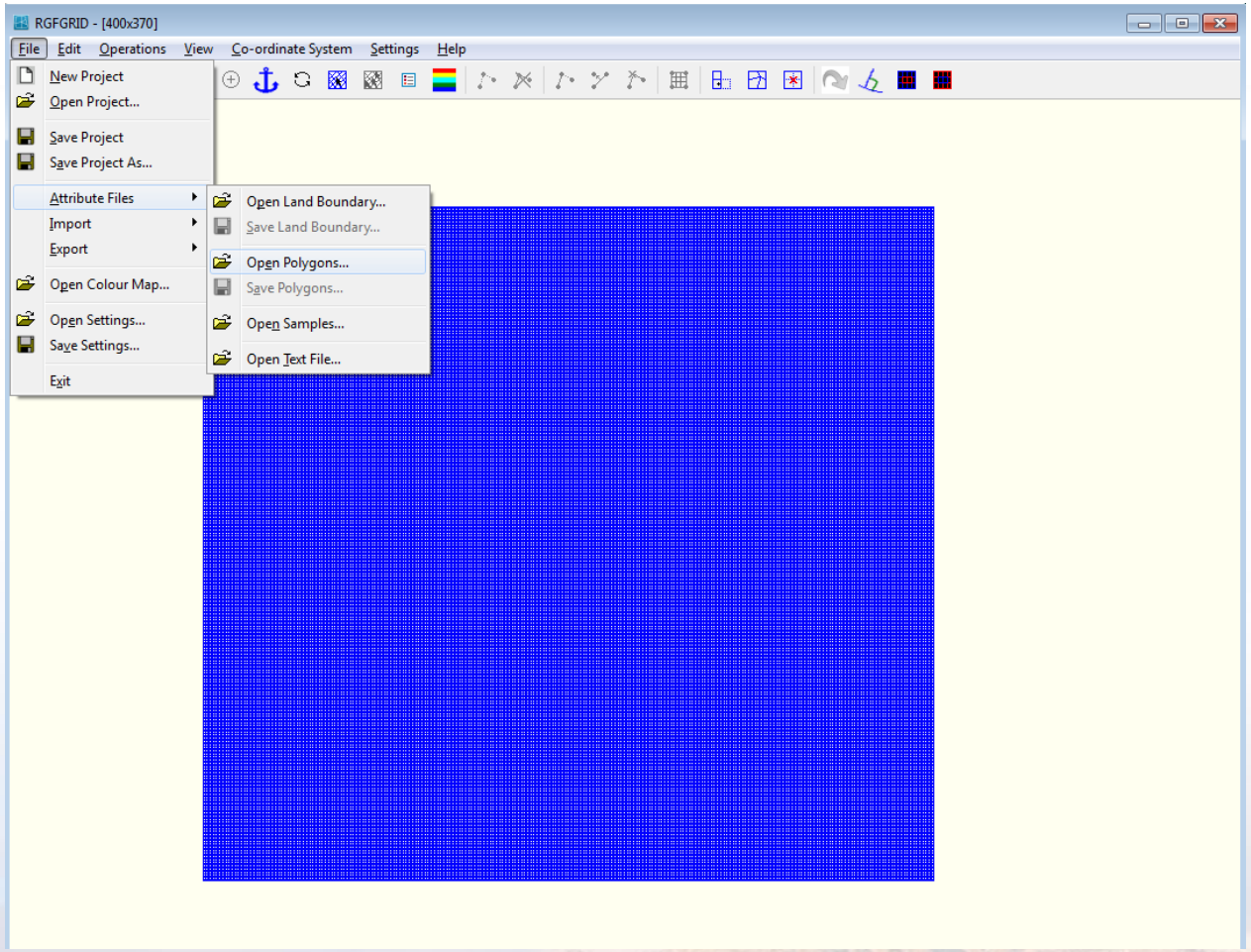
- **Create grid to match these values.**

RGFGRID



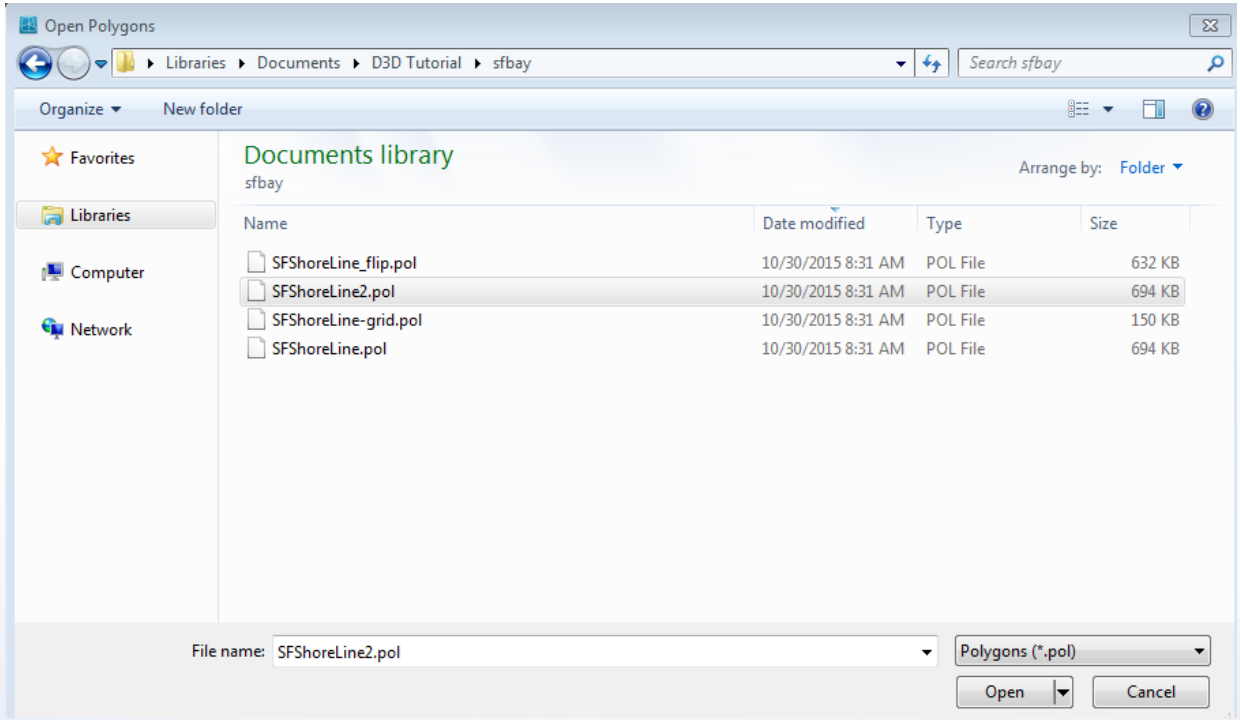
- This is what you should see.

RGFGRID



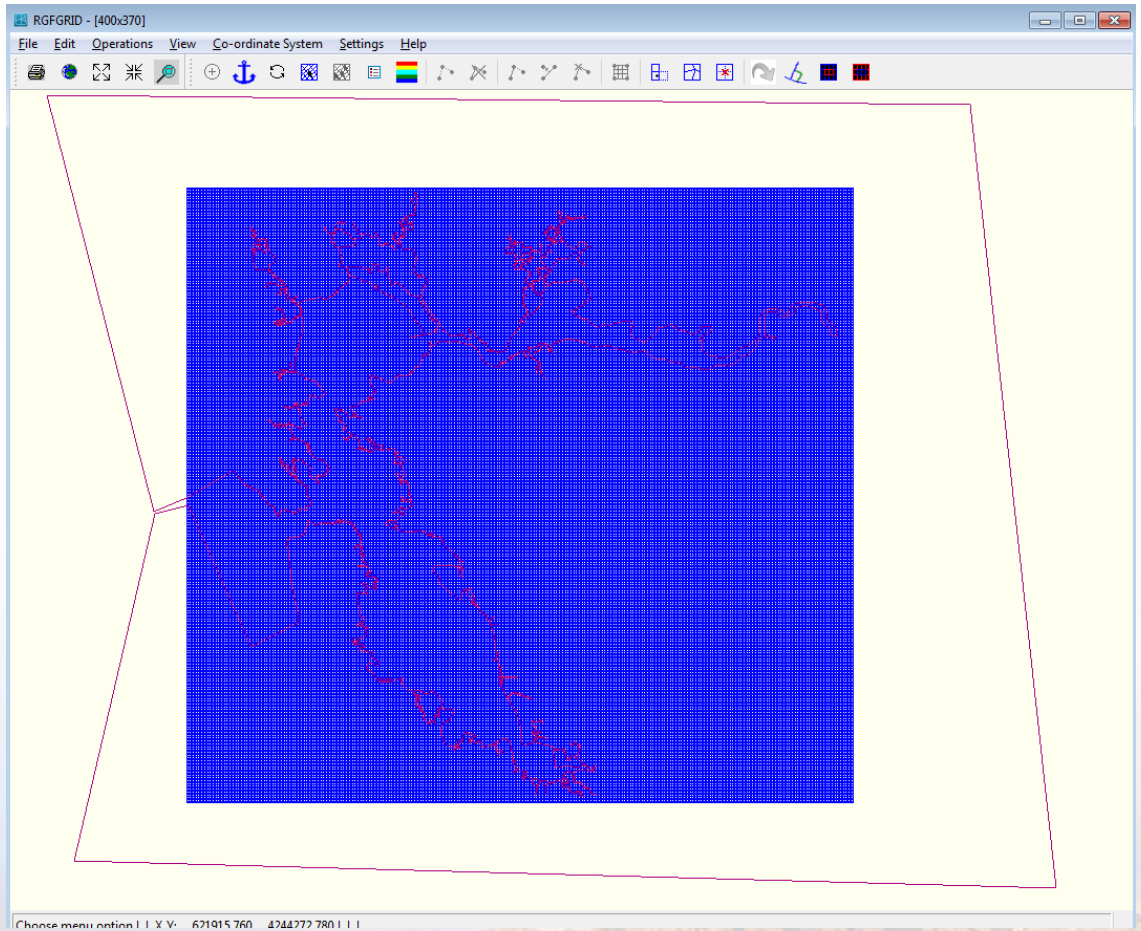
- Select File →Attribute Files→ Open Polygon

RGFGRID



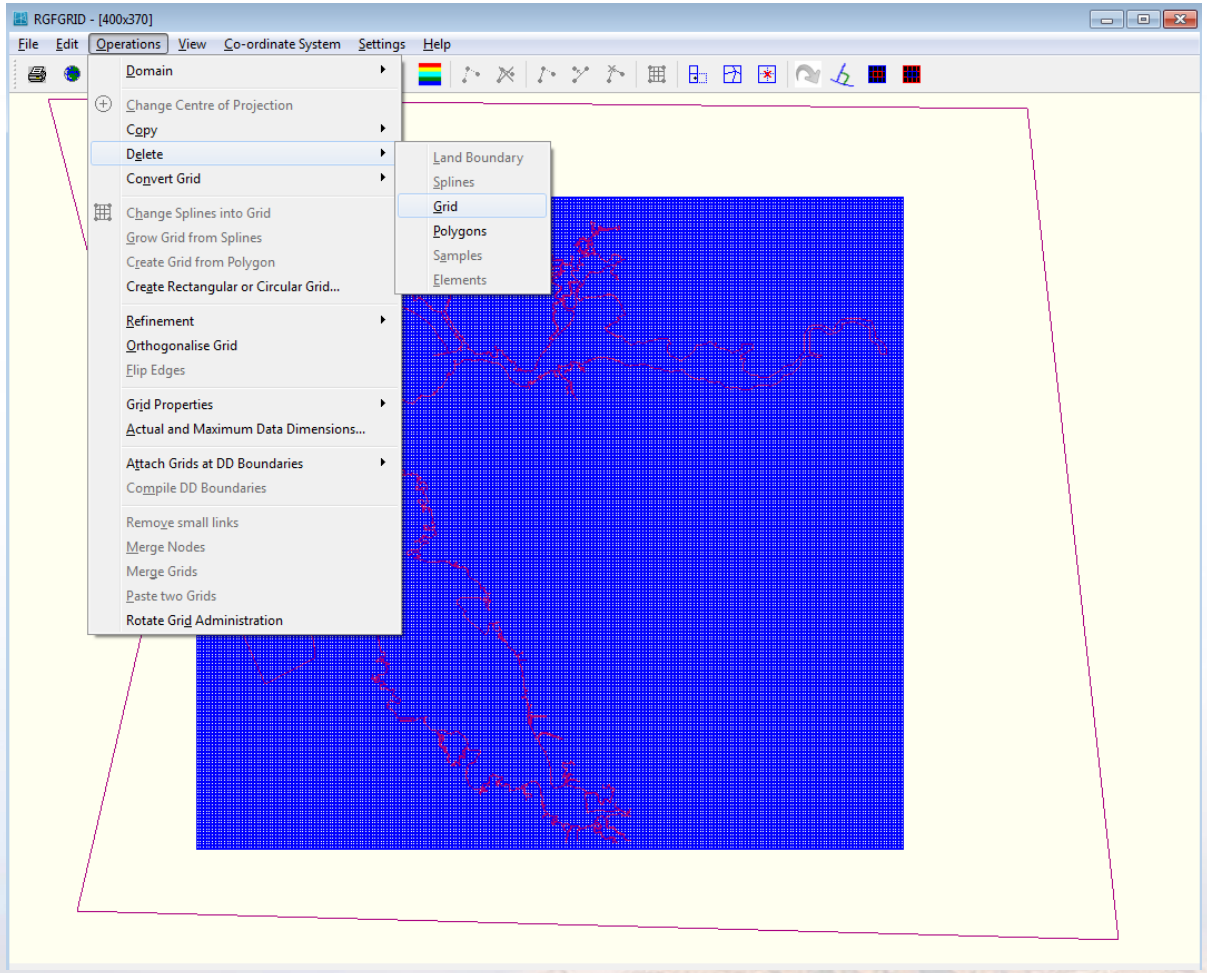
- Select “SFShoreline2.pol”

RGFGRID



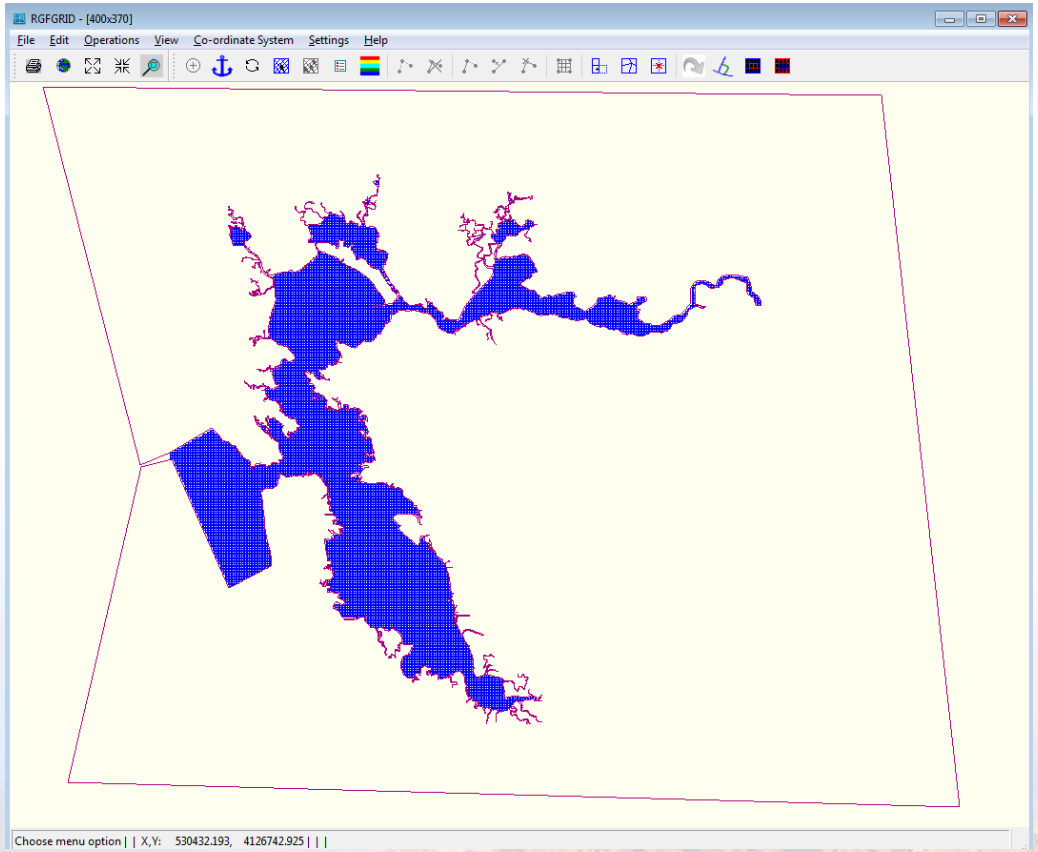
- This is the result.

RGFGRID



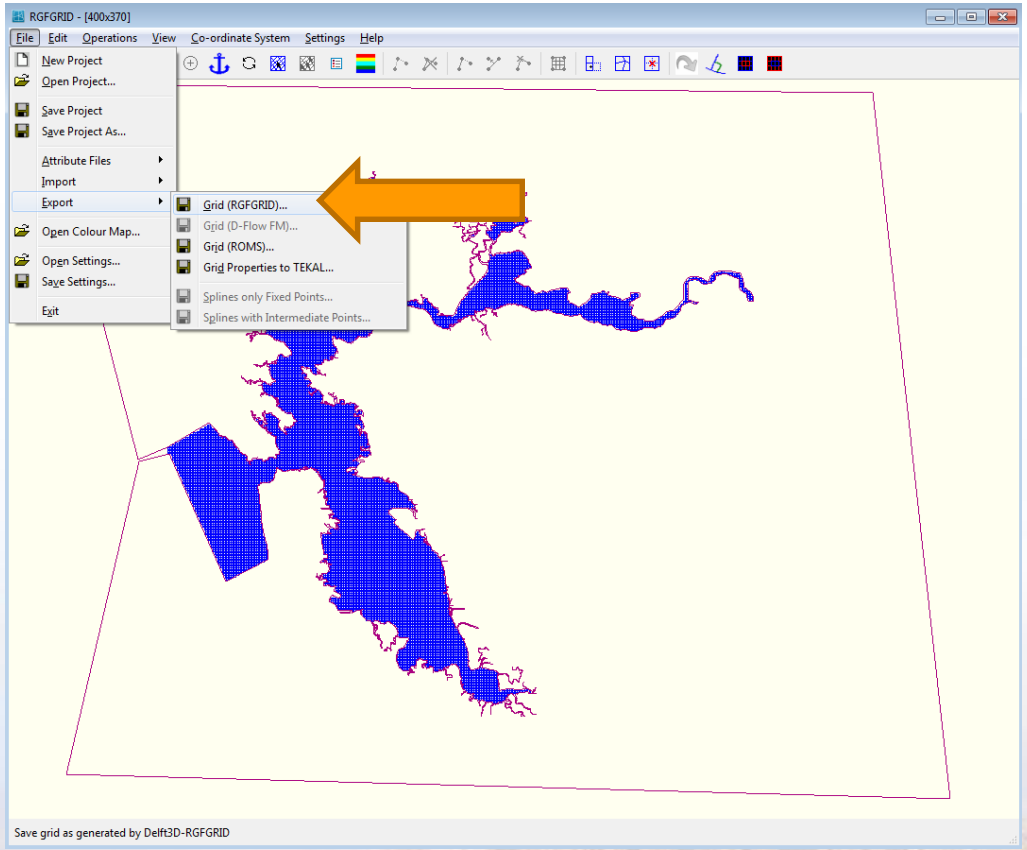
- Select Operations → Delete → Grid
- This will remove all points “inside” of the polygon.

RGFGRID



- This is the result.

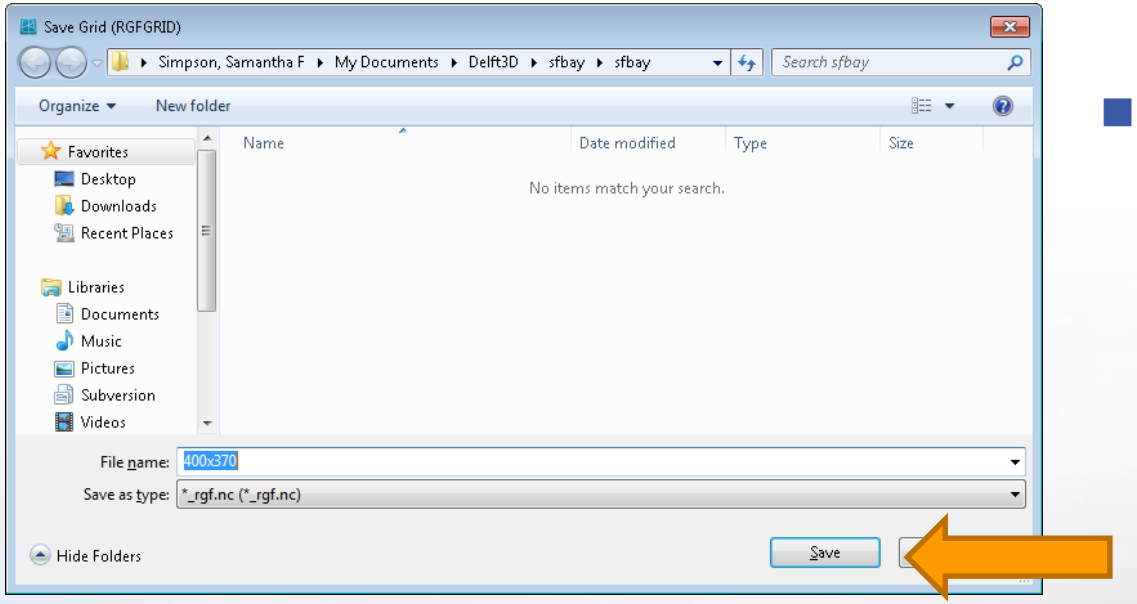
RGFGRID



- Select File→ Export → RGFGRID

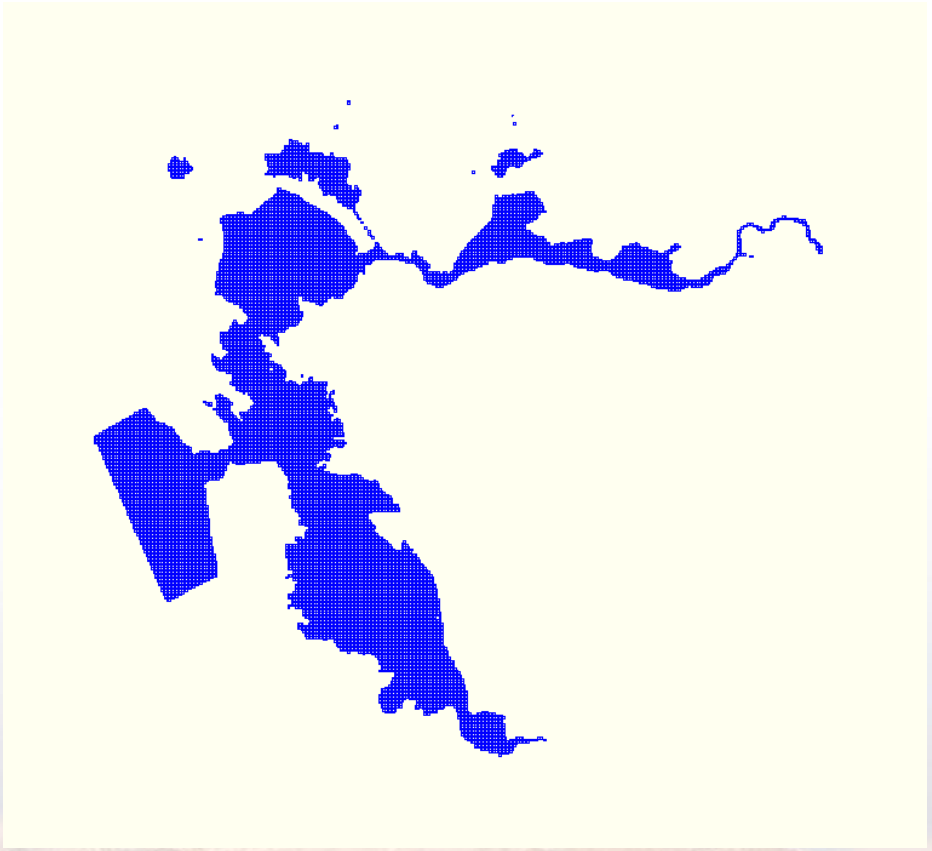
RGFGRID

- Save the grid file as “400x370.grd”.
- Close RGFGRID and import it again.



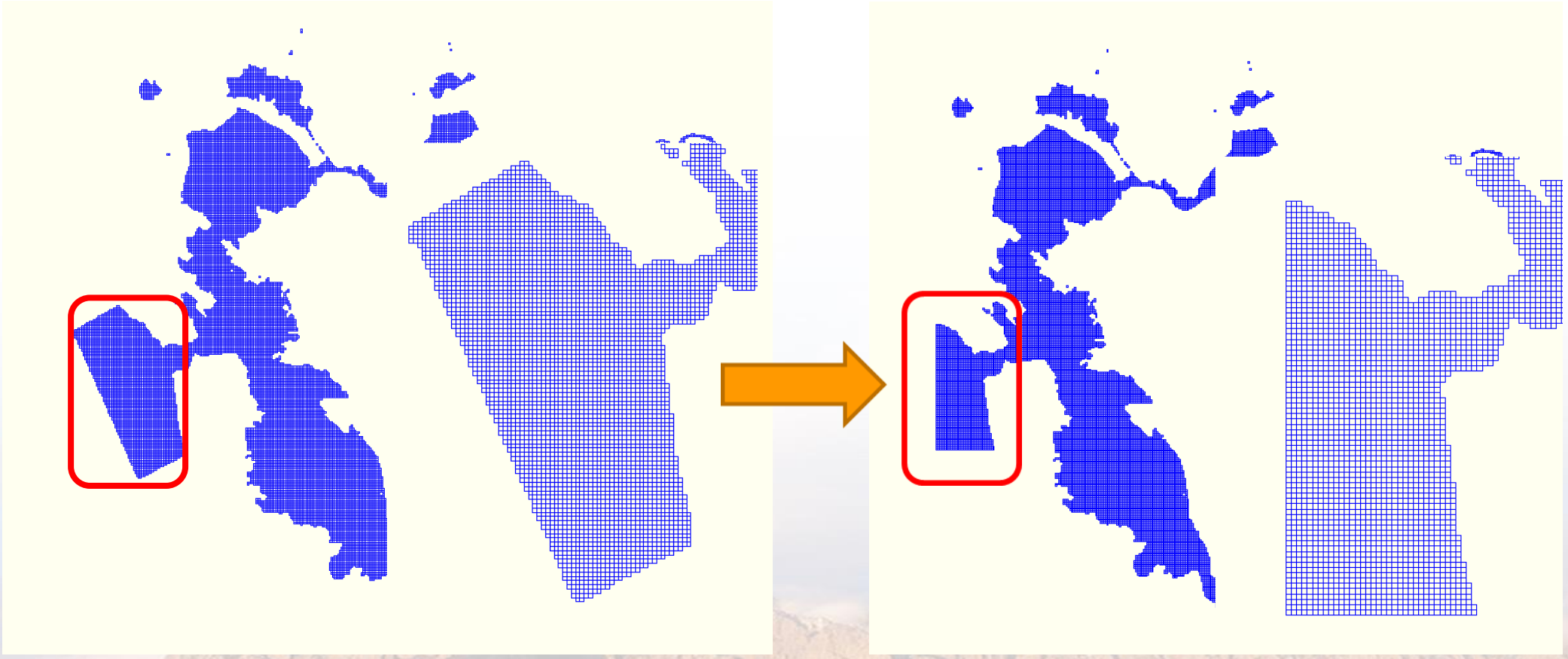
Editing the Grid

- Import the newly created grid.
- In order to make creating the boundary easier, grid cells will be deleted to allow for smooth boundary edges instead of jagged.



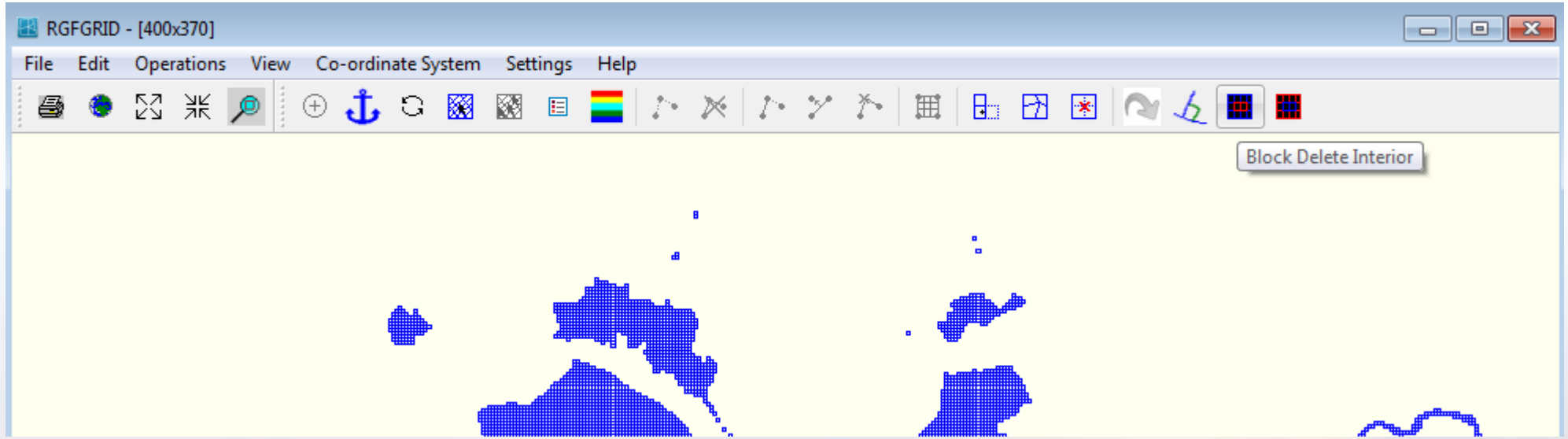
Editing the Grid

The end goal is to transform the boxed region on the left to the one on the right.

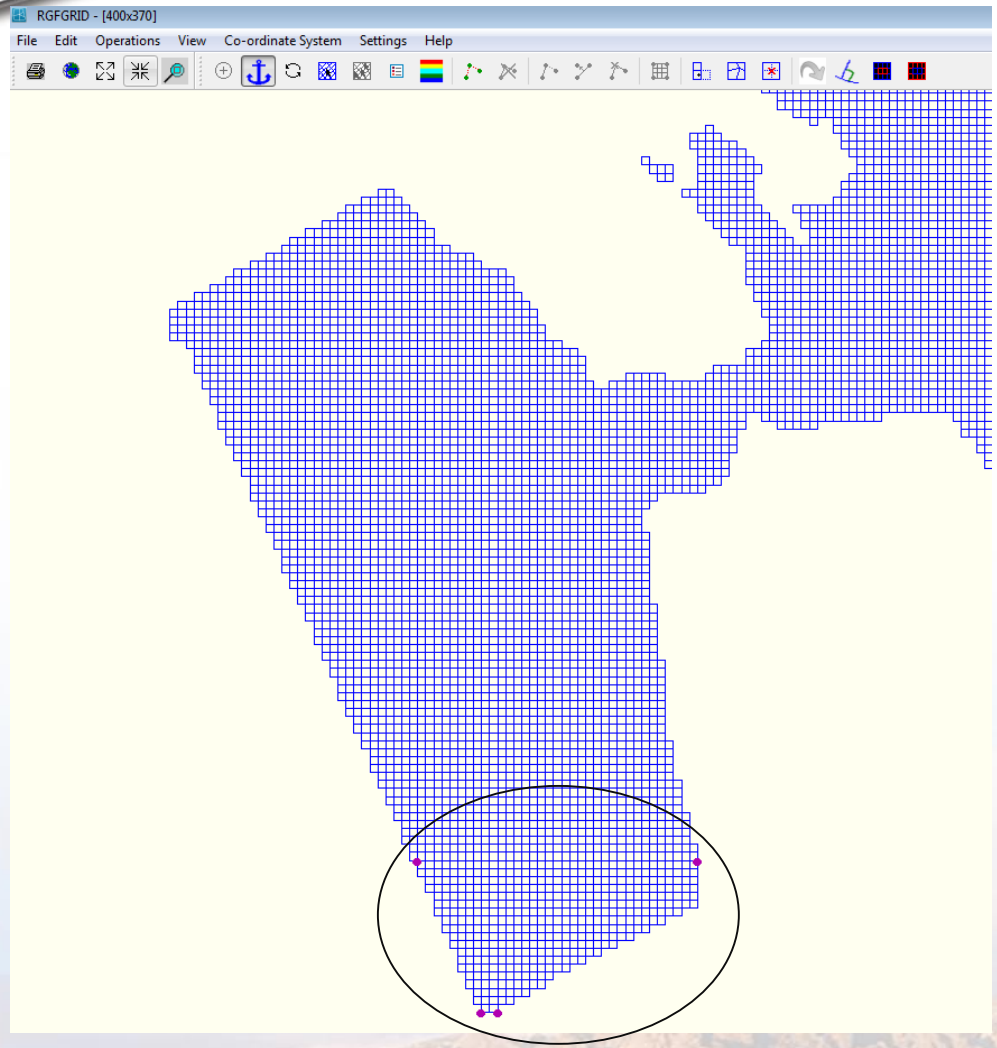


Creating a Linear Boundary

- Remove parts of the grid with the “Block Delete Interior” tool.
- Use the tool to delete grid cells from the bottom, left side, and top of the SF Bay boundary

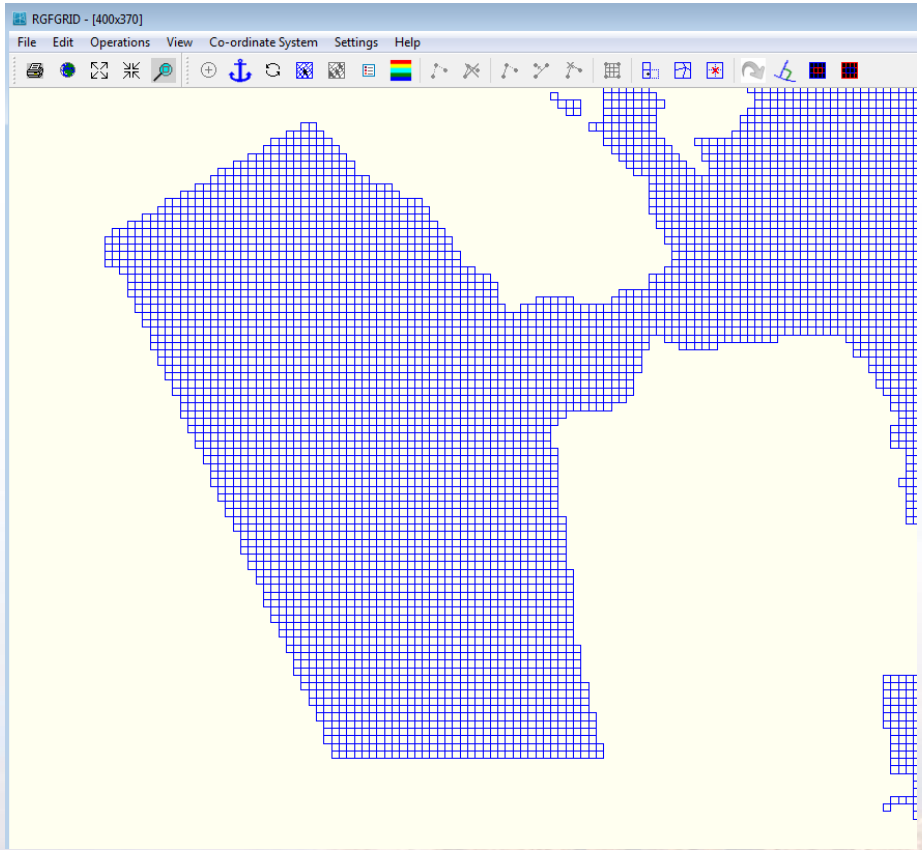


Creating a Linear Boundary



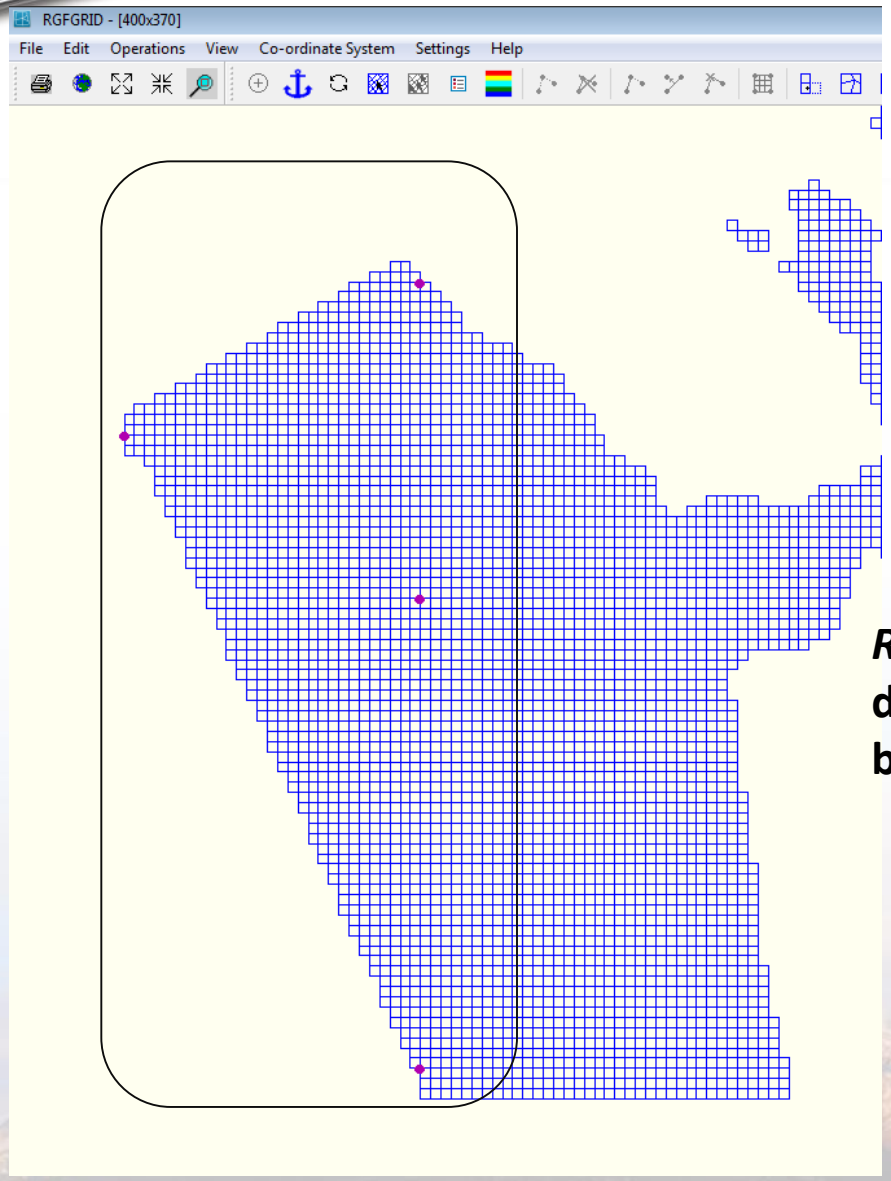
- Create a block by selecting four points on the grid.
- Have top points be on the same line, and have bottom along the bottom corner of the grid.
- Right click to delete the grid points within the selected grid points.

Creating a Linear Boundary

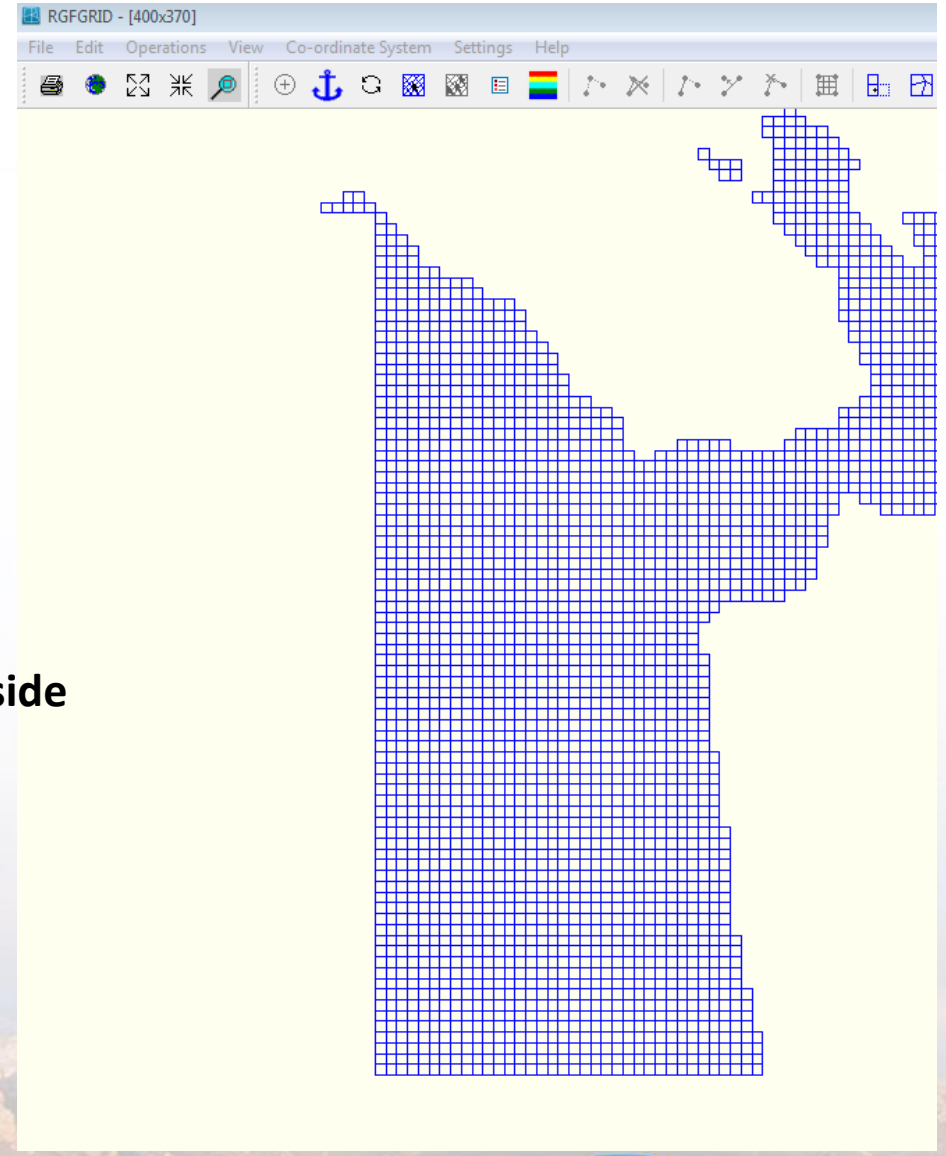


- The bottom should now be smooth.
- Repeat process for left side and top.

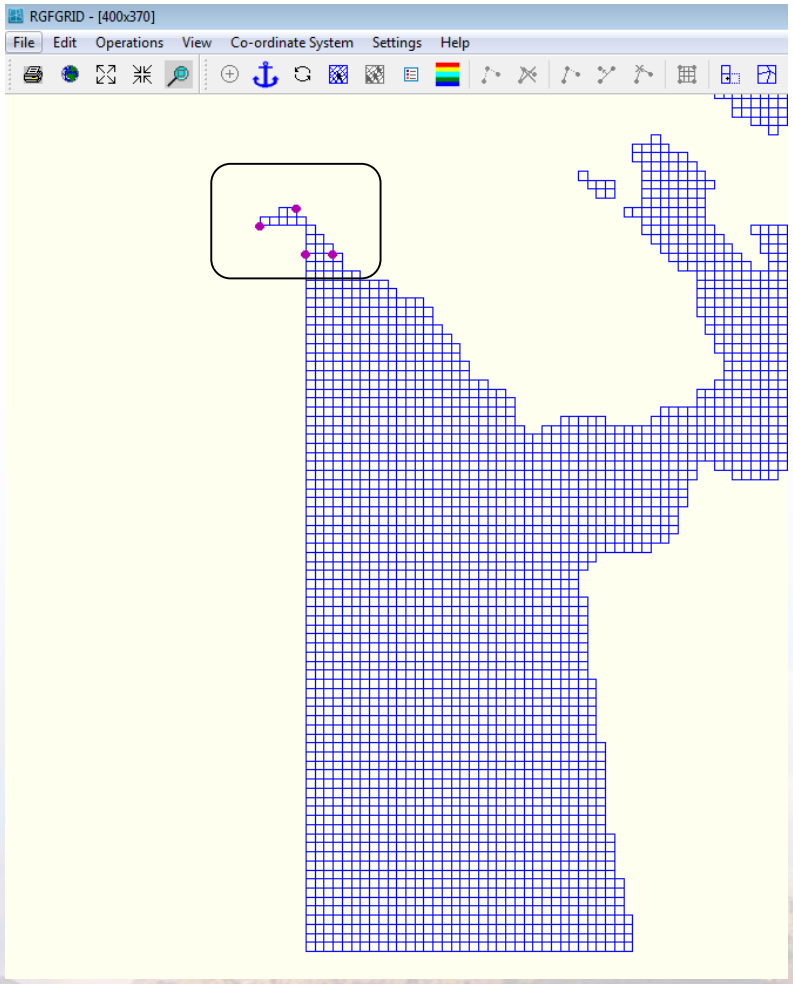
Delete Left cells



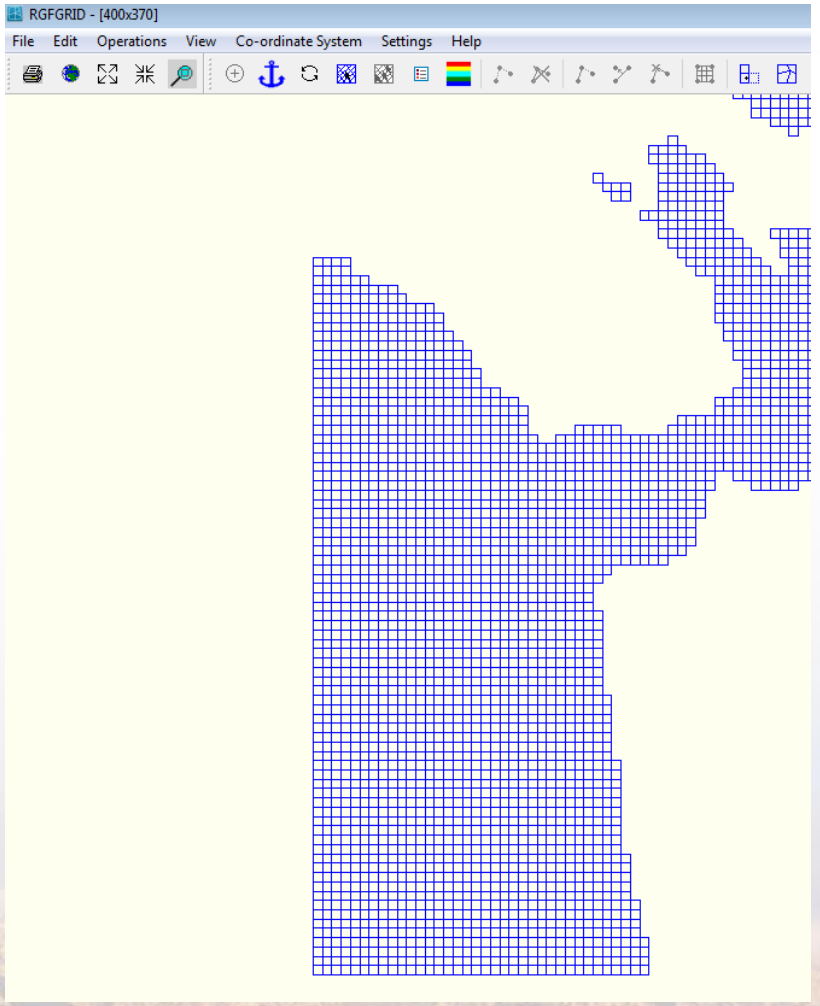
Right click **to delete cells inside blocked area.**



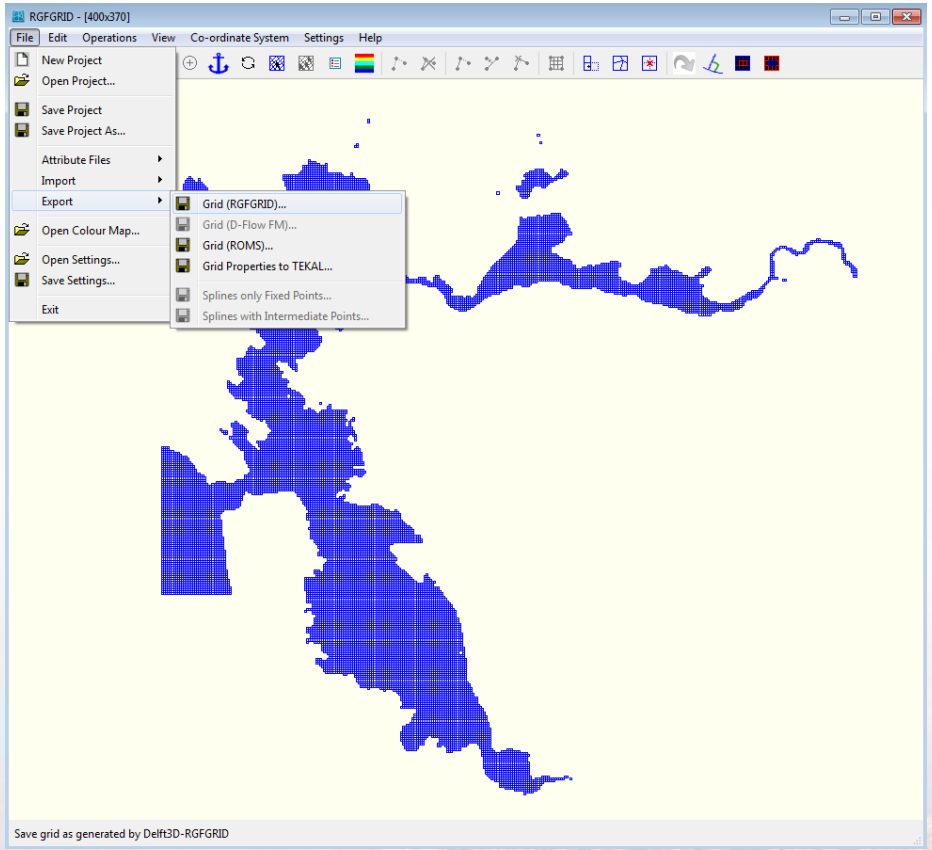
Delete Cells on top



Right click **to delete cells inside blocked area.**

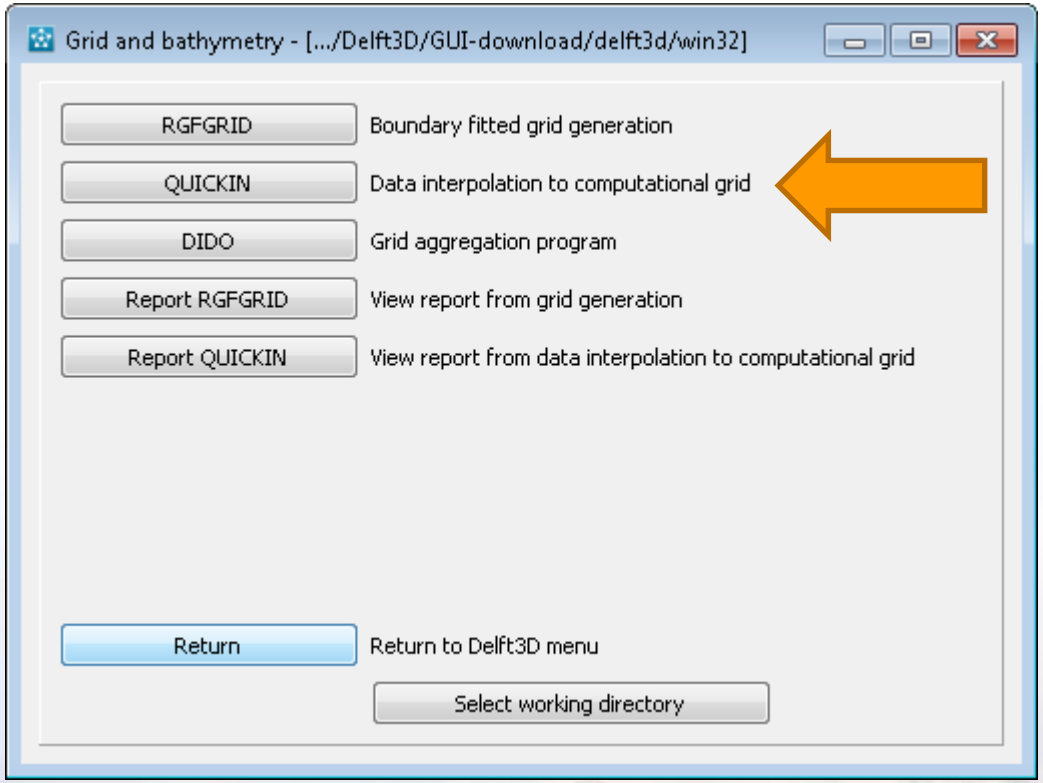


Saving Your Grid



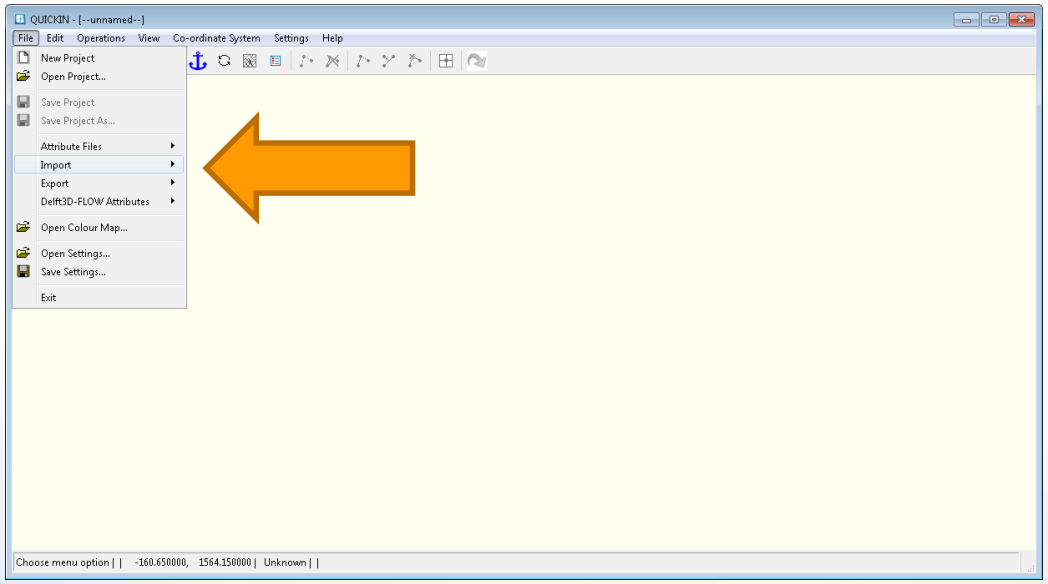
- **Export the completed grid as “400x370.grd”**

QUICKIN



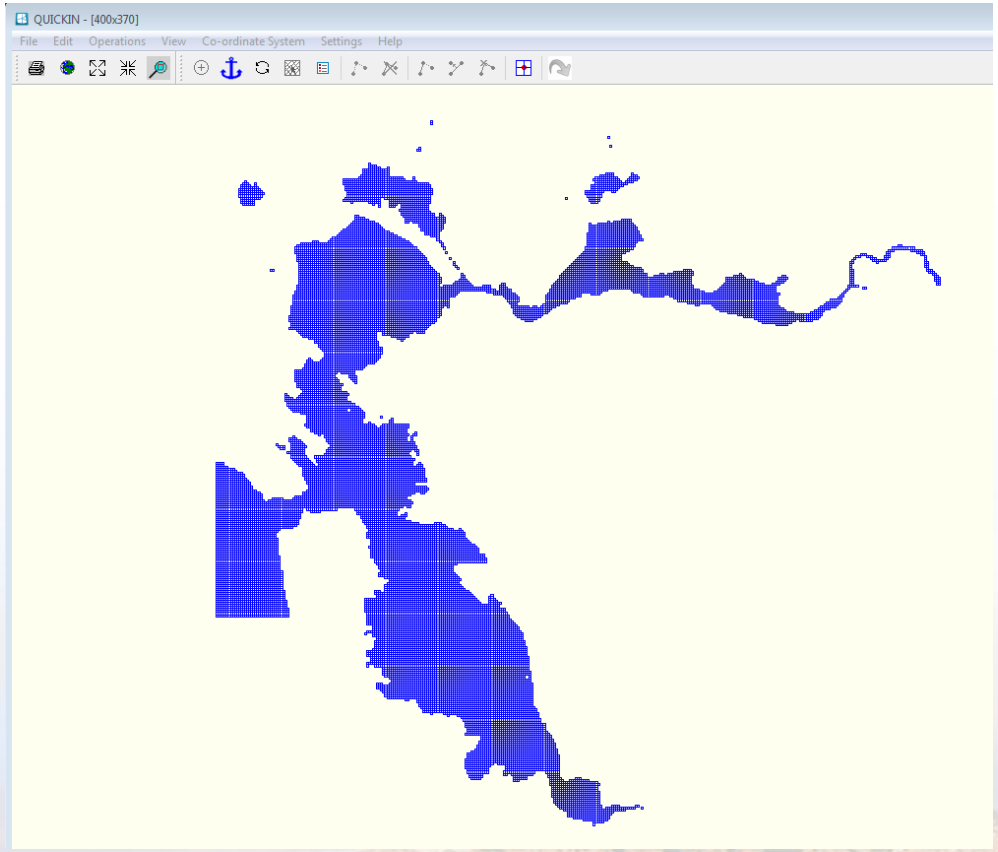
- Exit RGFGRID and return to the Grid and Bathymetry menu.
- Select QUICKIN

QUICKIN



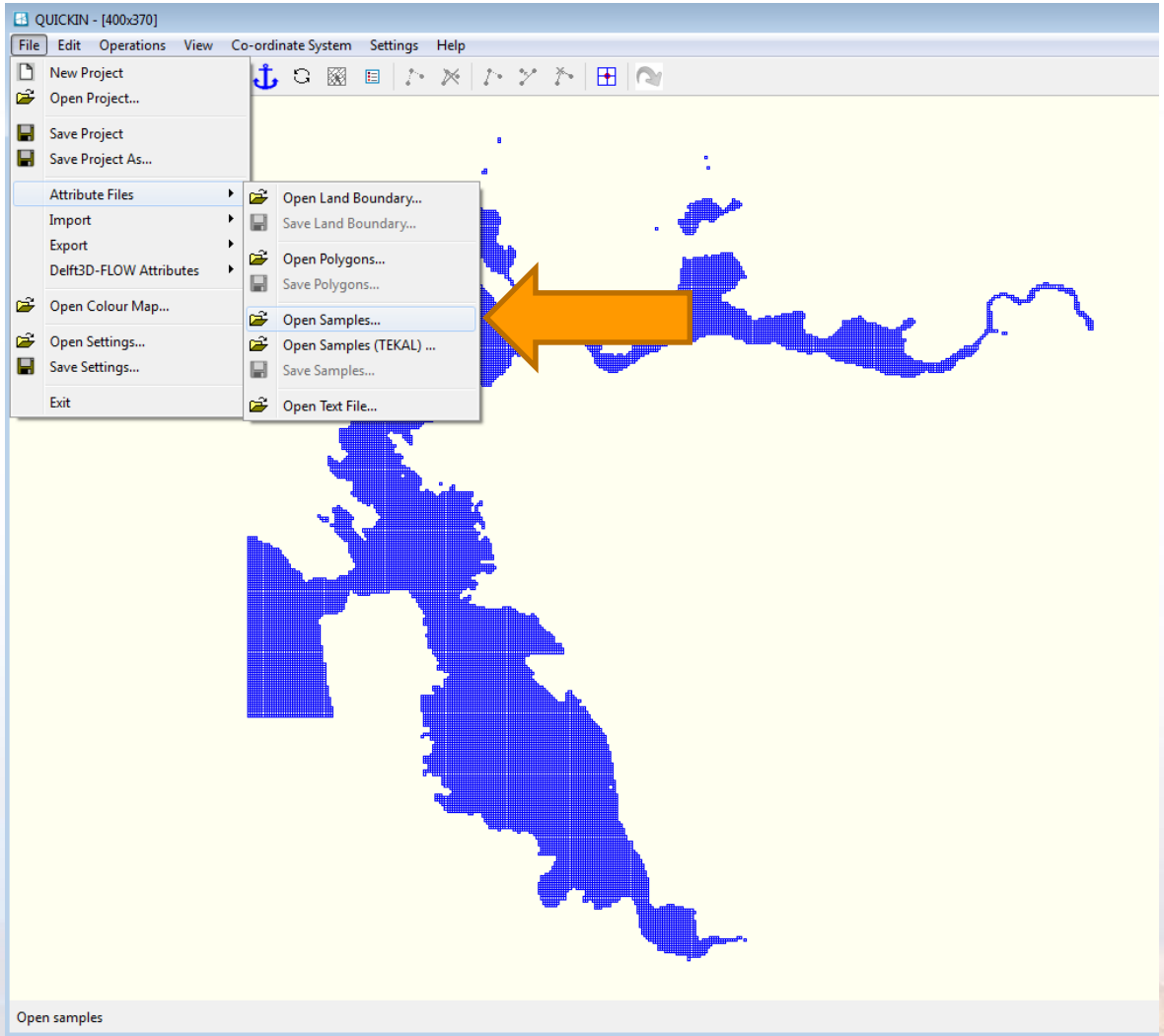
- Select File → Import → RGFGRID and open the file that we just created.
- QUICKIN will be used to add attribute files and interpolate and extrapolate sample data.

QUICKIN



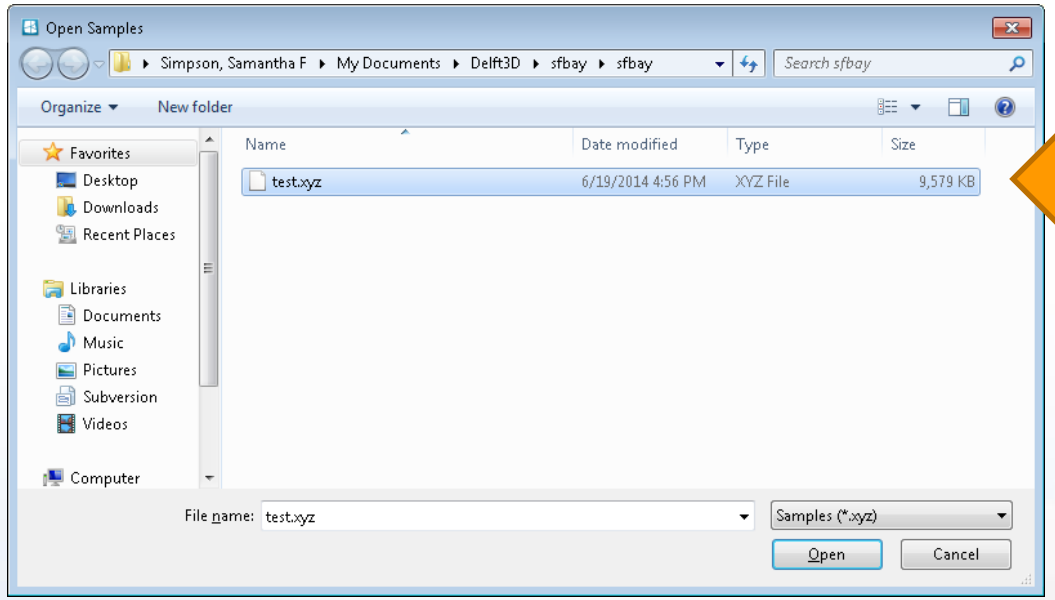
- This is what you will see.

QUICKIN



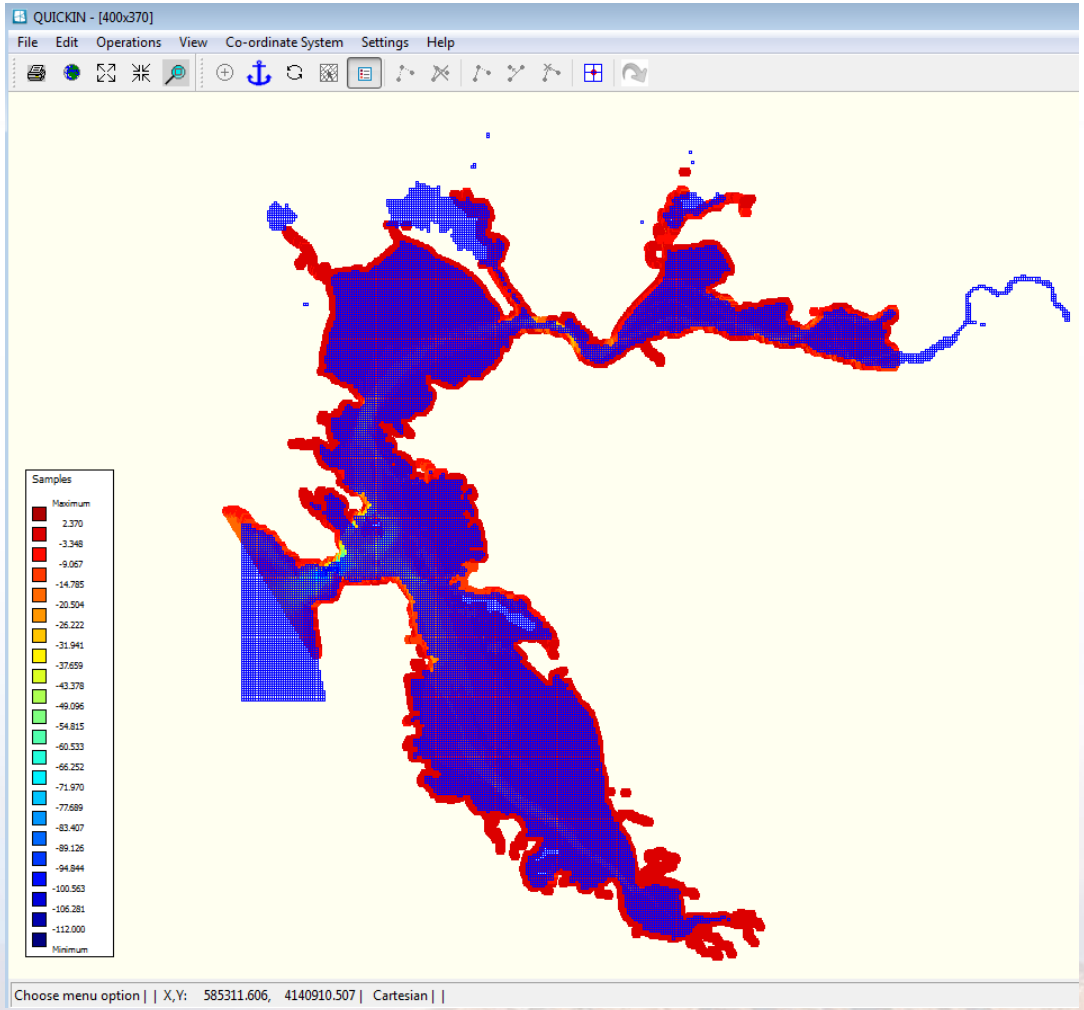
- Select File → Attribute Files → Open Samples

Adding a Sample File



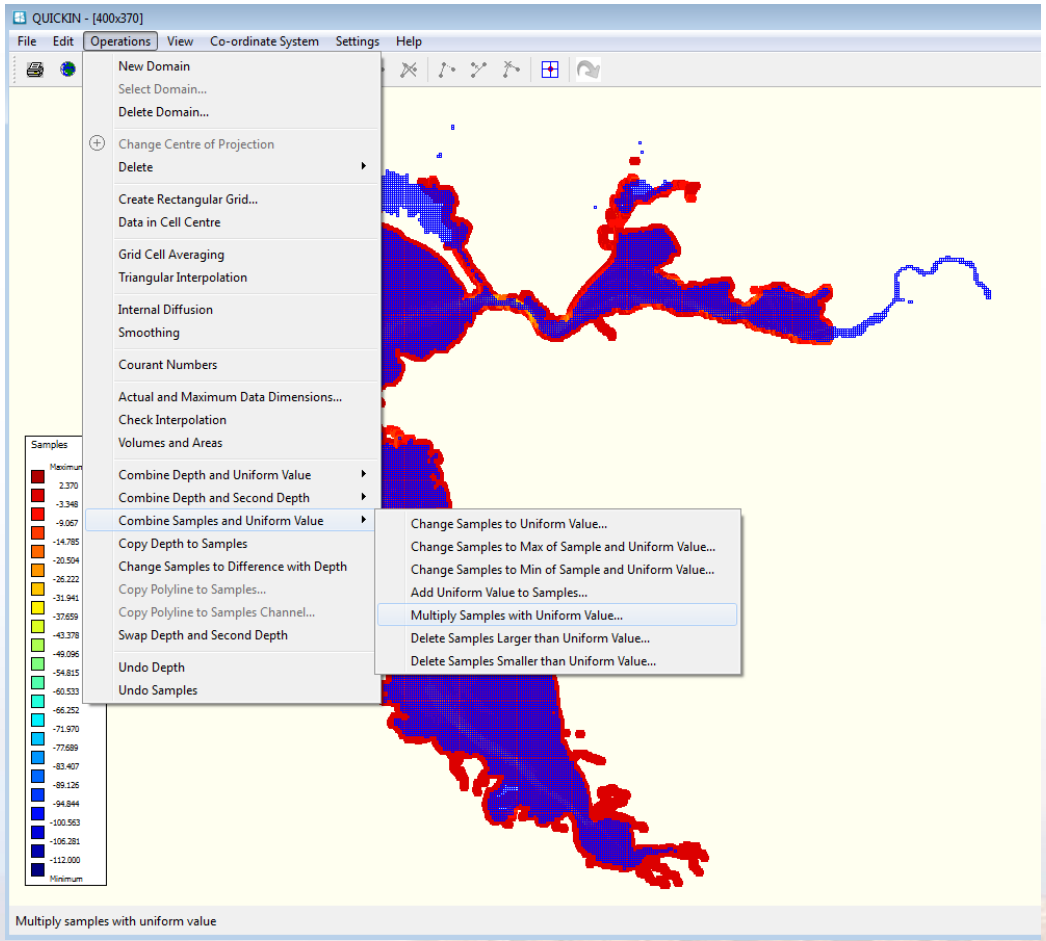
- **Select test.xyz**
- **test.xyz contains data samples at each (x,y) coordinate on the grid.**

Adding a Sample File



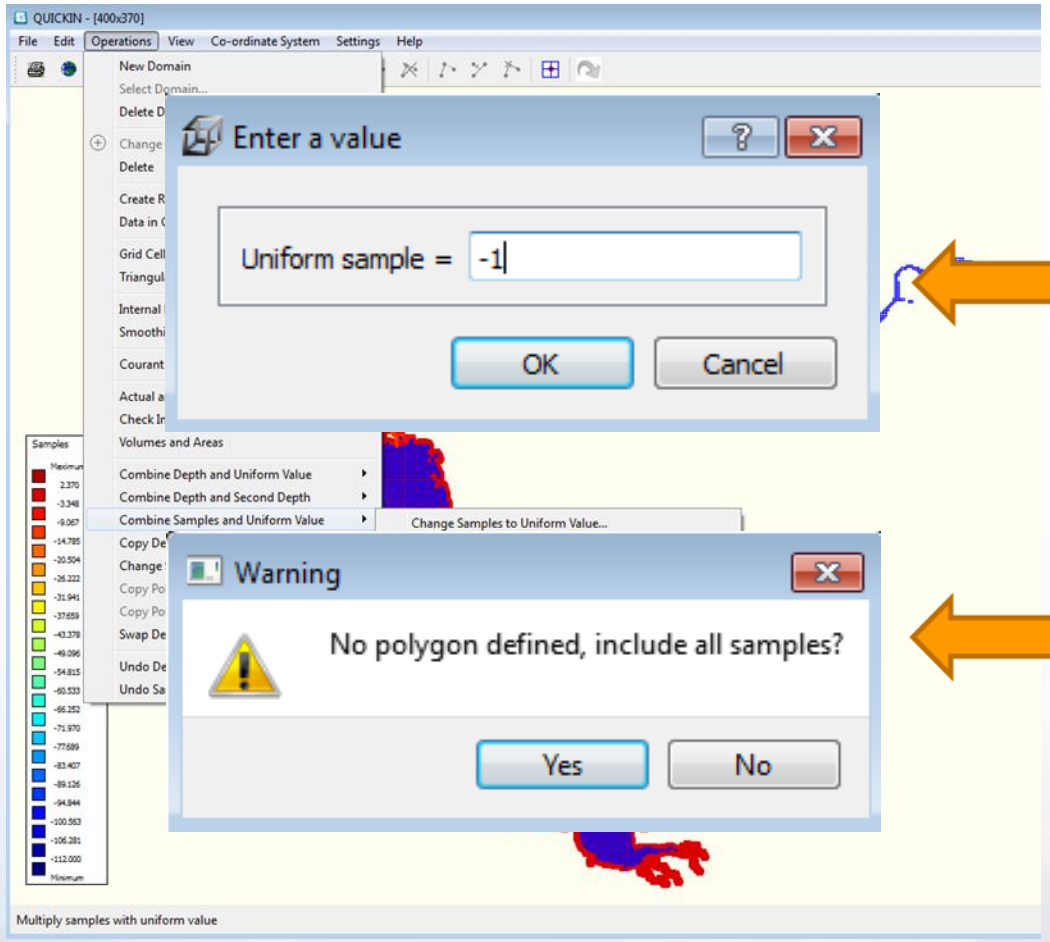
- This is the result. Notice the samples (z-values) range from 2.37 to -112.

Adding a Sample File



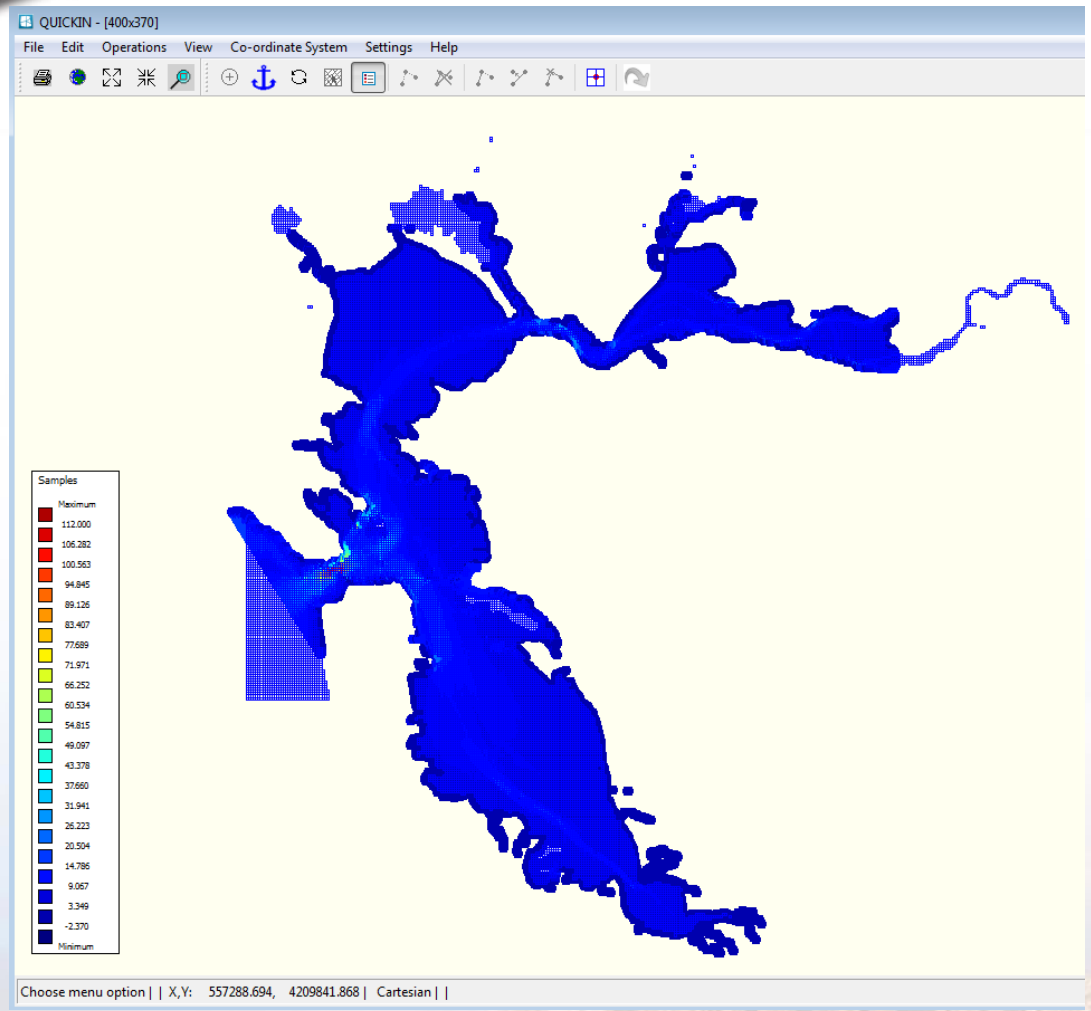
- Flip the signs of the sample values
- Select Operations → Combine Samples and Uniform Value → Multiply Samples with Uniform Value…

Adding a Sample File



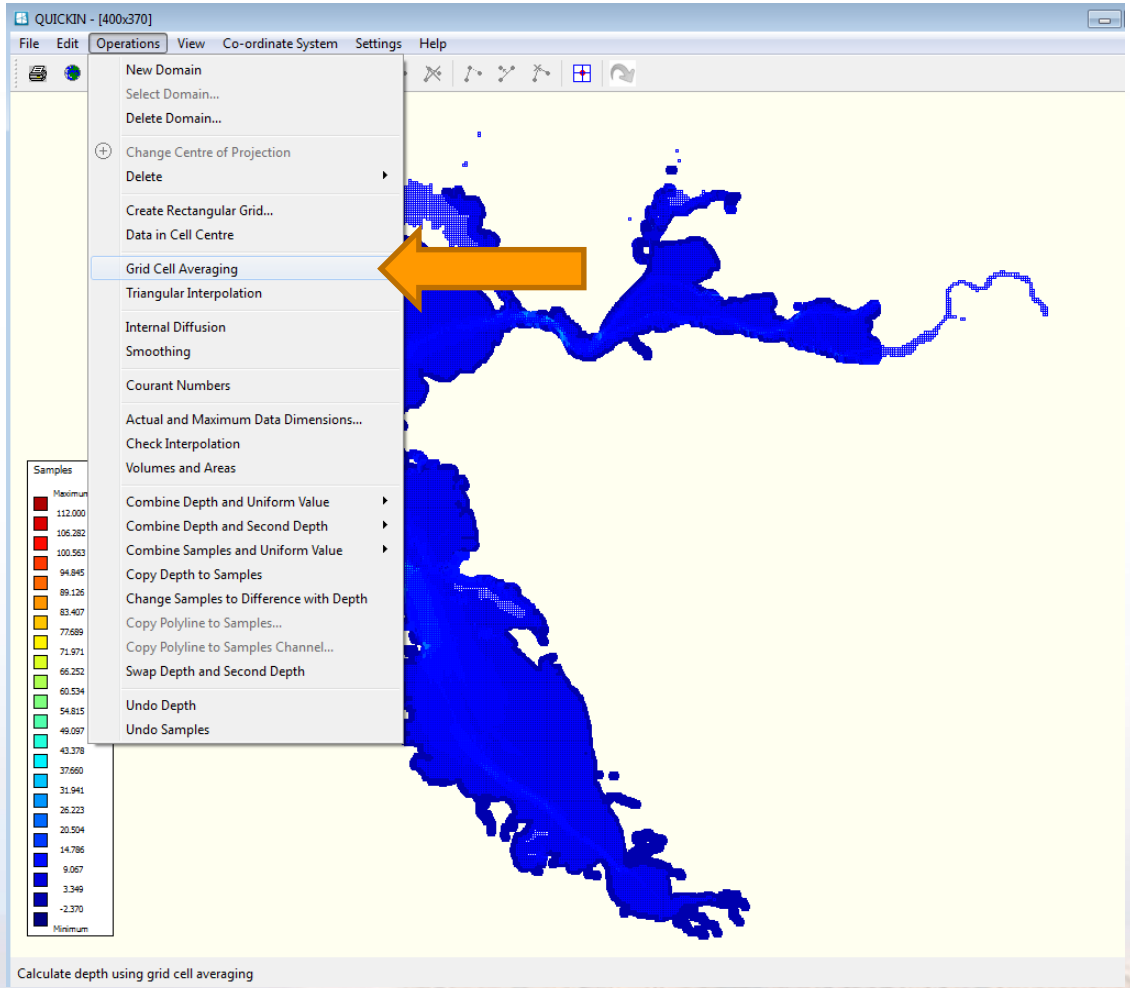
- Change from -999 to -1, click OK.
- A warning window will pop up. Click Yes.

QUICKIN



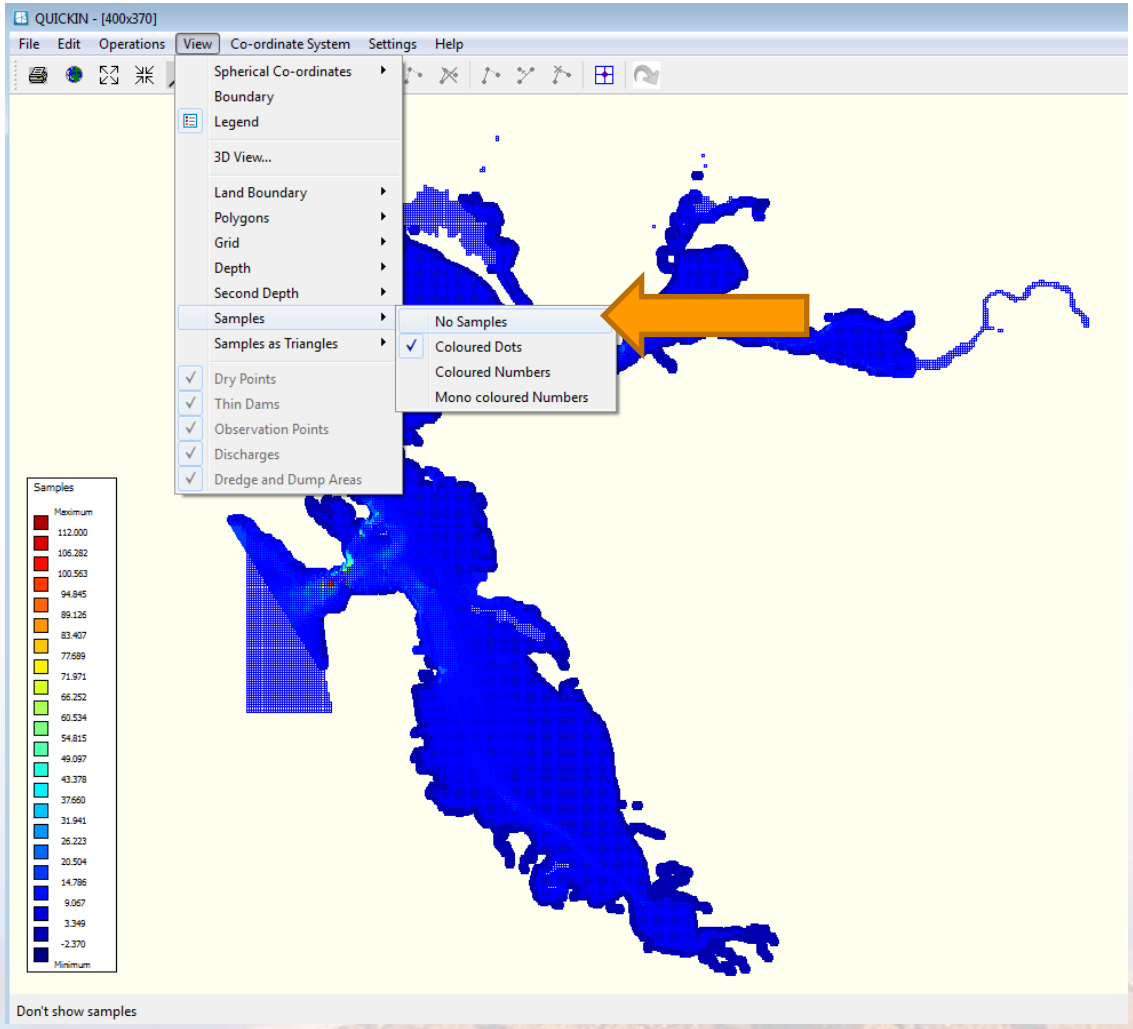
- This is the result.
- Notice the difference in the sample values from before.

Grid Cell Averaging



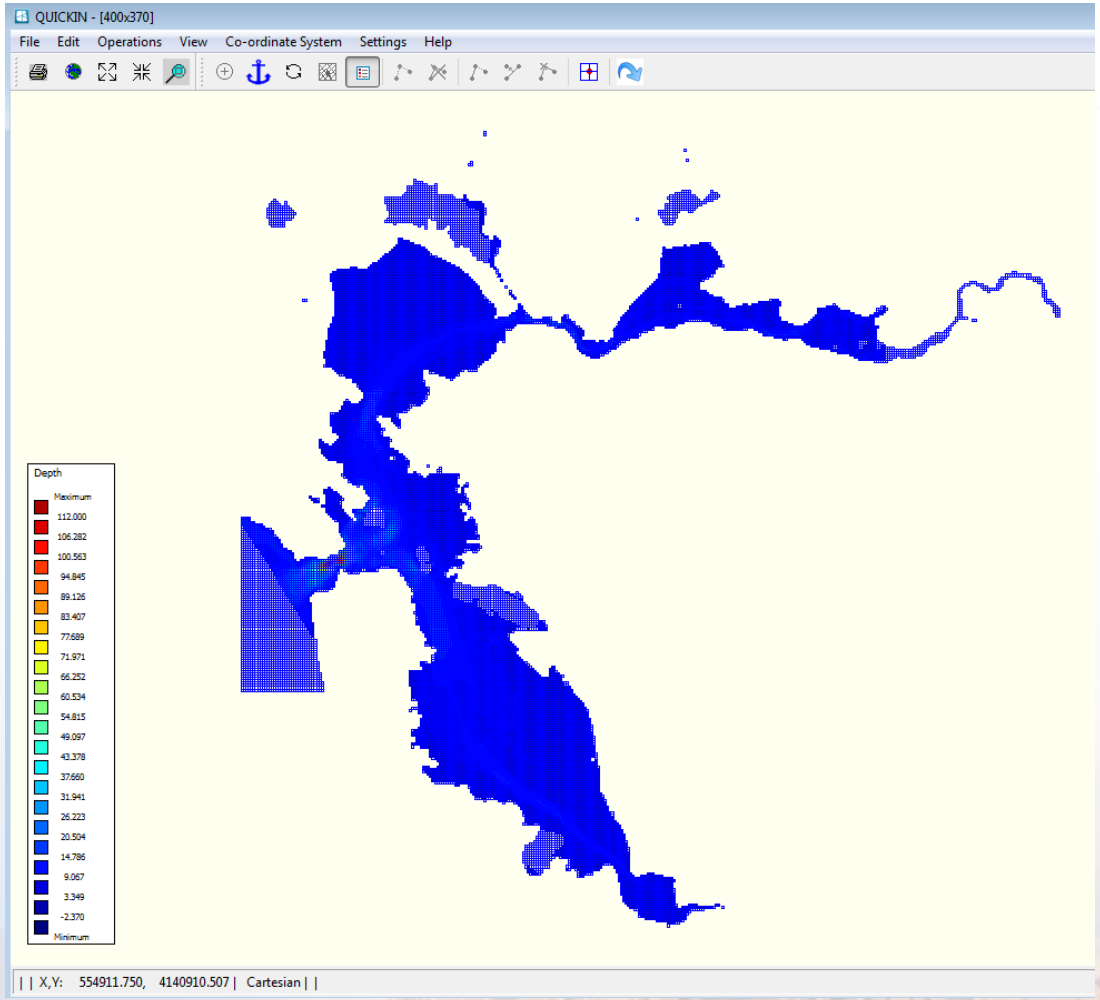
- Select Operations → Grid Cell Averaging
- This will interpolate the samples on to the grid.

Sample Removal



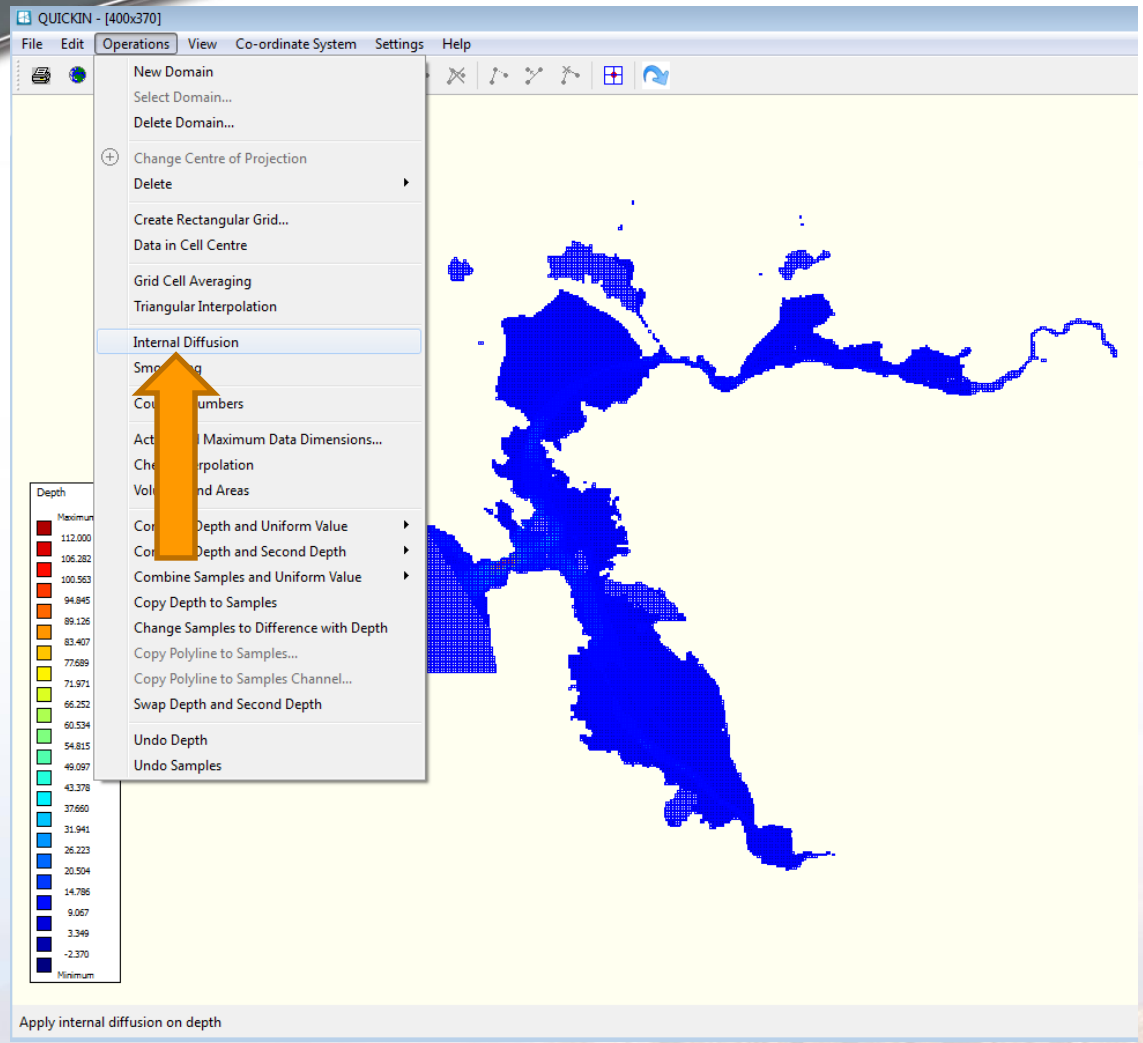
- **Select View → Samples → No Samples**

Sample Removal



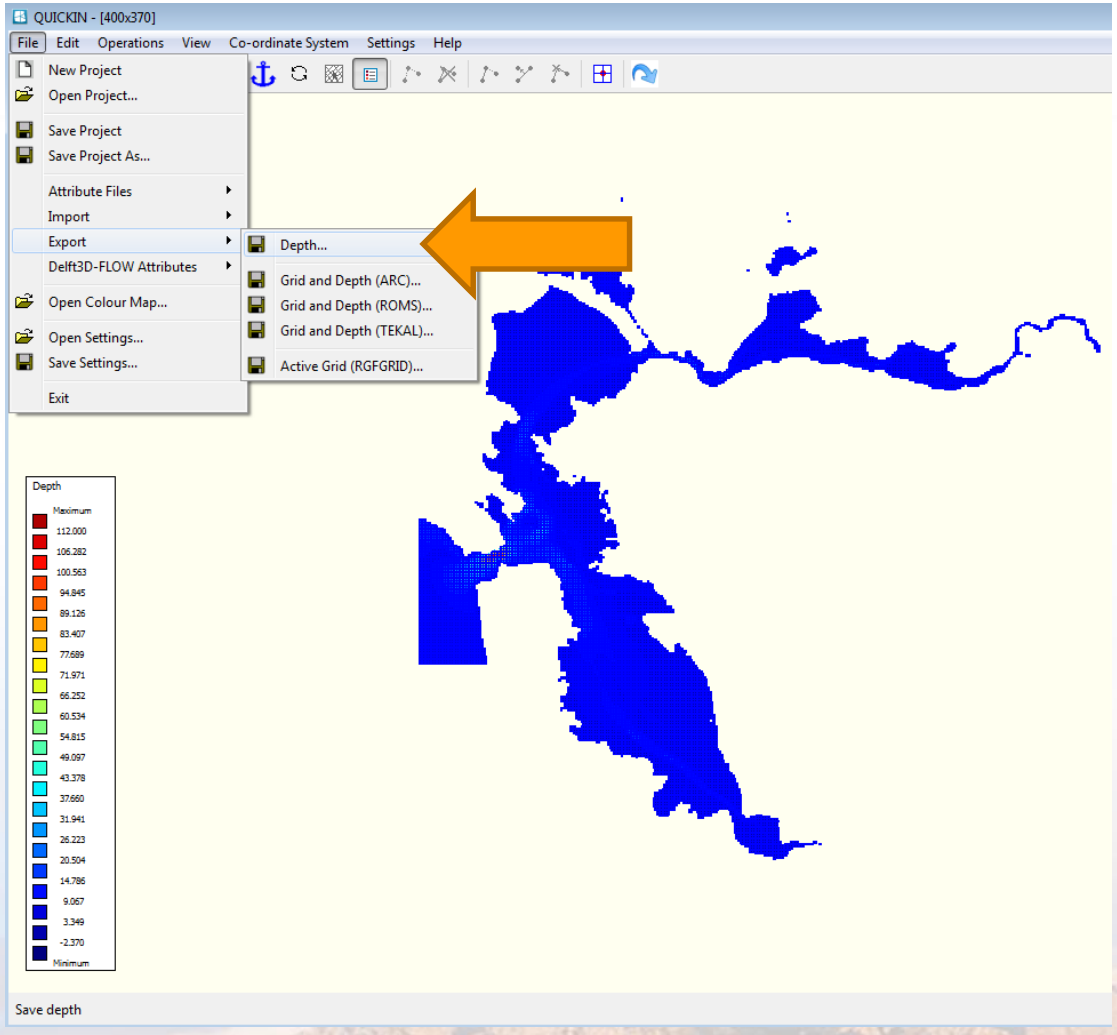
- **The removal of the contour indicates that the interpolation was successful.**

QUICKIN



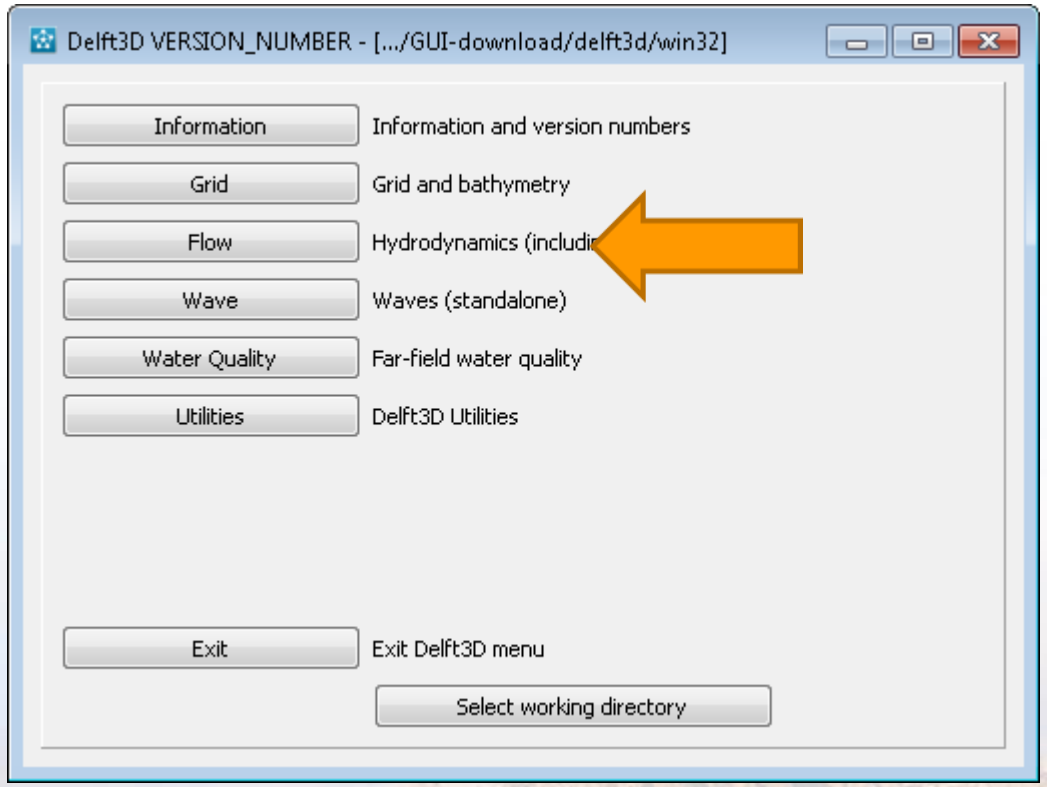
- Select Operations → Internal Diffusion
- This will extrapolate the depths to the remaining unpopulated cells.

QUICKIN



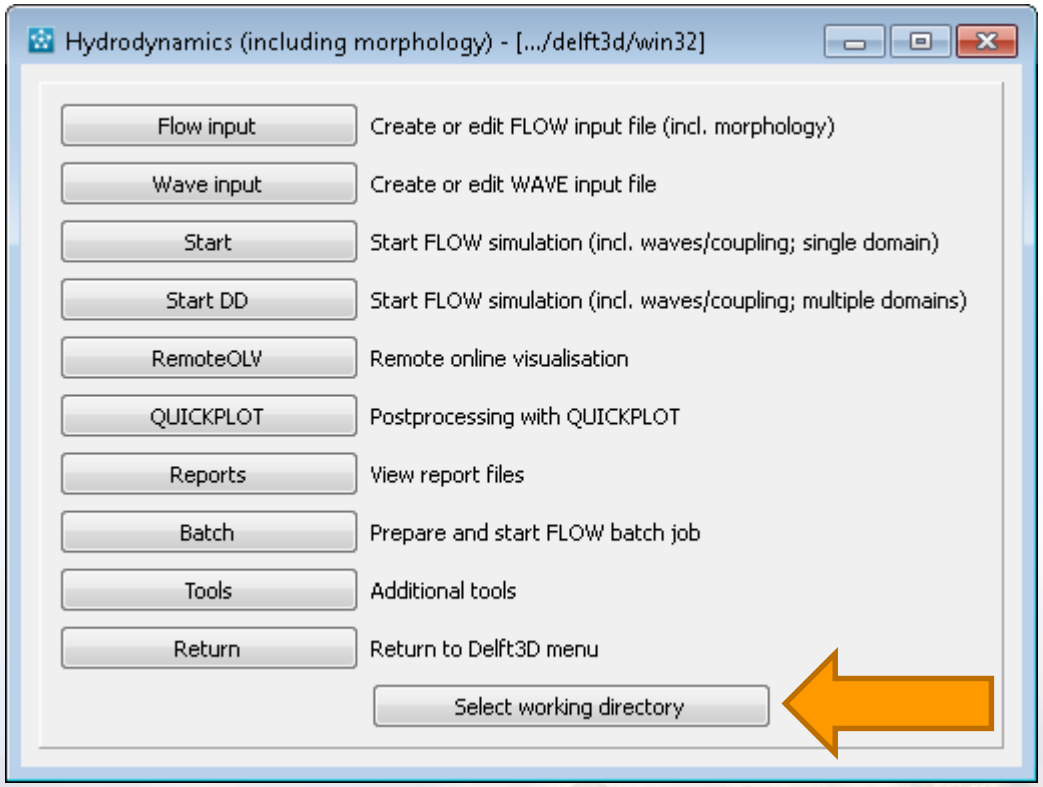
- Select File → Export → Depth
- Save as sfbay.dep
- This depth file will be used in the Flow setup.

Flow



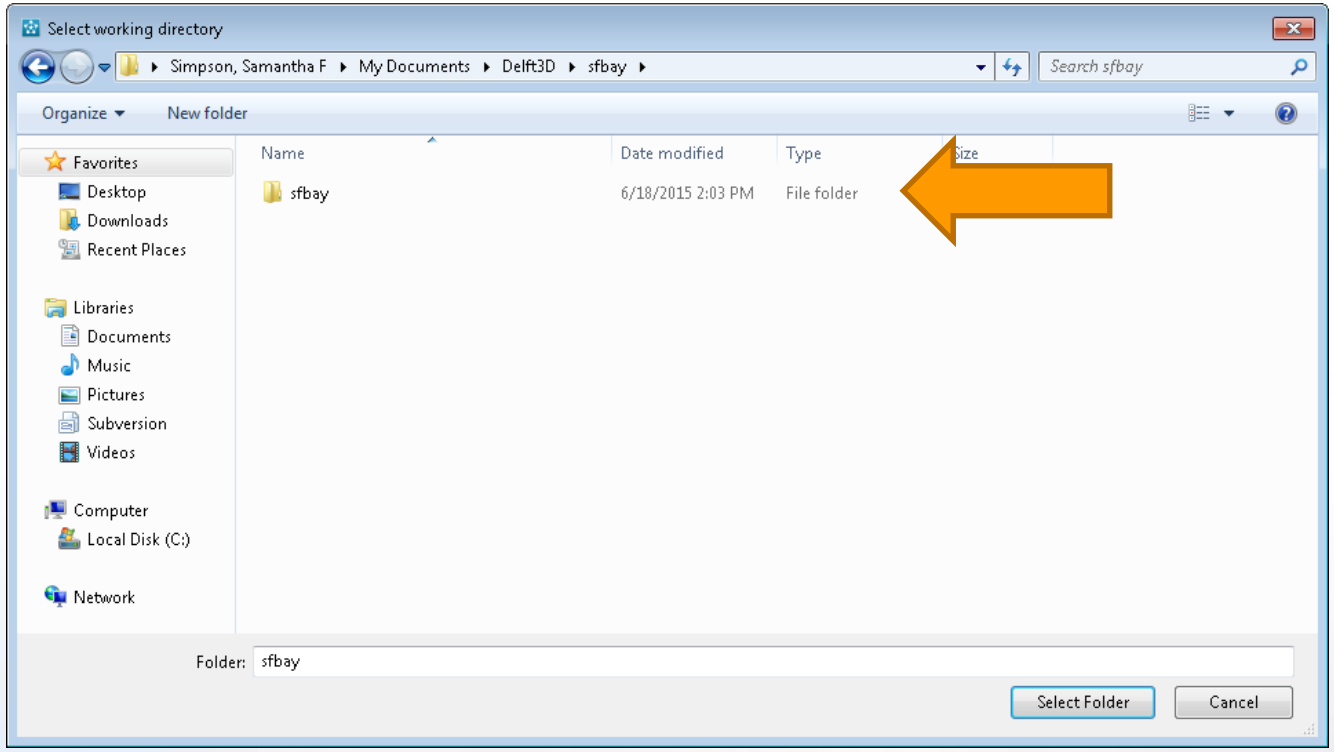
- Return to the main Delft 3D menu.
- Select Flow

Flow



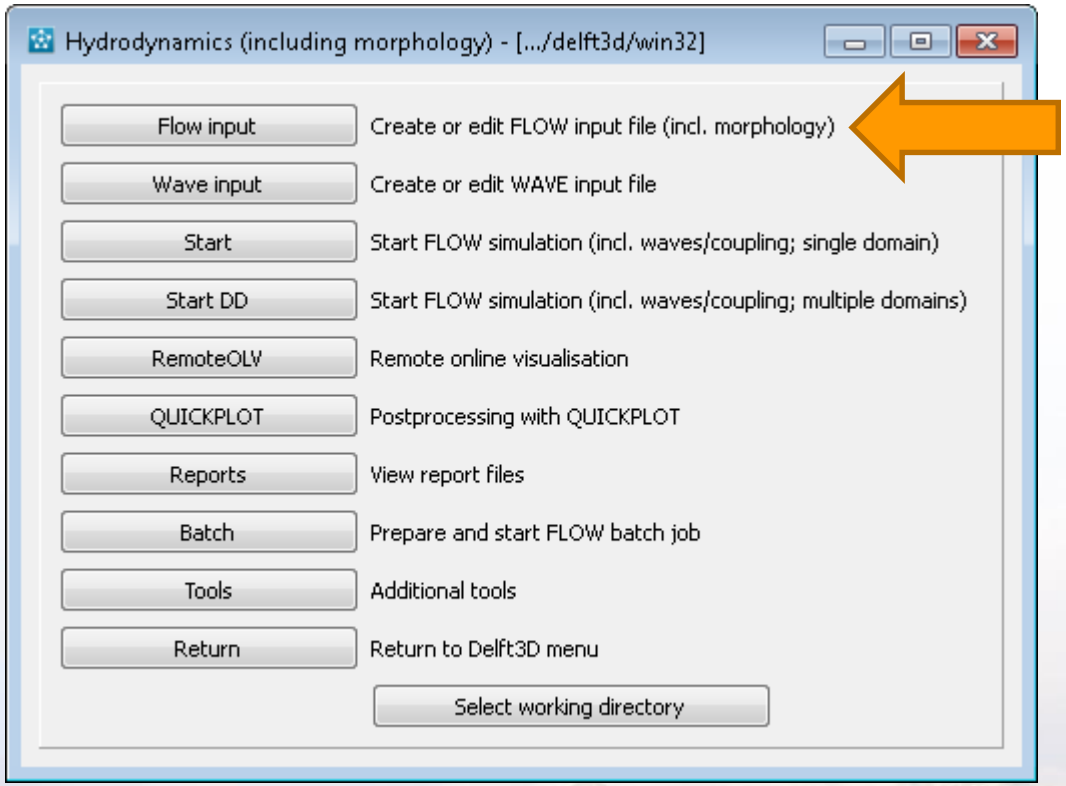
- **Select Working Directory**

Flow



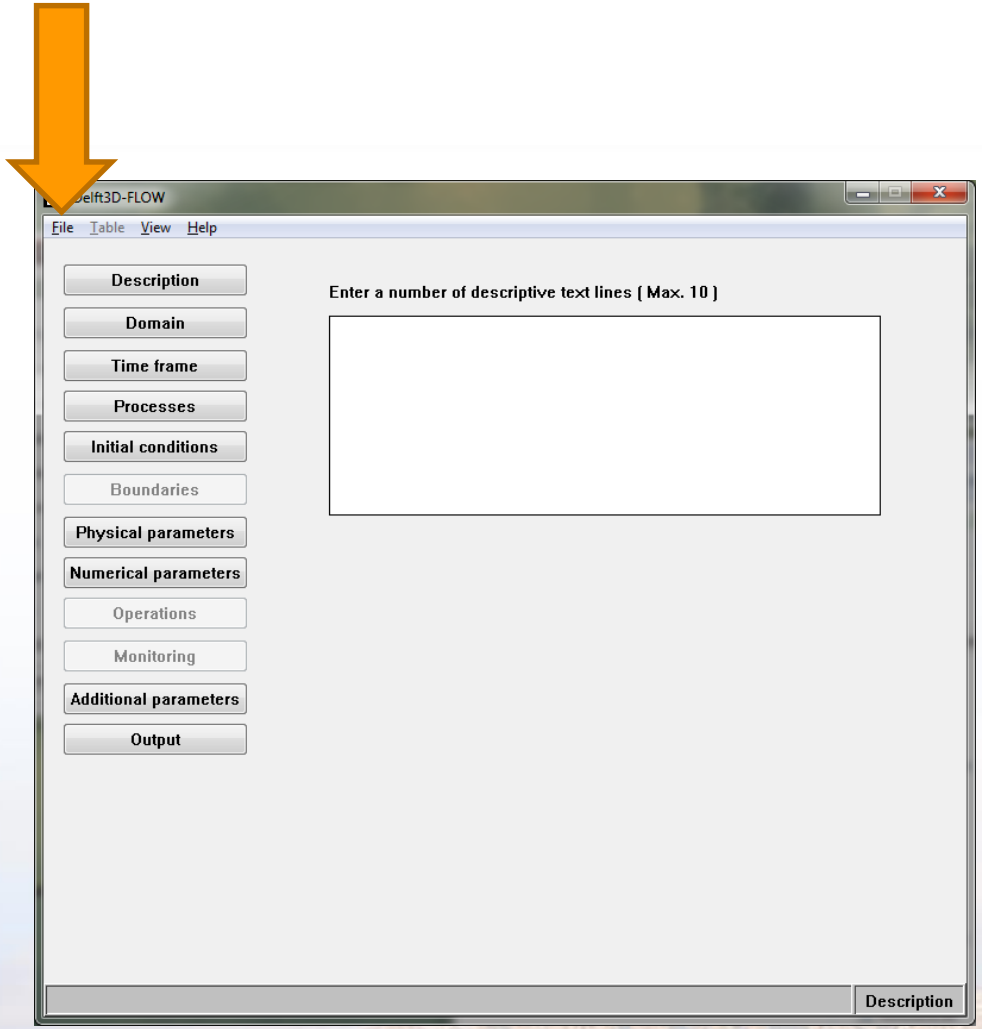
- Choose the folder currently being worked in.

Flow



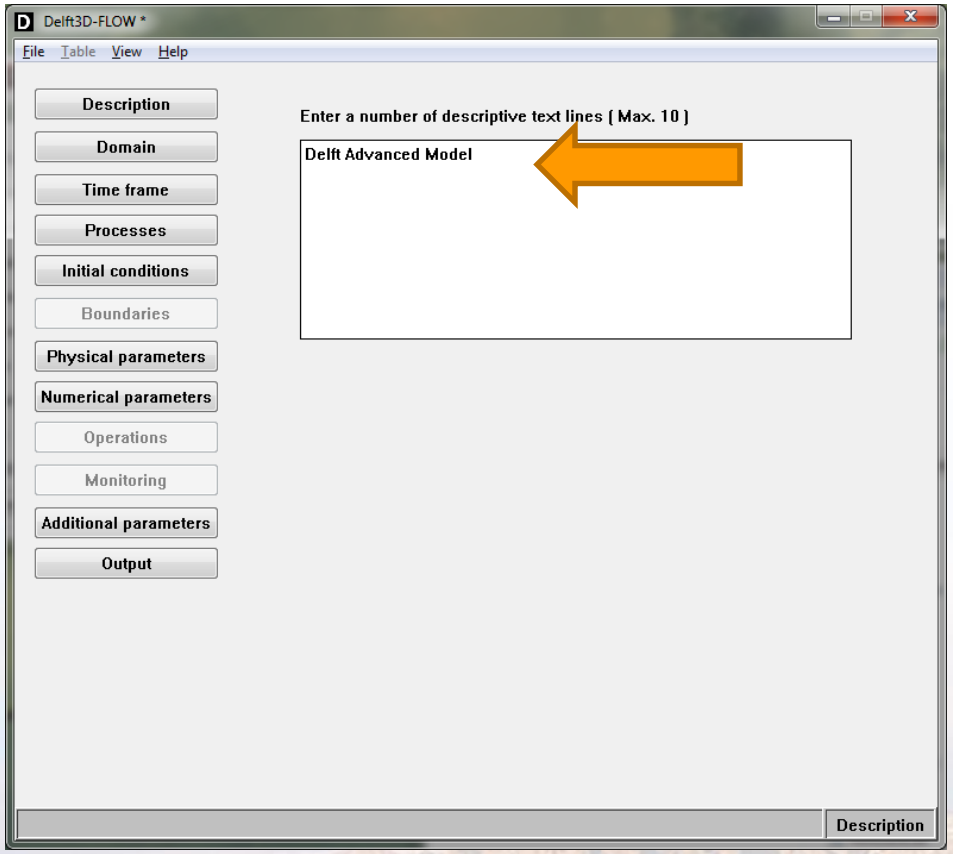
- **Return to the main menu and select Flow Input**

Delft3D-FLOW



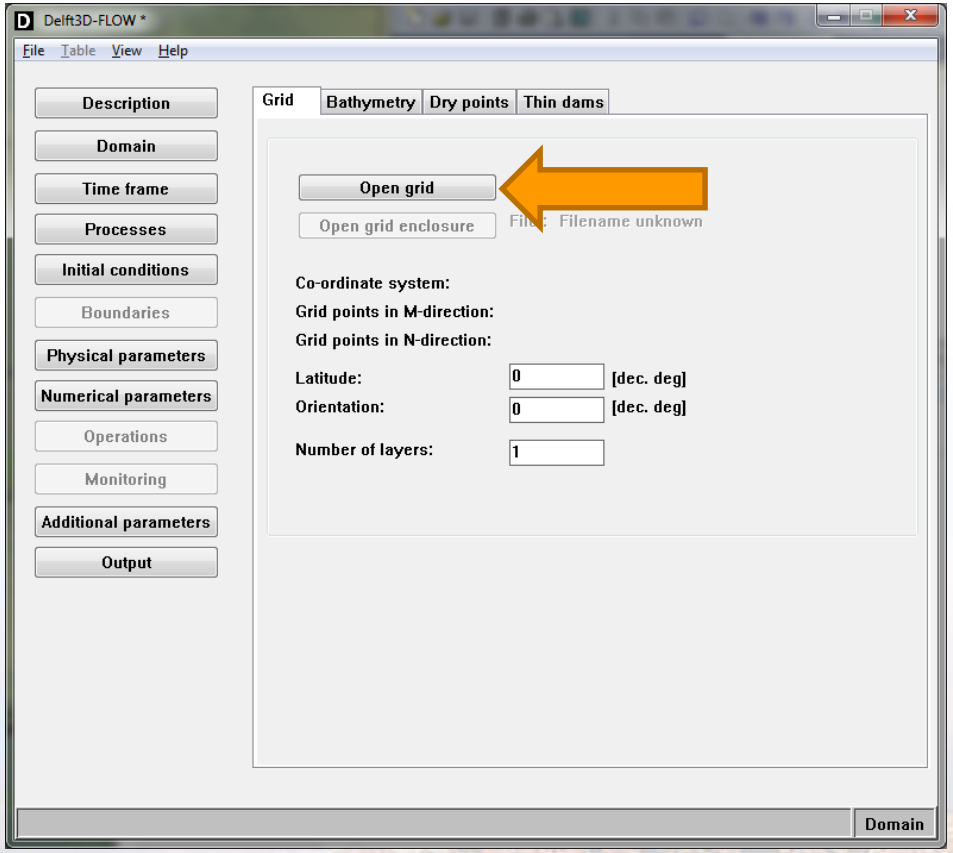
- **Select File → New**

Delft3D-FLOW



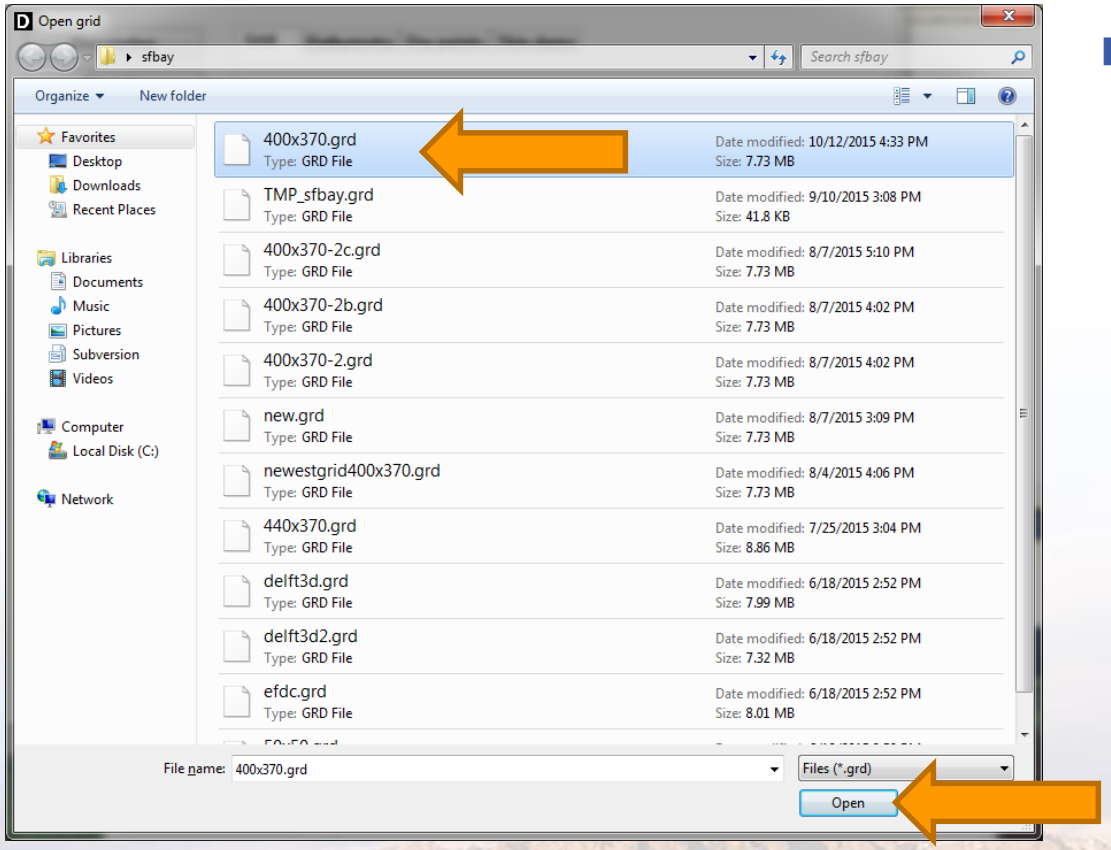
- **Under the Description tab, label your .mdf file.**

Delft3D-FLOW



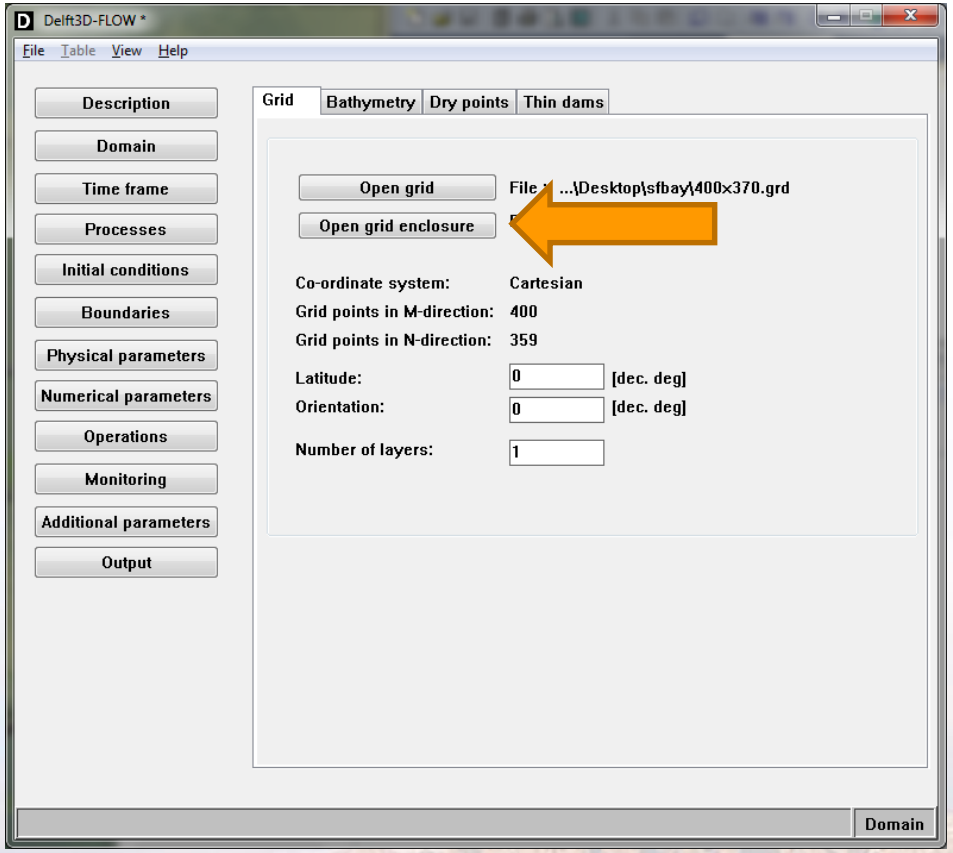
- Under Domain → Grid, select Open Grid

Delft3D-FLOW



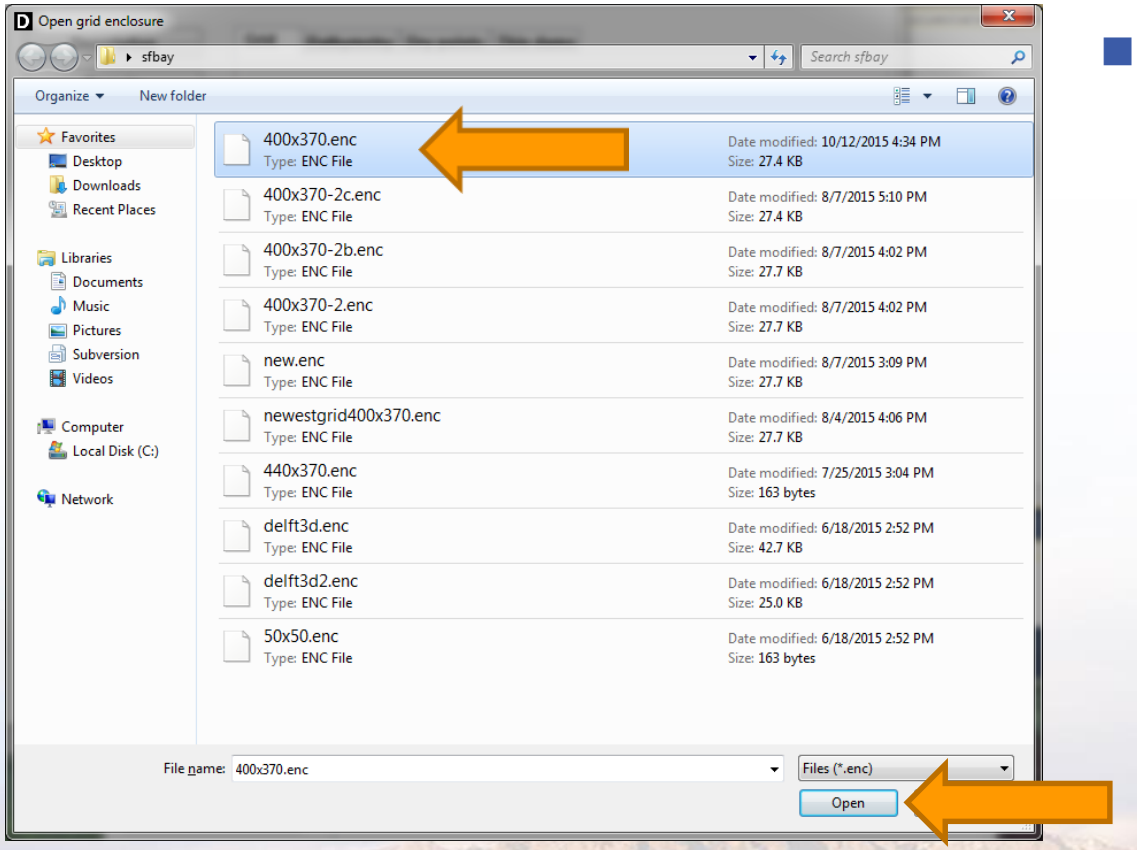
- Select 400x370.grd

Delft3D-FLOW



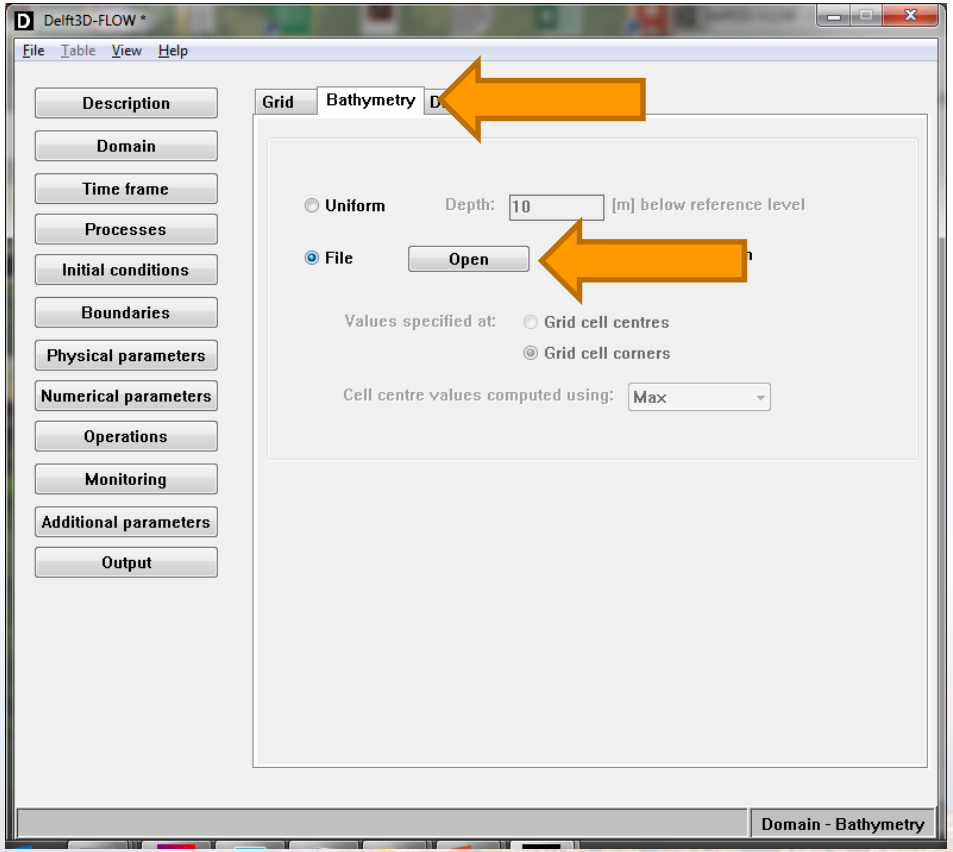
- Select Open grid enclosure

Delft3D-FLOW



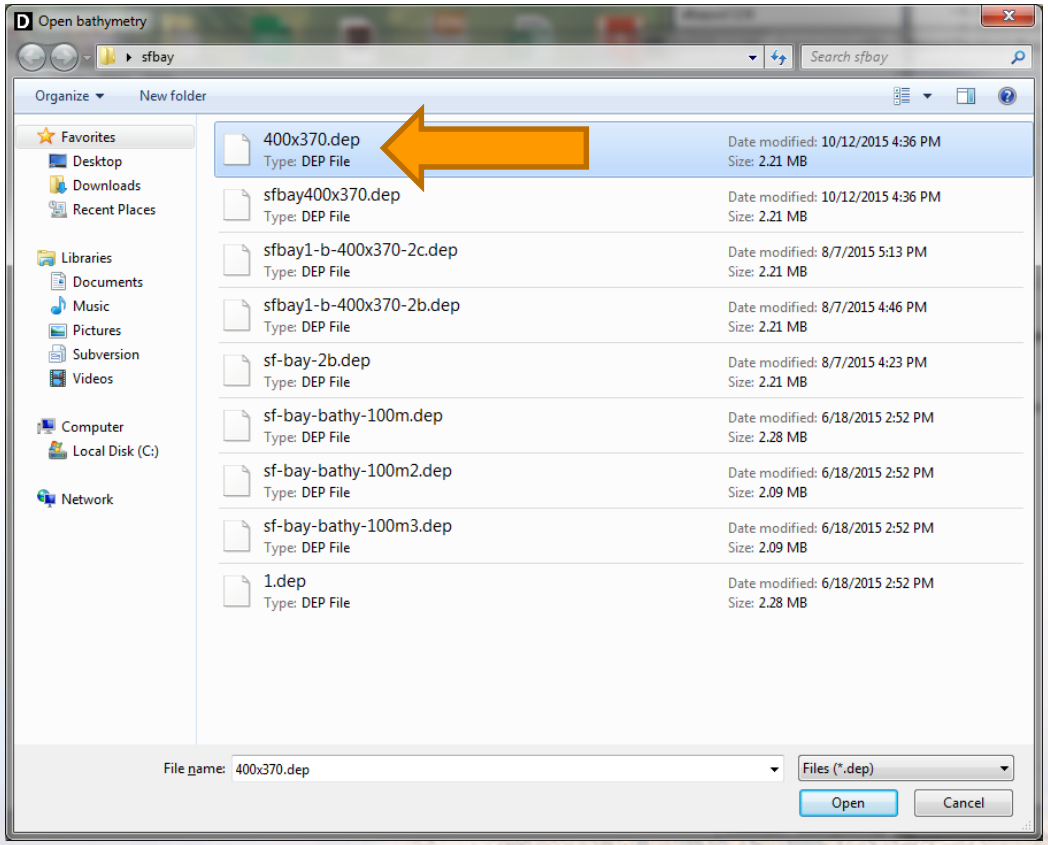
- Select 400X370.enc

Delft3D-FLOW



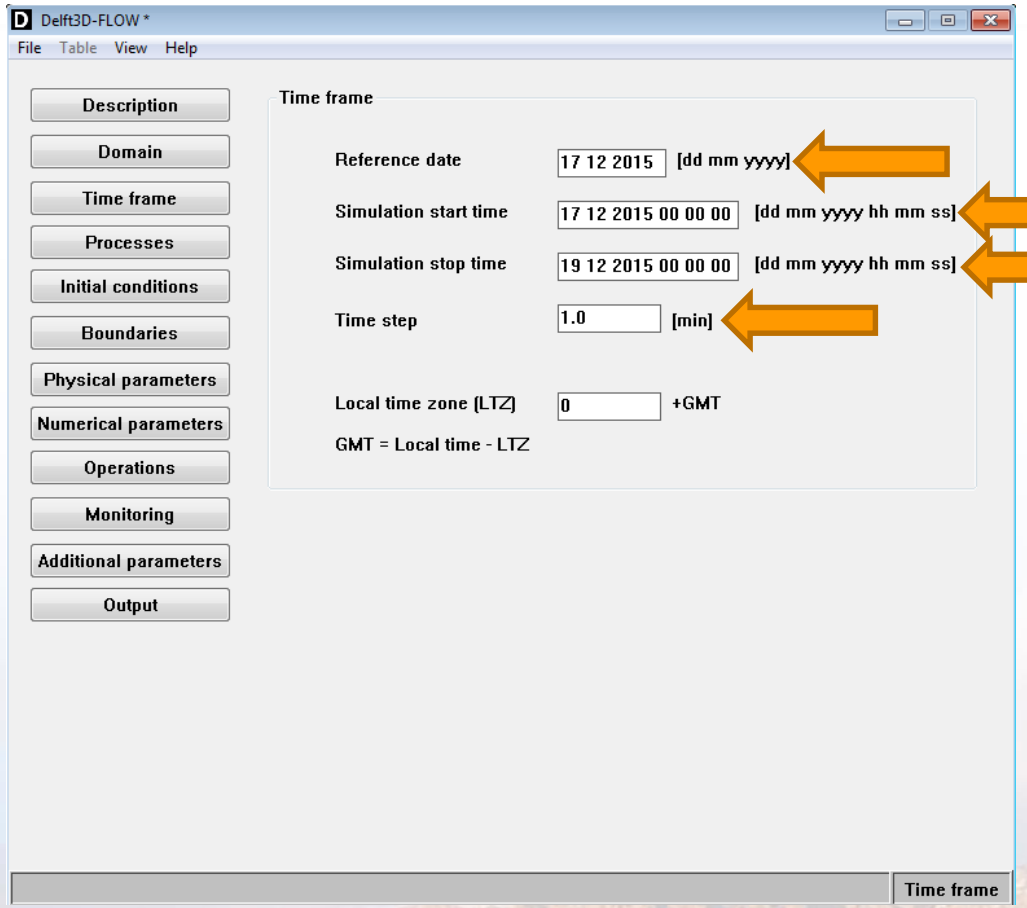
- Select Bathymetry and then Select Open

Delft3D-FLOW



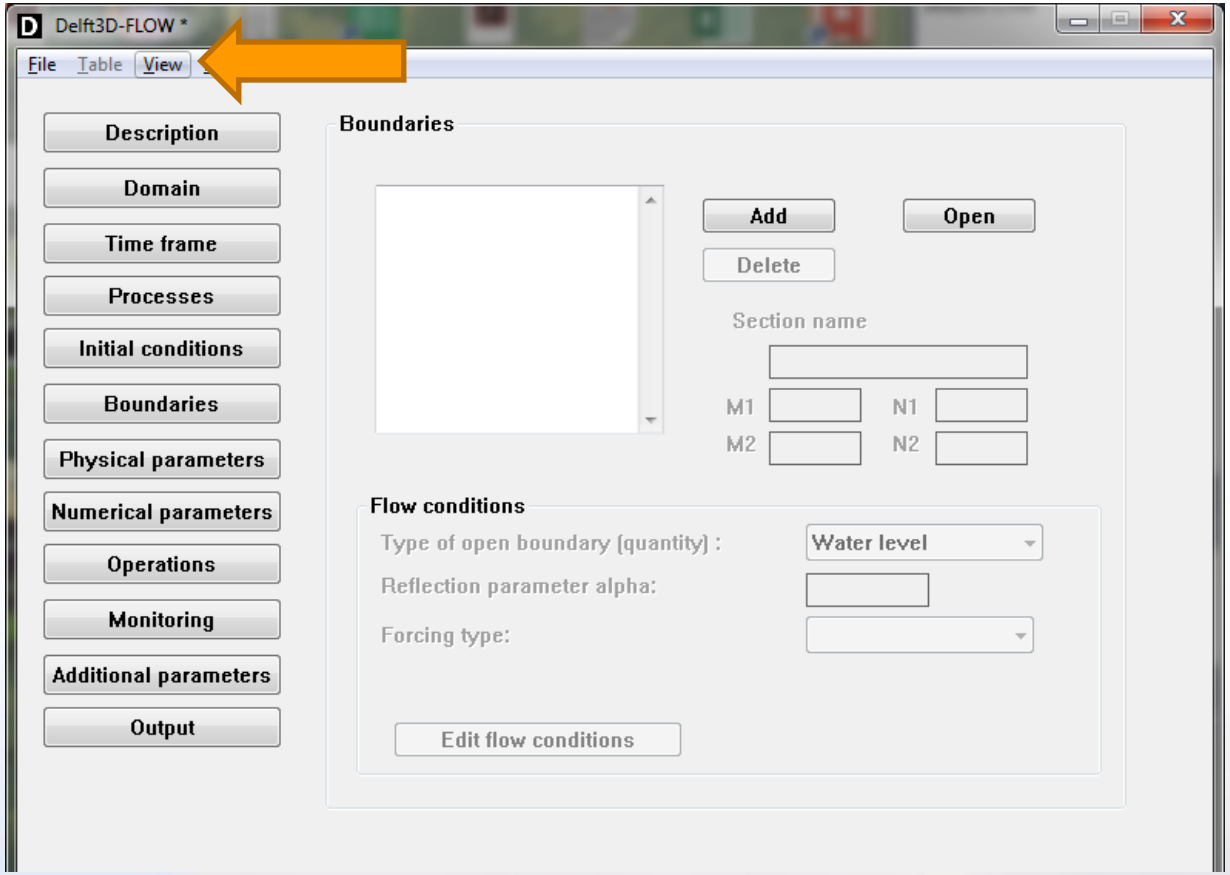
- Select the 400x370.dep depth file

Delft3D-FLOW



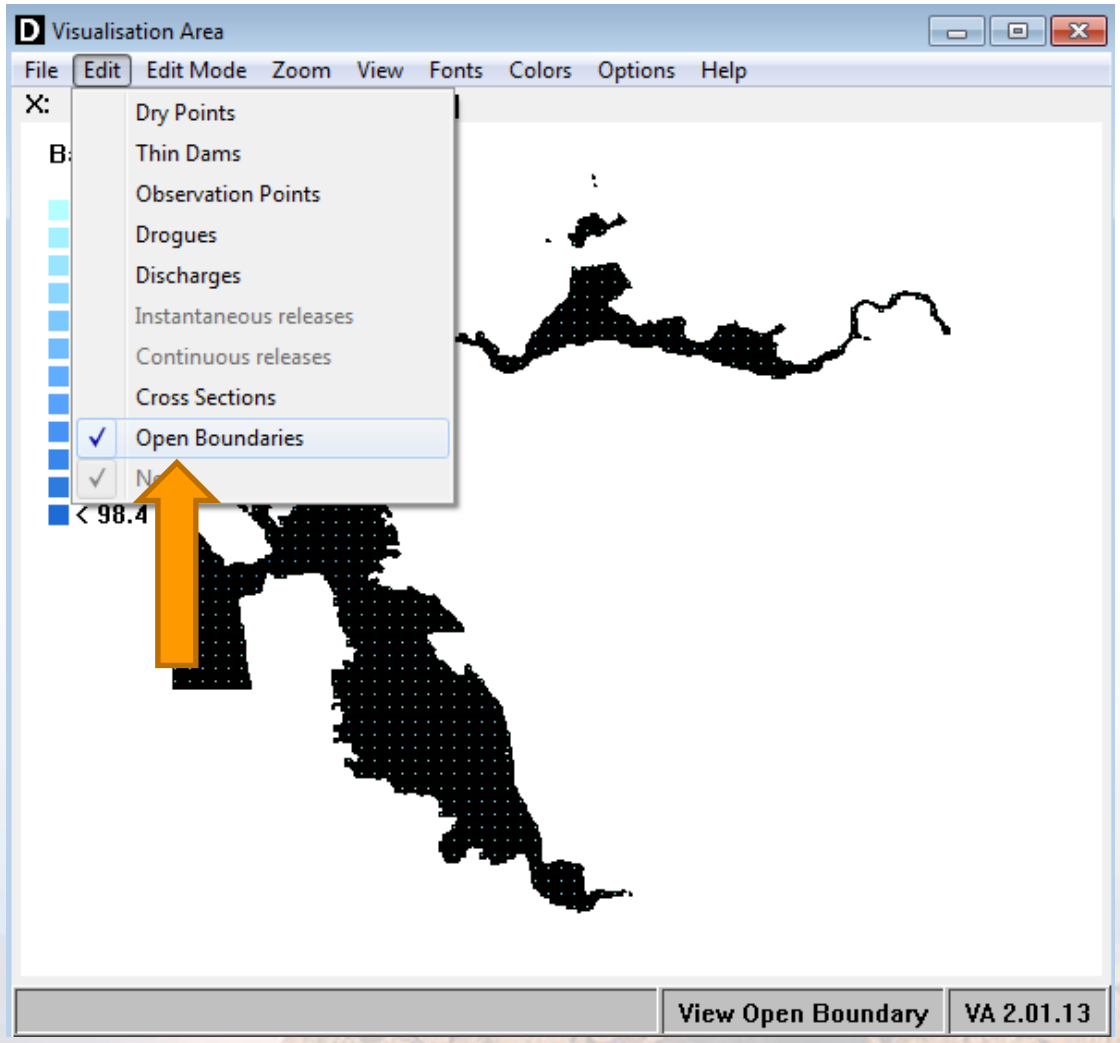
- Under Time frame, fill in the Reference date, Simulation start time, Simulation stop time, and Time step.

Delft3D-FLOW



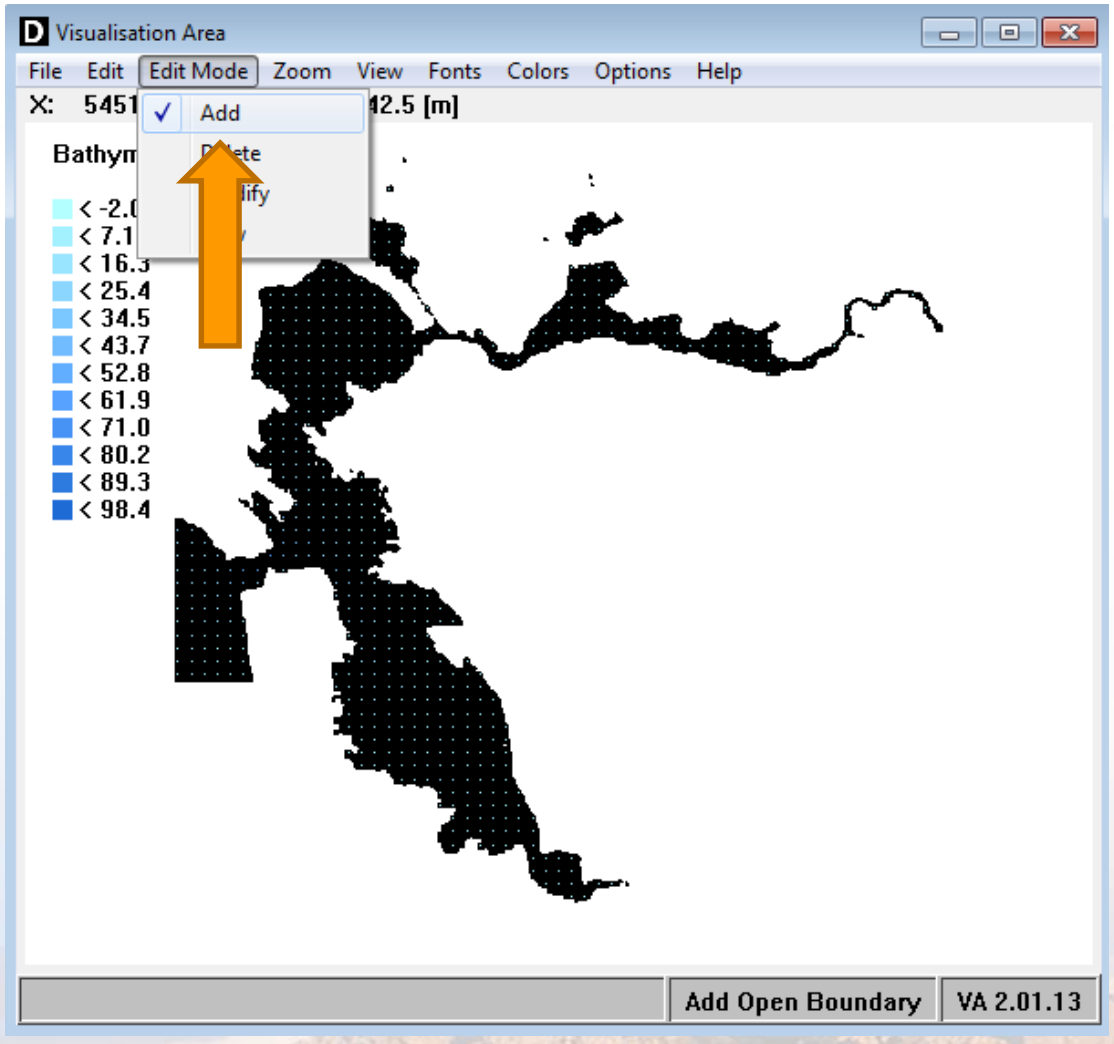
- Select the Boundaries Tab
- Then, select View → Visualization Area

Visualization Area



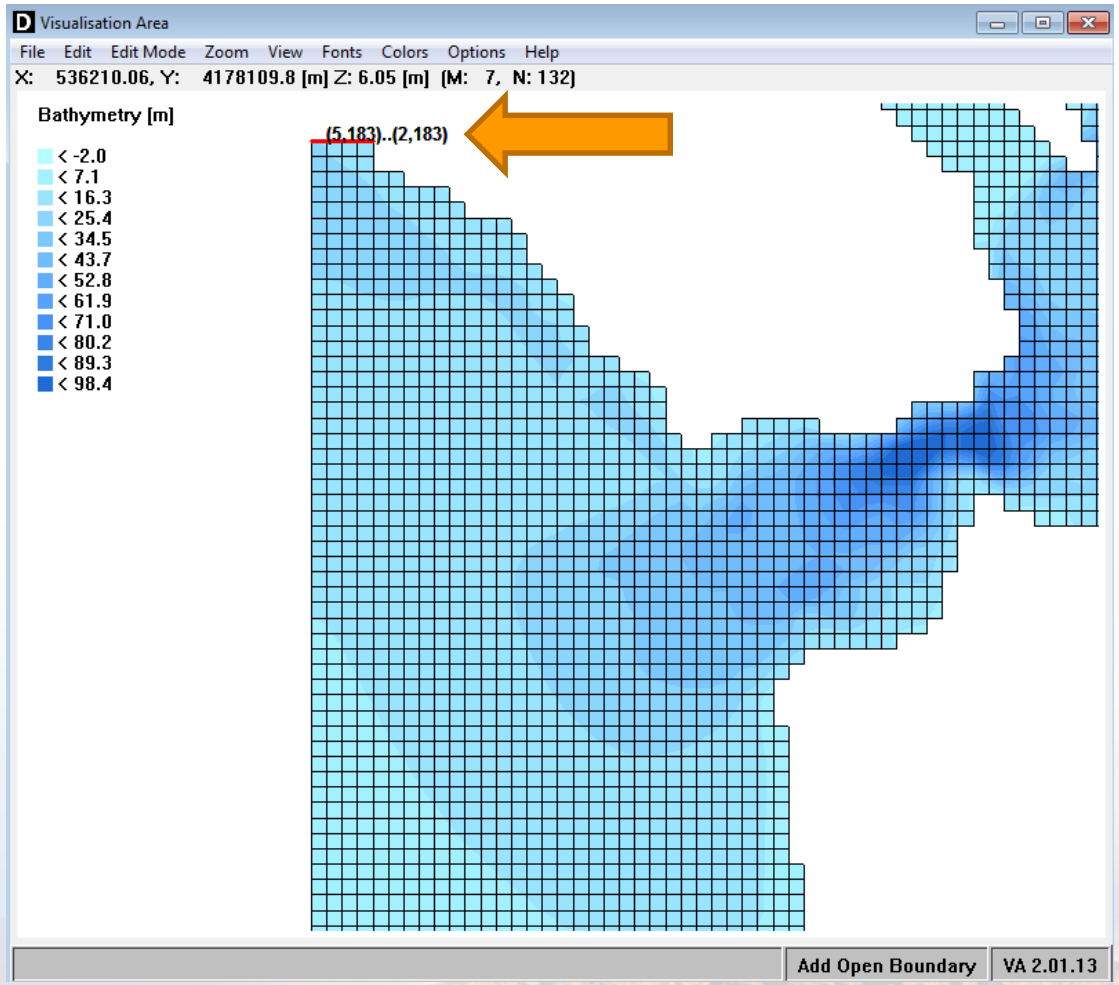
- Select Edit → Open Boundaries

Visualization Area



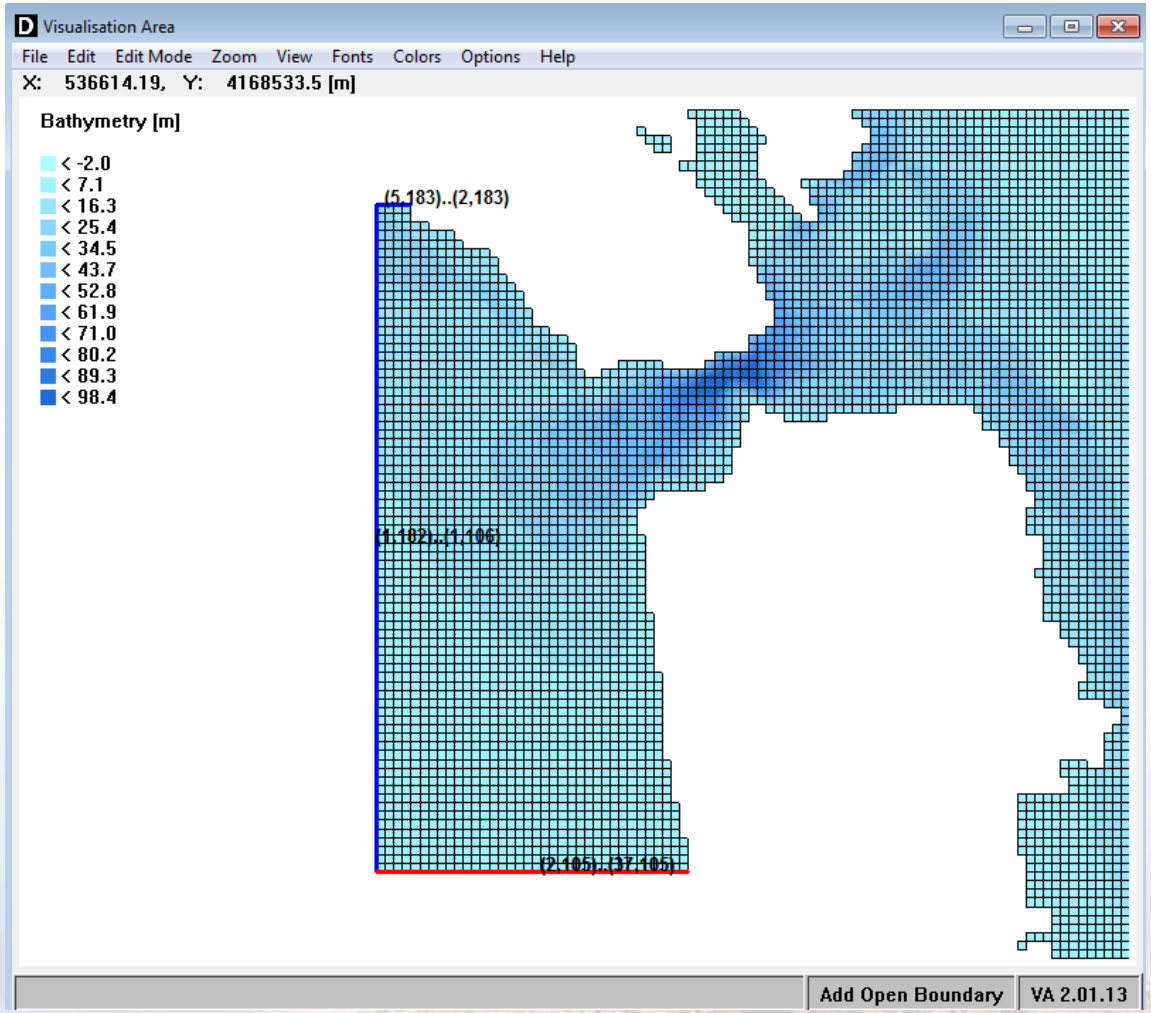
- **Select Edit Mode → Add**

Visualization Area



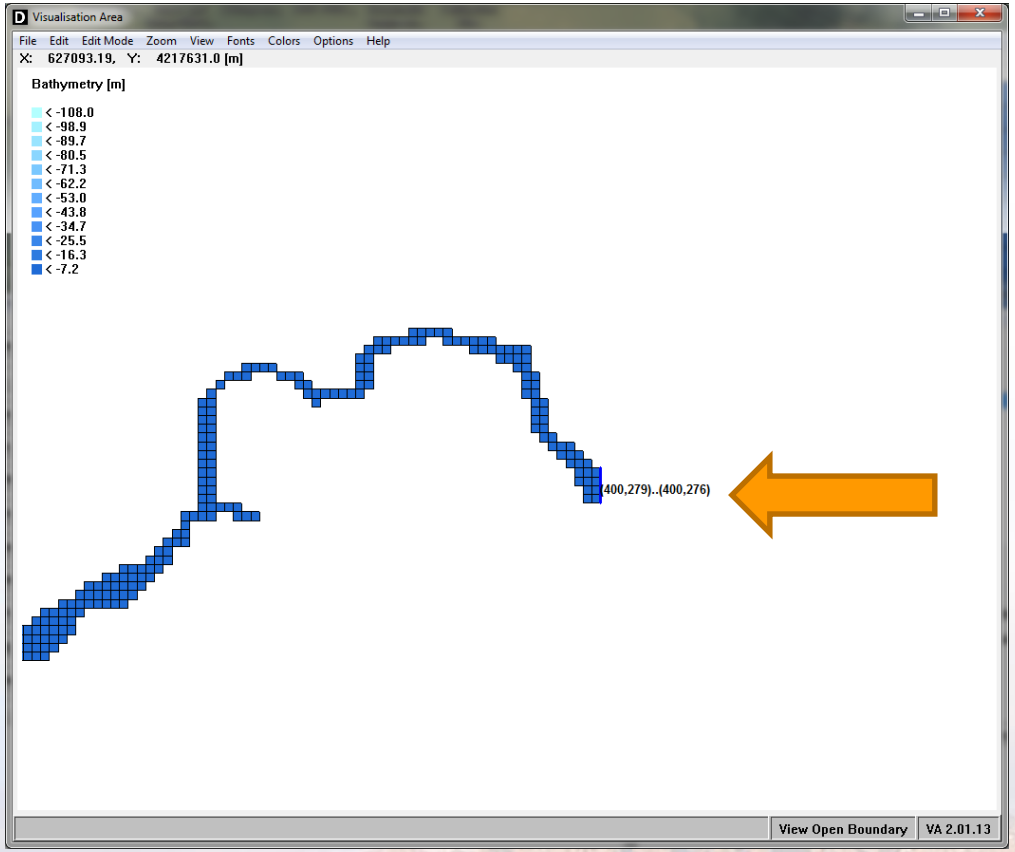
- Using the mouse, drag and add boundaries along the edge of the grid.

Visualization Area



- The left side should look like this.

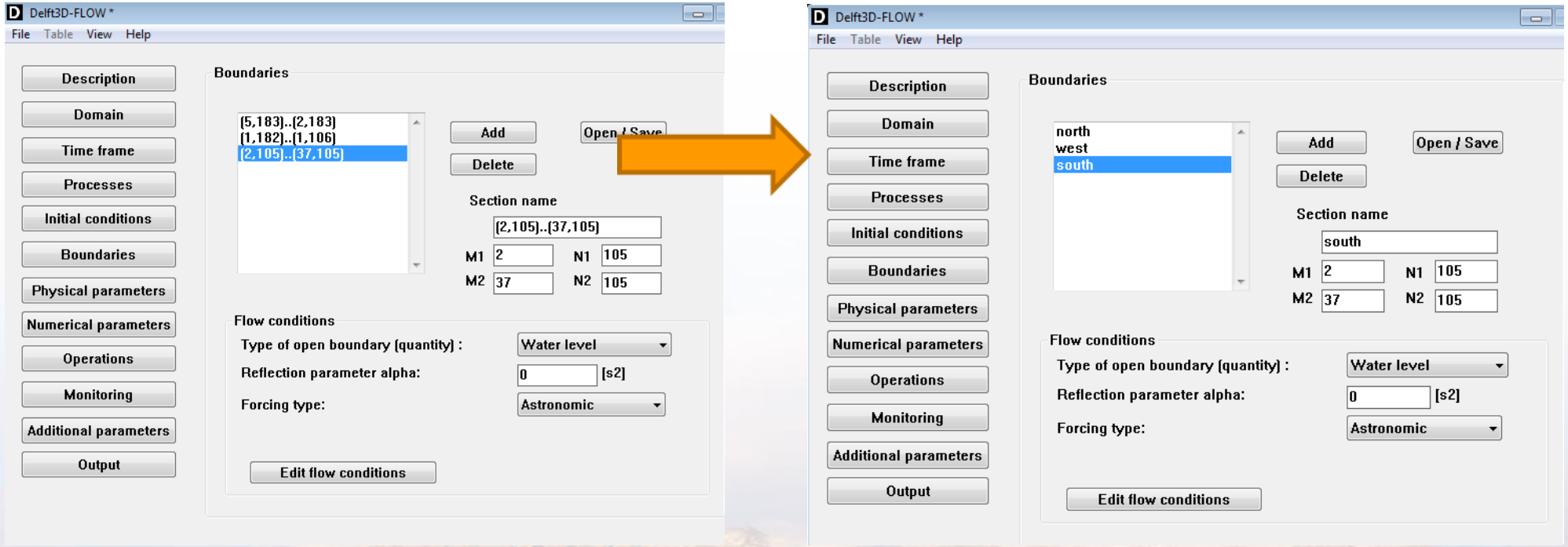
Visualization Area



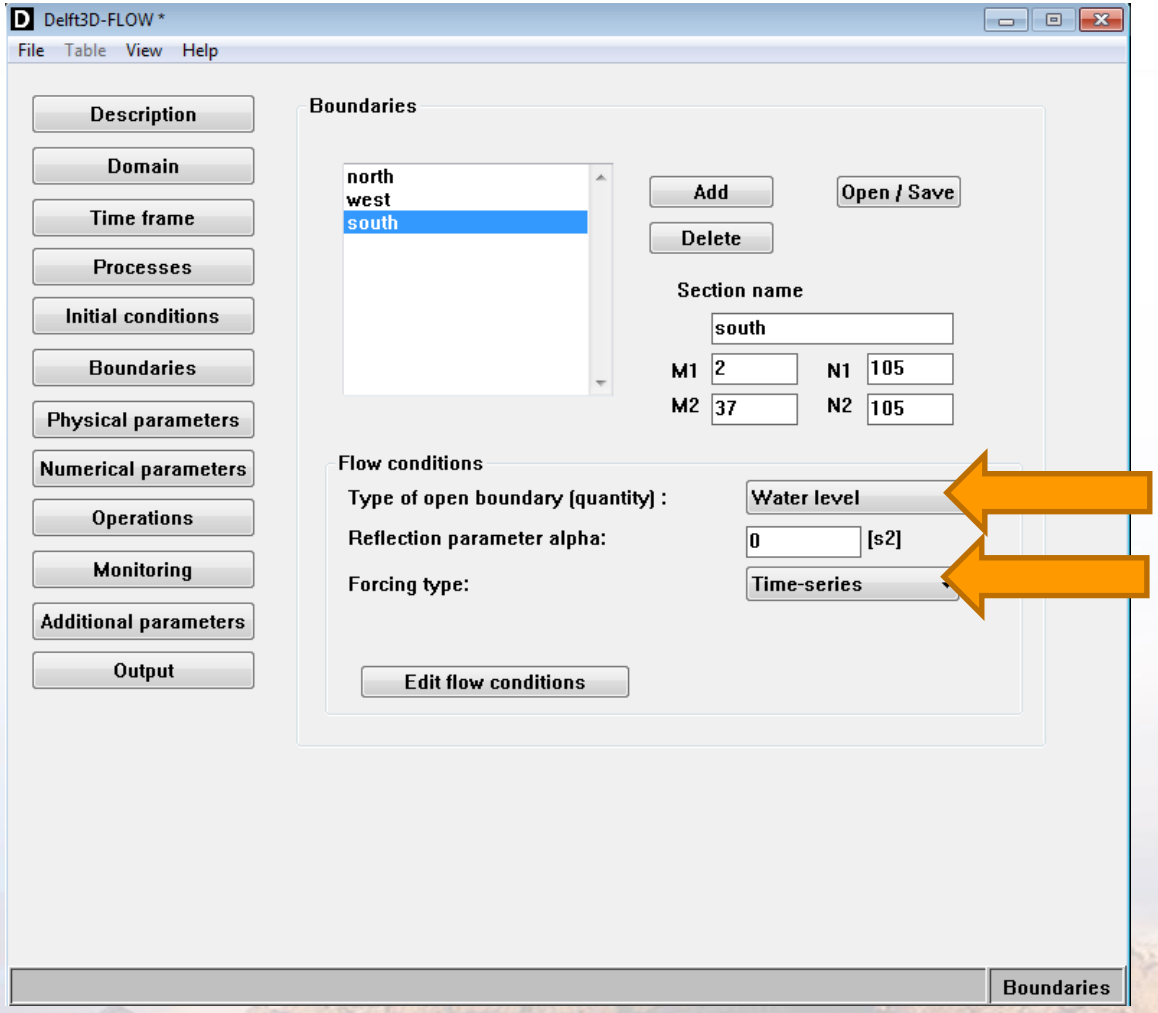
- The boundary on the far right side of the bay (river inlet) could be added to allow for the input of river data.
- For the purposes of this tutorial, this will not be necessary.

Labeling Boundaries

Rename the newly created boundaries to north, west, and south depending on the boundary's location on the grid.

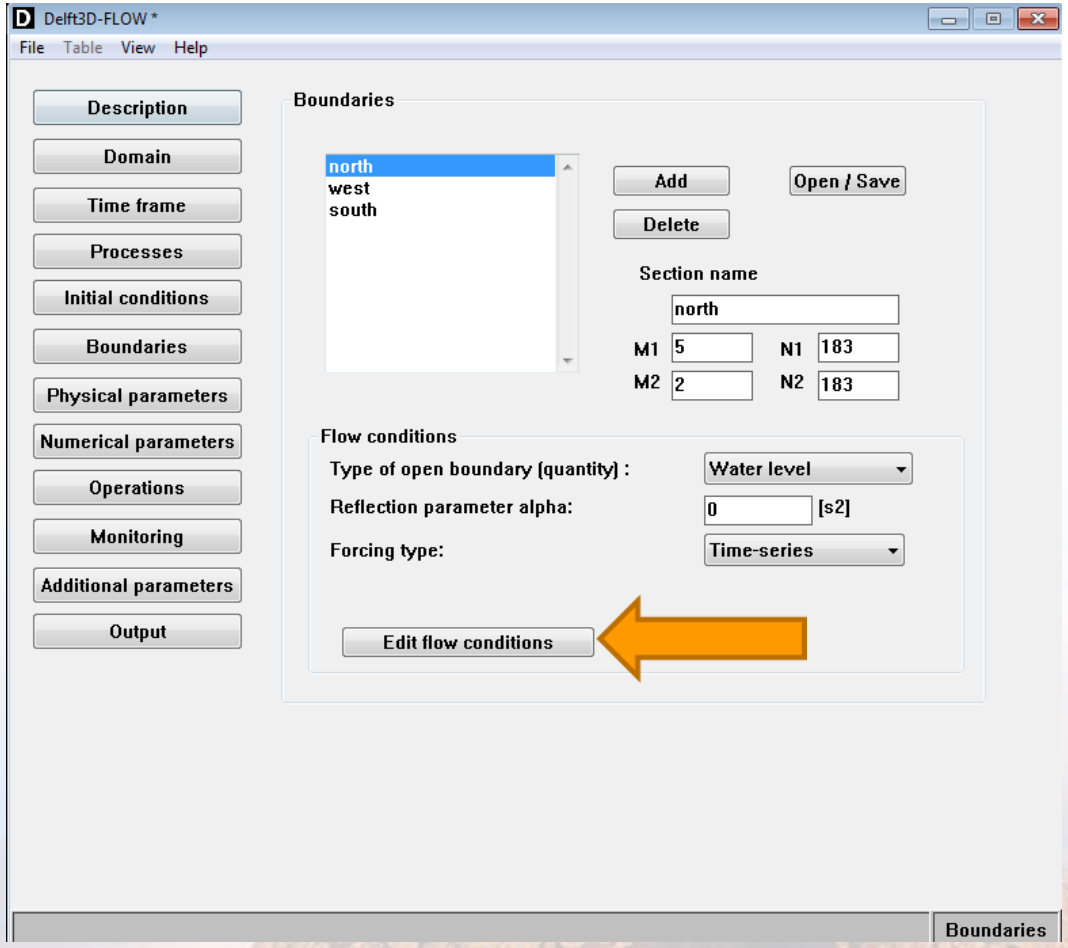


Editing Boundaries



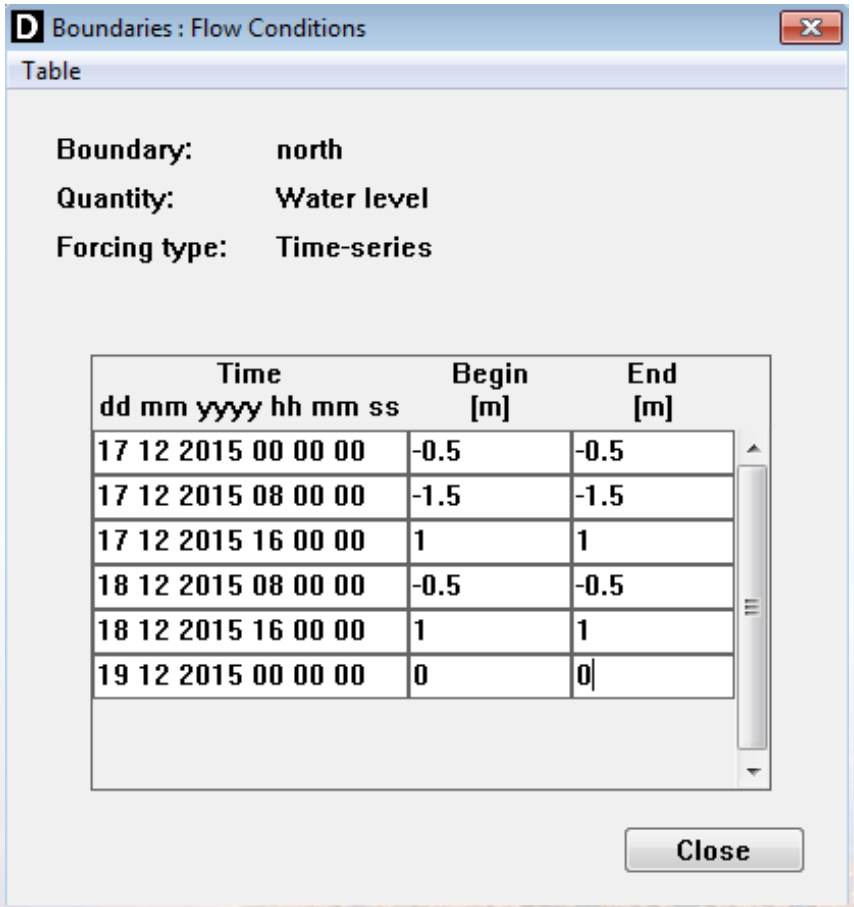
- Change the Forcing type for all boundaries
- Forcing Type → Time Series

Editing Boundary Flow Conditions



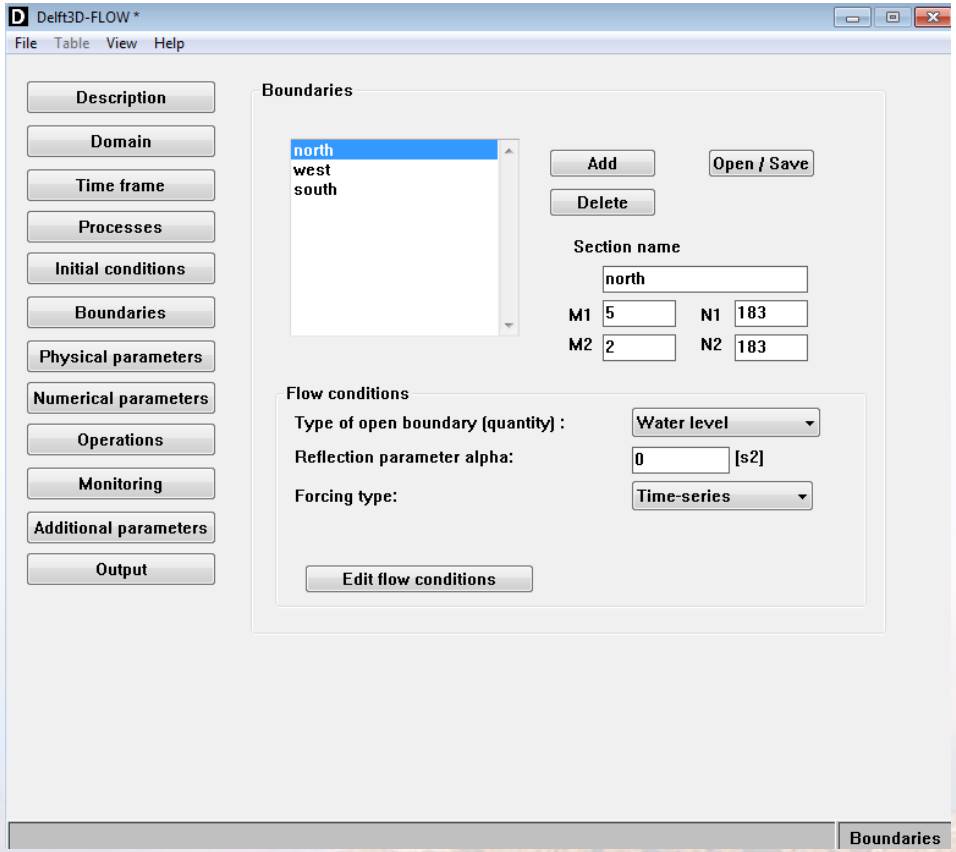
- Edit flow conditions for all boundaries (north, west, south).

Editing Boundary Flow Conditions



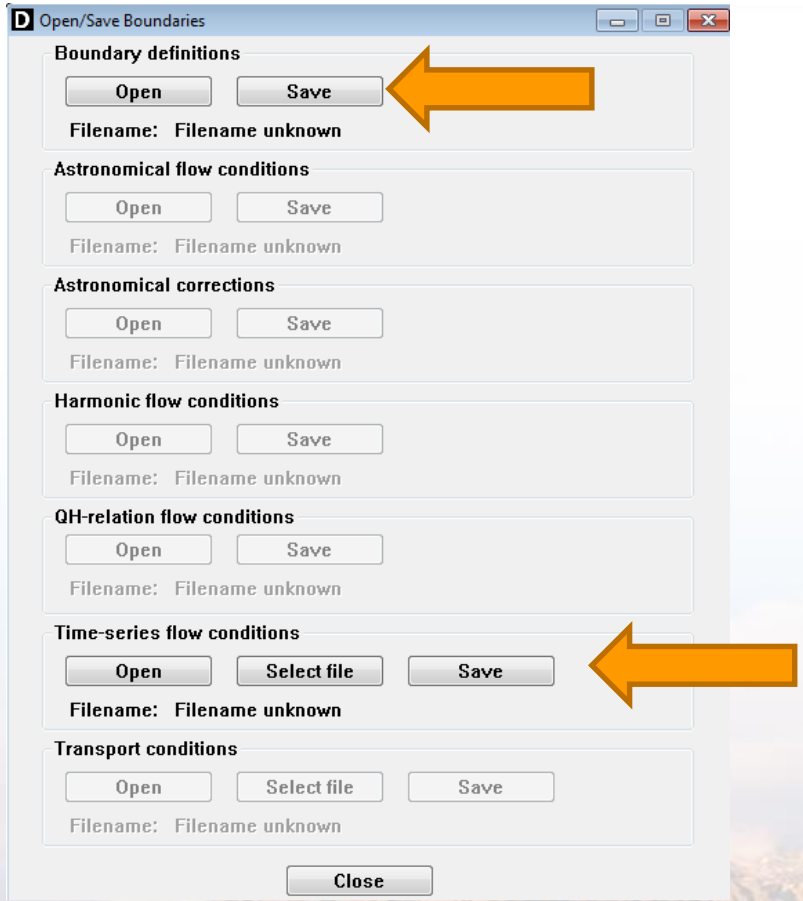
- For *each* boundary, edit flow conditions by entering water level data.
- For our purposes, have the flow conditions match this table (entry for "dd" will change depending on your model). Or choose to import water level data from external sites.
- Add rows to table by selecting:
  Table → Insert row above.
- Note: Data could be added manually through the GUI or entered in your .bct file.

Editing Boundary Flow Conditions



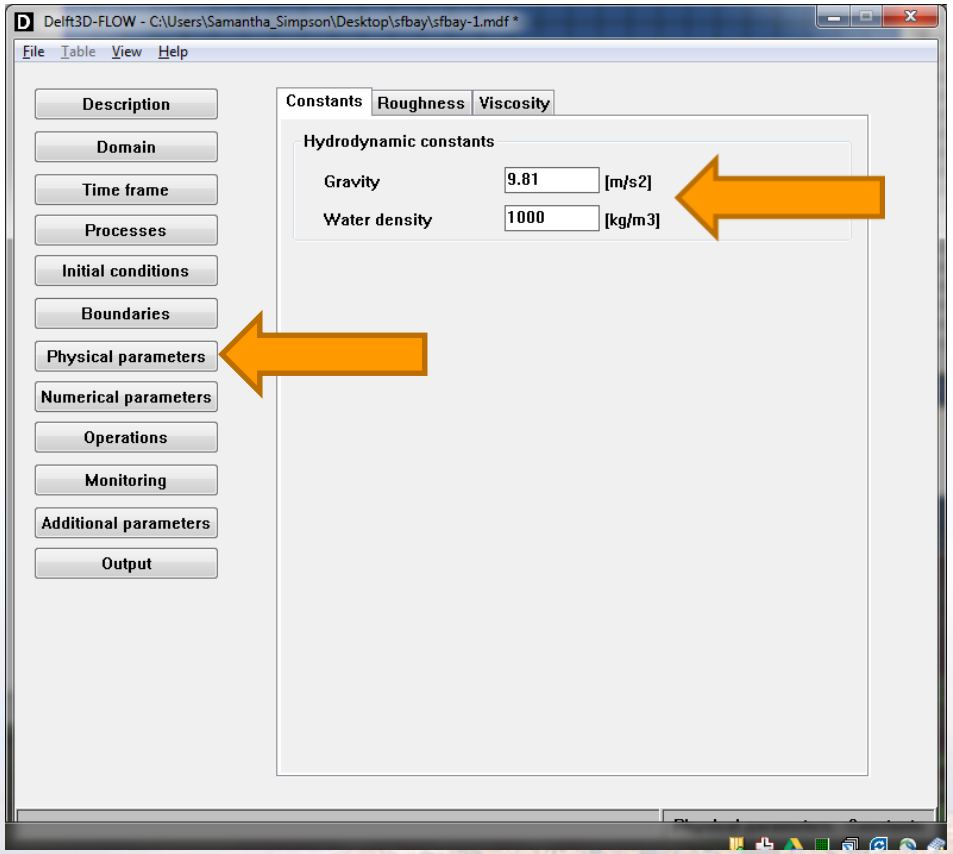
- Click on Open/Save to save the Boundary definitions and Time-series flow conditions

Editing Boundary Flow Conditions



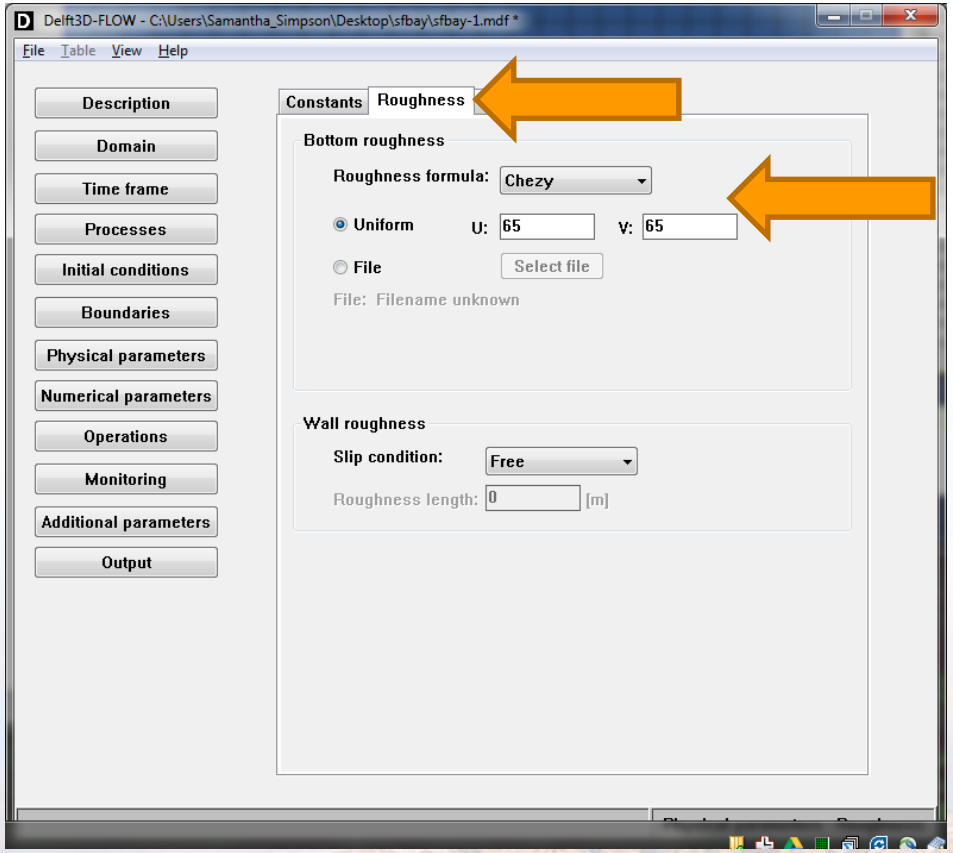
- Save Boundary definition as bnds.bnd
- Save Time-series flow conditions as boundaries.bct
- You have now created a .bnd and a .bct file.

Physical Parameters



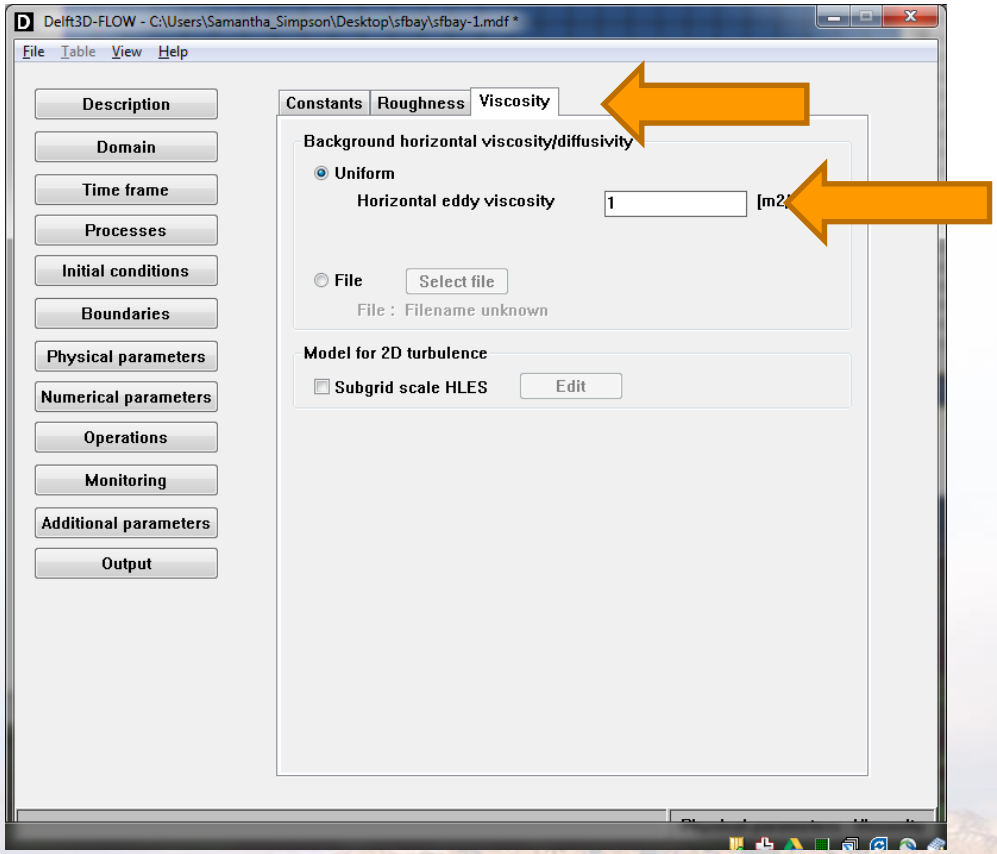
- Then under Physical parameters → Constants , add the gravity and water density

Physical Parameters



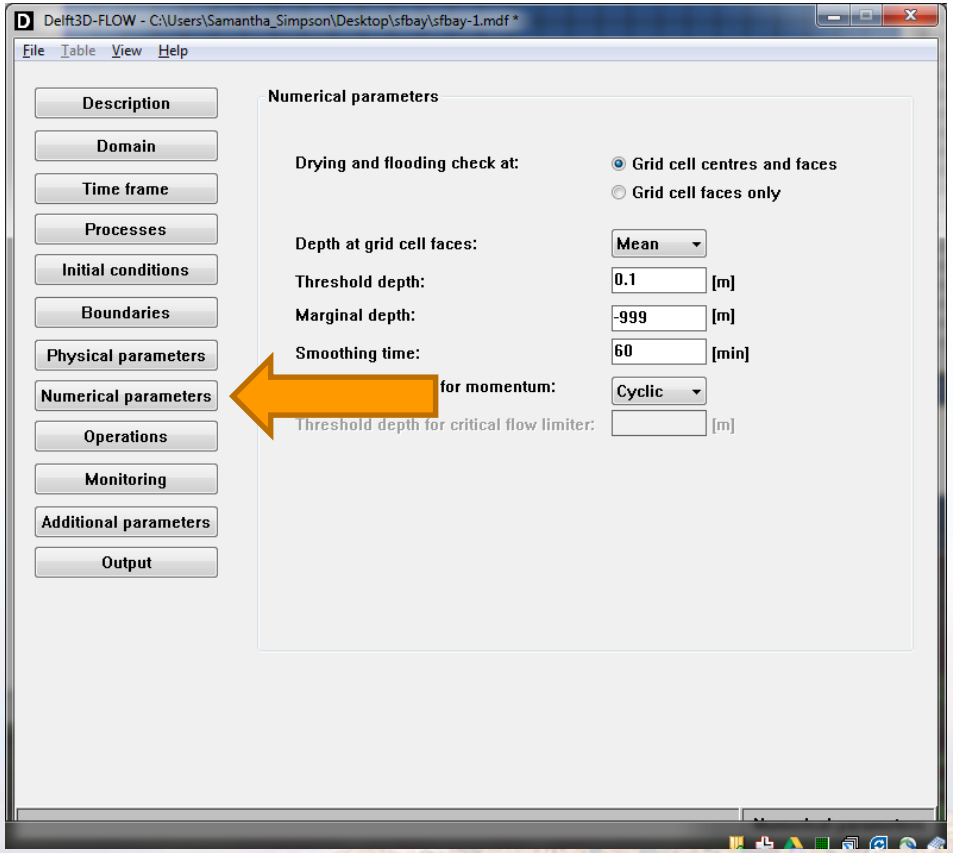
- Under the Roughness tab, use the Chezy Roughness Formula with Uniform U and V

Physical Parameters



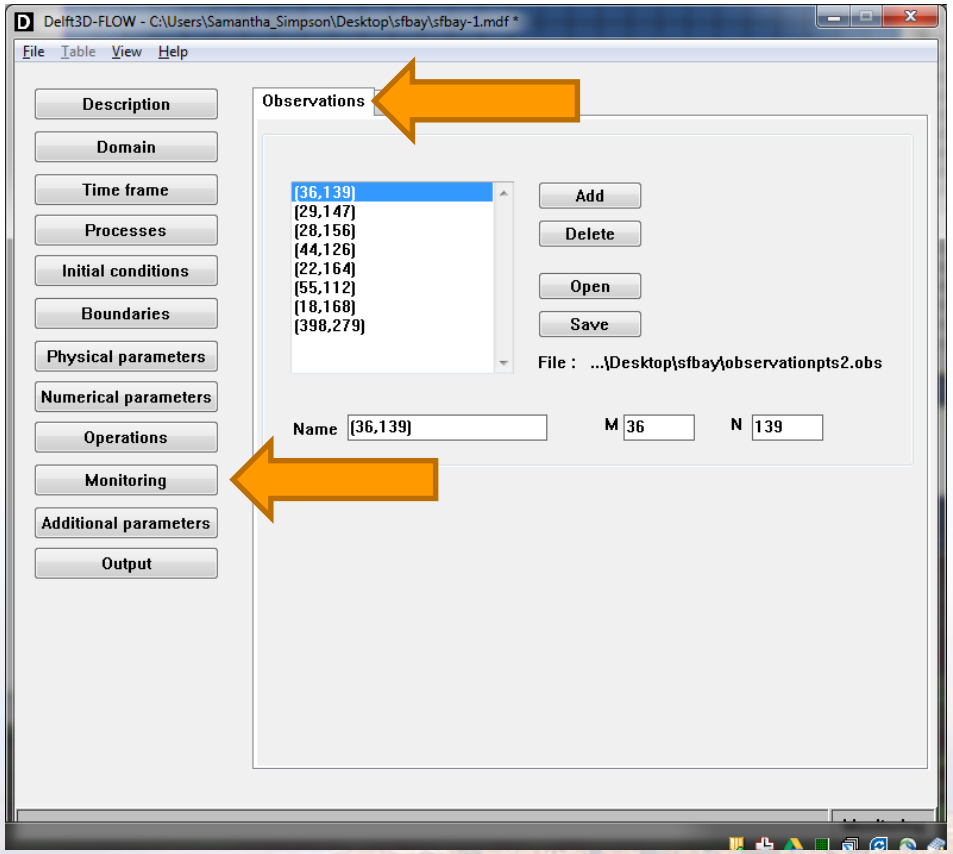
- Under Viscosity, select Uniform Horizontal Eddy Viscosity

Numerical Parameters



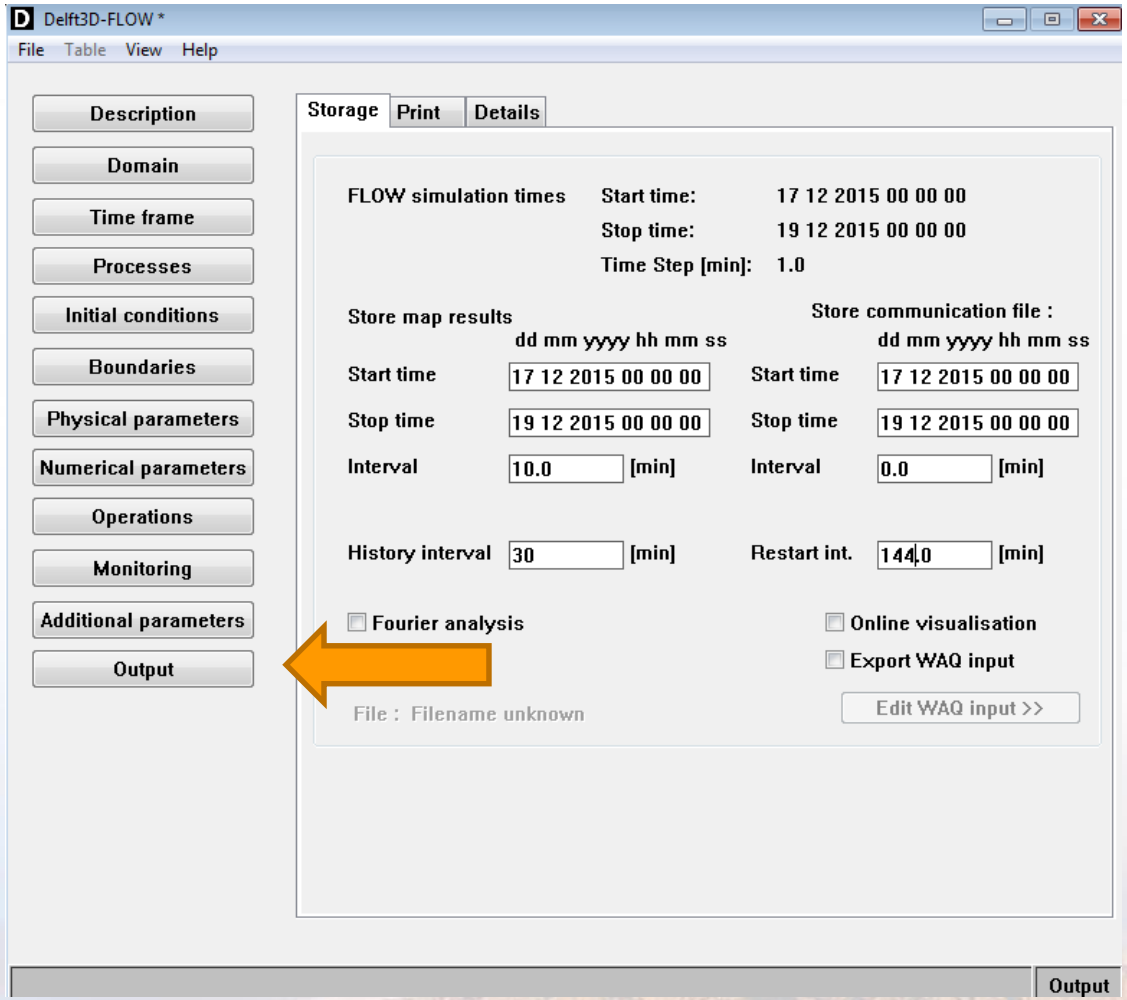
- Under Numerical Parameters, match these values.

Monitoring



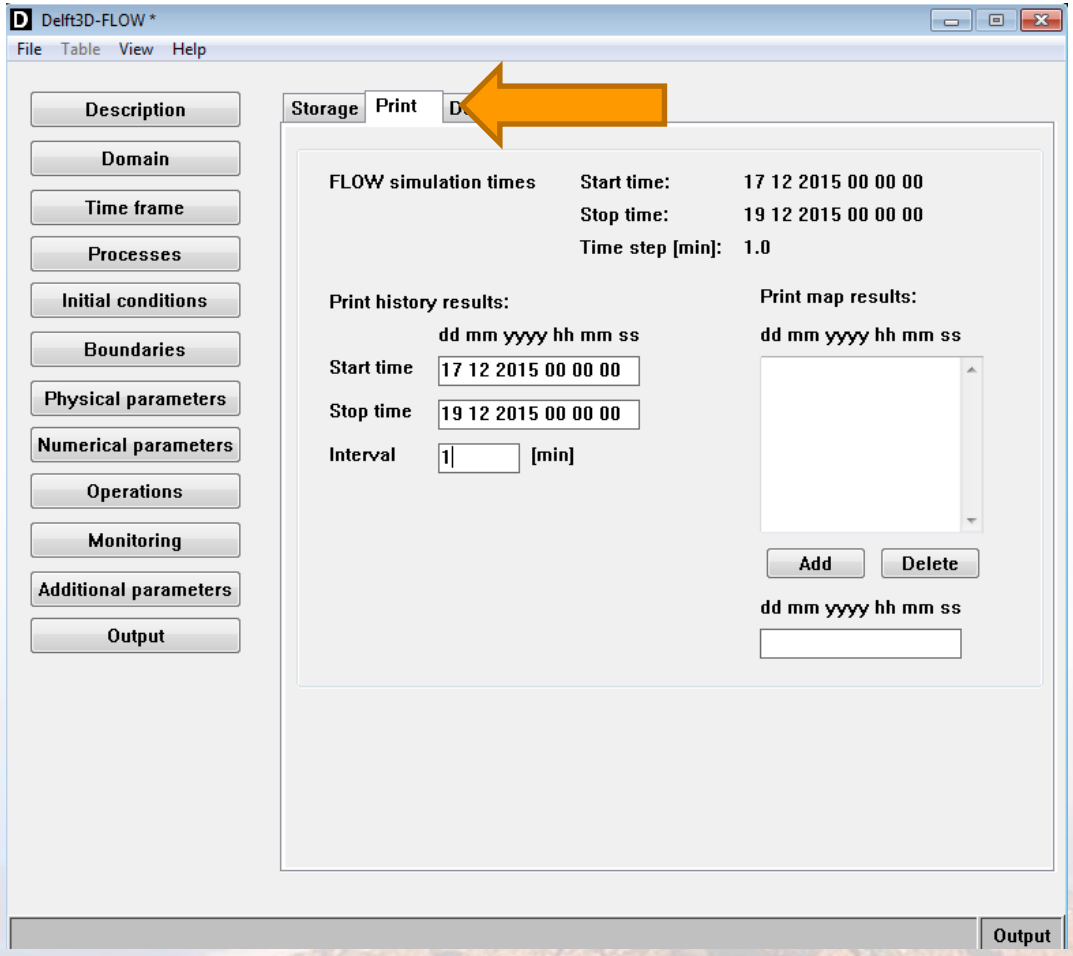
- Using the visualization area, select observation points and save.
- Or open file of pre-selected observation points.

Output

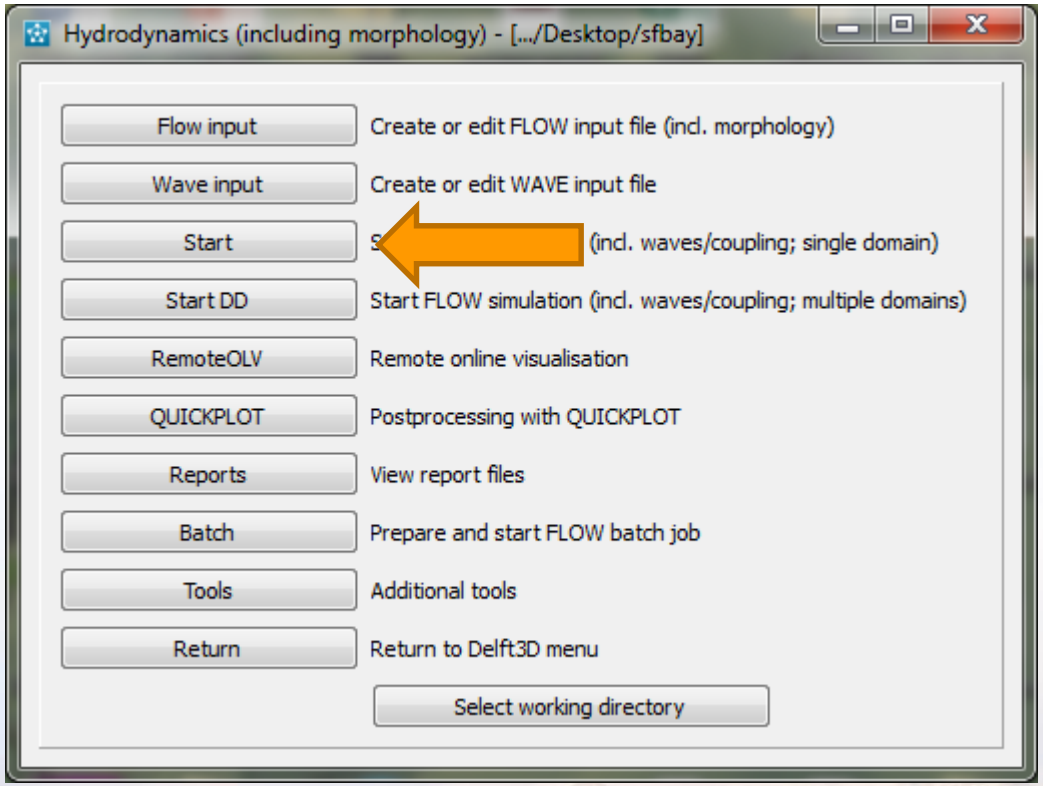


- Under the Output → Storage tab:

Output


Delft3D-FLOW *
File Table View Help
Description
Domain
Time frame
Processes
Initial conditions
Boundaries
Physical parameters
Numerical parameters
Operations
Monitoring
Additional parameters
Output
Storage
Print
FLOW simulation times
Start time: 17 12 2015 00 00 00
Stop time: 19 12 2015 00 00 00
Time step [min]: 1.0
Print history results:
dd mm yyyy hh mm ss
Start time 17 12 2015 00 00 00
Stop time 19 12 2015 00 00 00
Interval 1 [min]
Print map results:
dd mm yyyy hh mm ss
Add
Delete
dd mm yyyy hh mm ss
Output

Delft3D-FLOW



- **After saving the .mdf file, return to the main FLOW menu.**
- **Select start. And choose your .mdf file.**
- **Expected run time: ~7 minutes**