

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.



Sandia National Laboratories

Overview

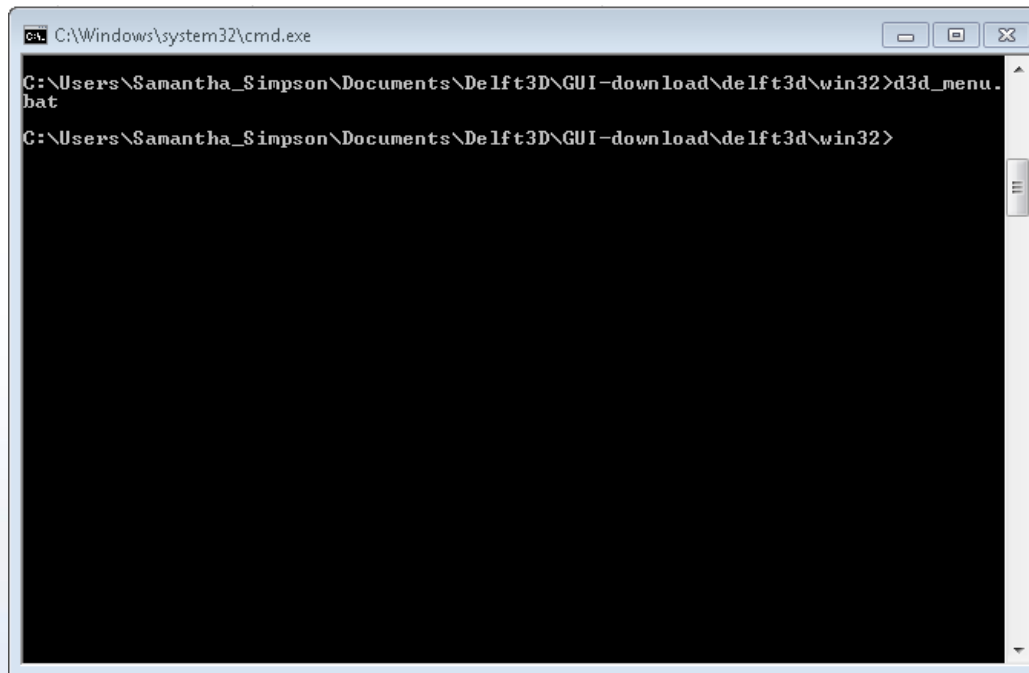
This tutorial will aim to:

- **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

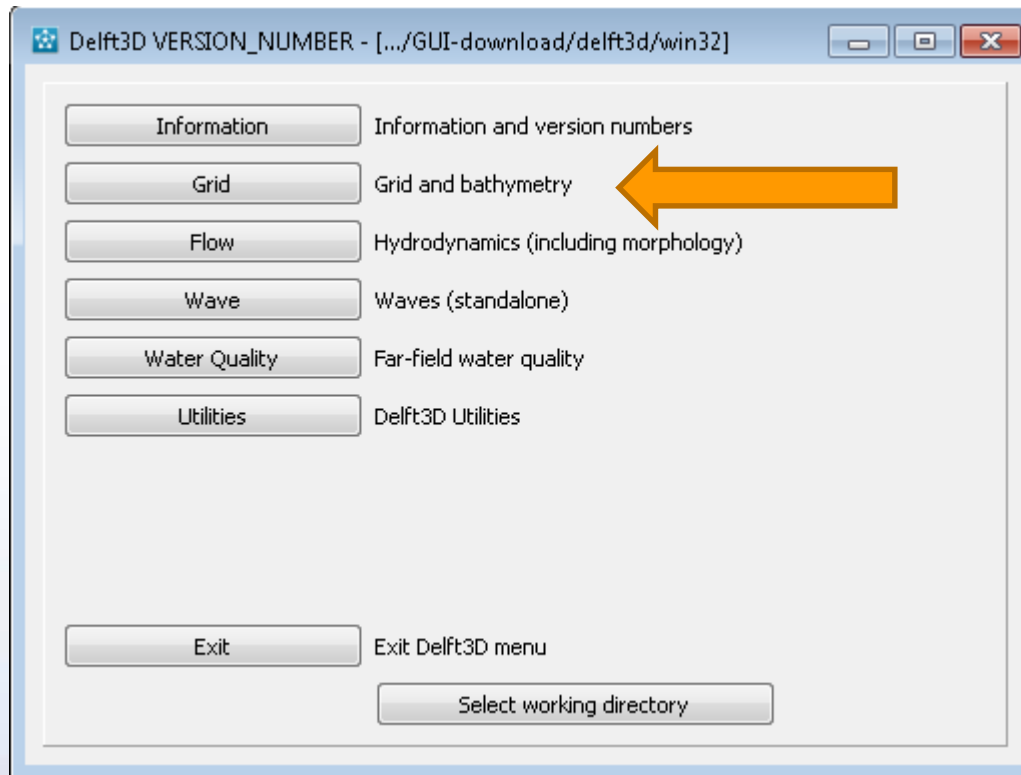
- Start the Delft 3D program.
- From the Windows command line, run d3d_menu.bat



```
C:\Windows\system32\cmd.exe
C:\Users\Samantha_Simpson\Documents\Delft3D\GUI-download\delft3d\win32>d3d_menu.
bat
C:\Users\Samantha_Simpson\Documents\Delft3D\GUI-download\delft3d\win32>
```

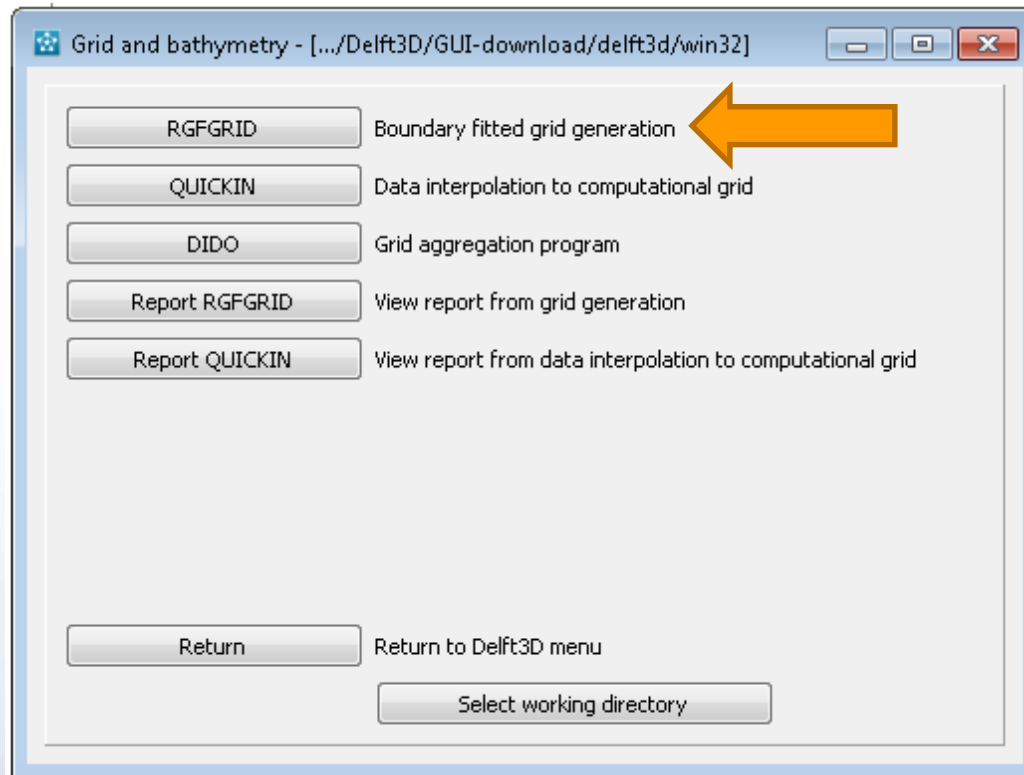
Grid

- From the main menu, select Grid

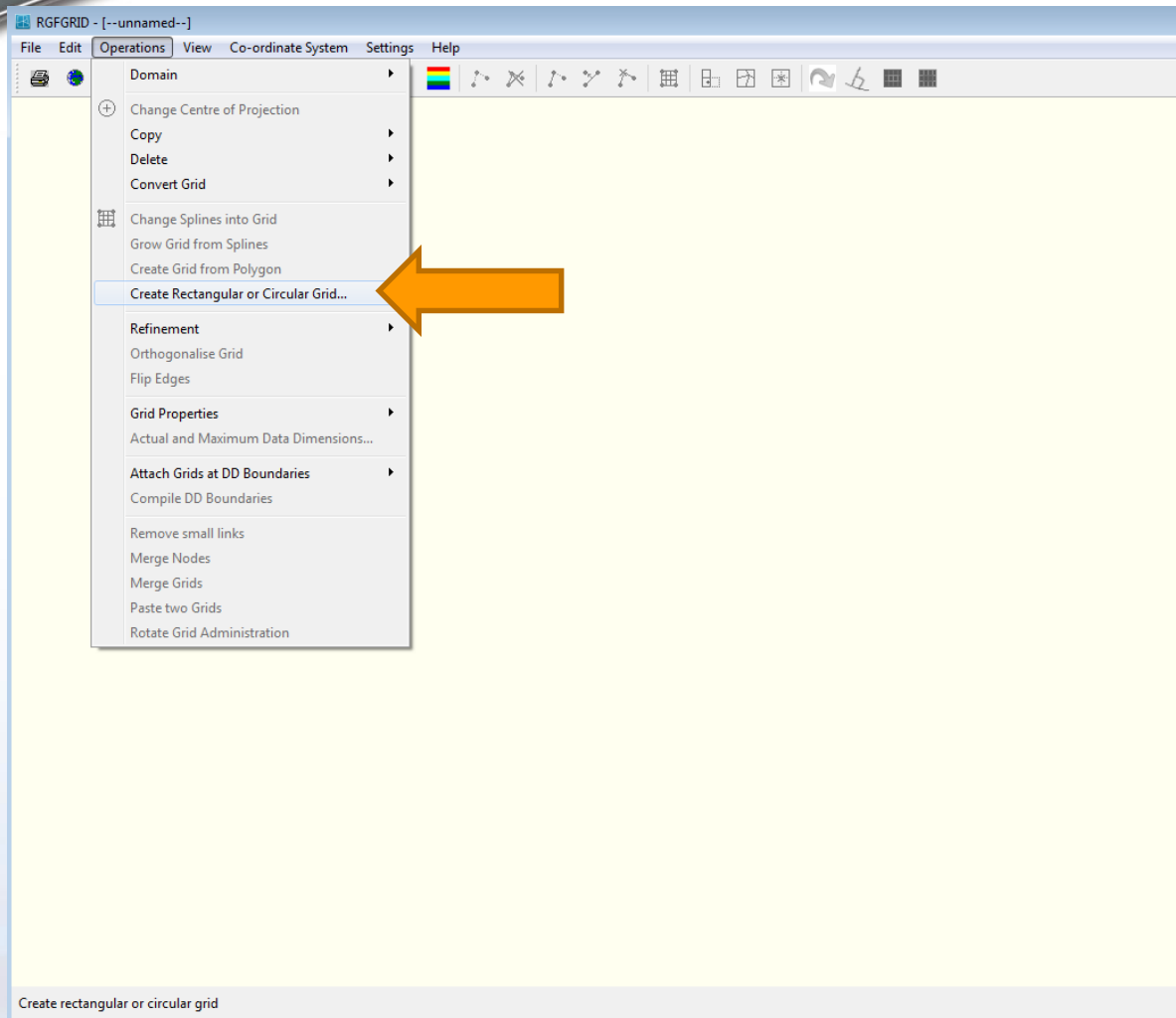


RGFGRID

- From the Grid and Bathymetry menu, select RGFGRID



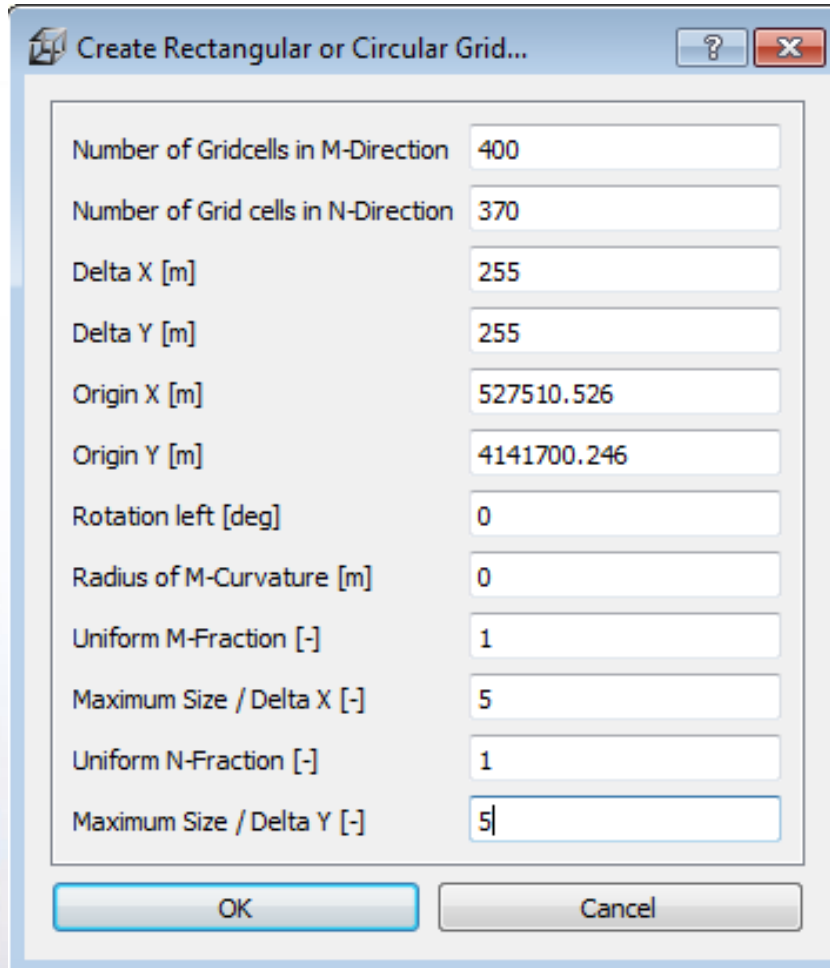
RGFGRID



- 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.



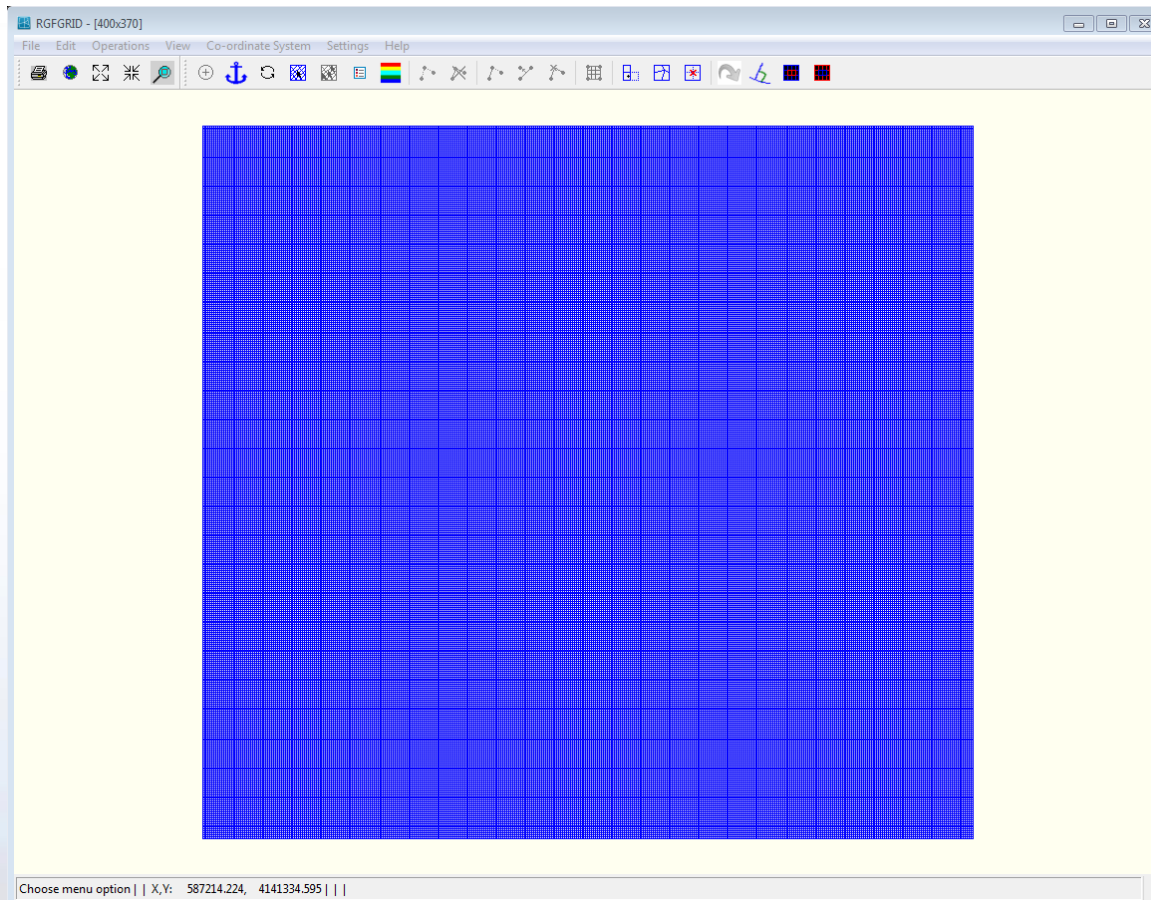
Create Rectangular or Circular Grid...

Number of Gridcells in M-Direction	400
Number of Grid cells in N-Direction	370
Delta X [m]	255
Delta Y [m]	255
Origin X [m]	527510.526
Origin Y [m]	4141700.246
Rotation left [deg]	0
Radius of M-Curvature [m]	0
Uniform M-Fraction [-]	1
Maximum Size / Delta X [-]	5
Uniform N-Fraction [-]	1
Maximum Size / Delta Y [-]	5

OK Cancel

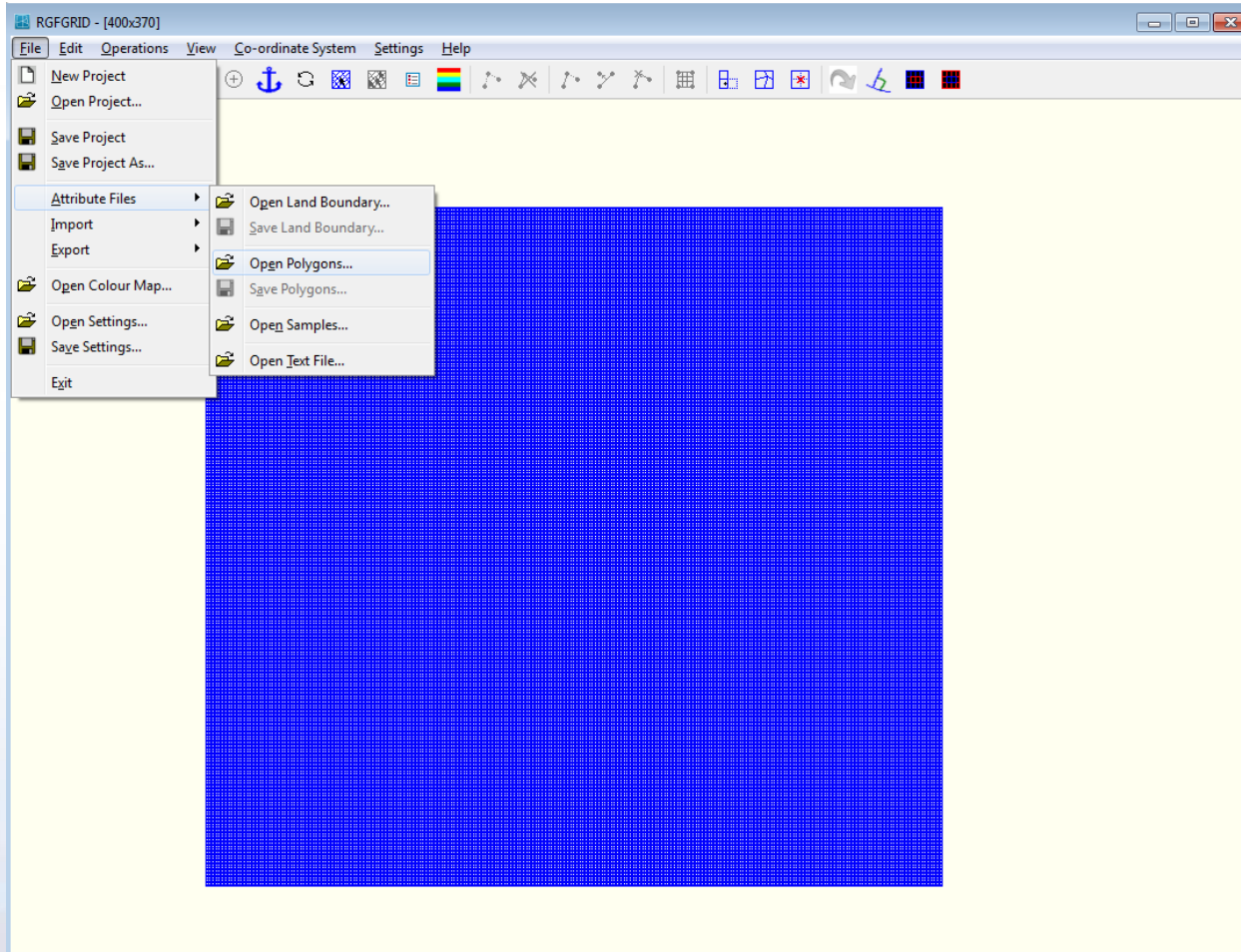
RGFGRID

- This is what you should see.



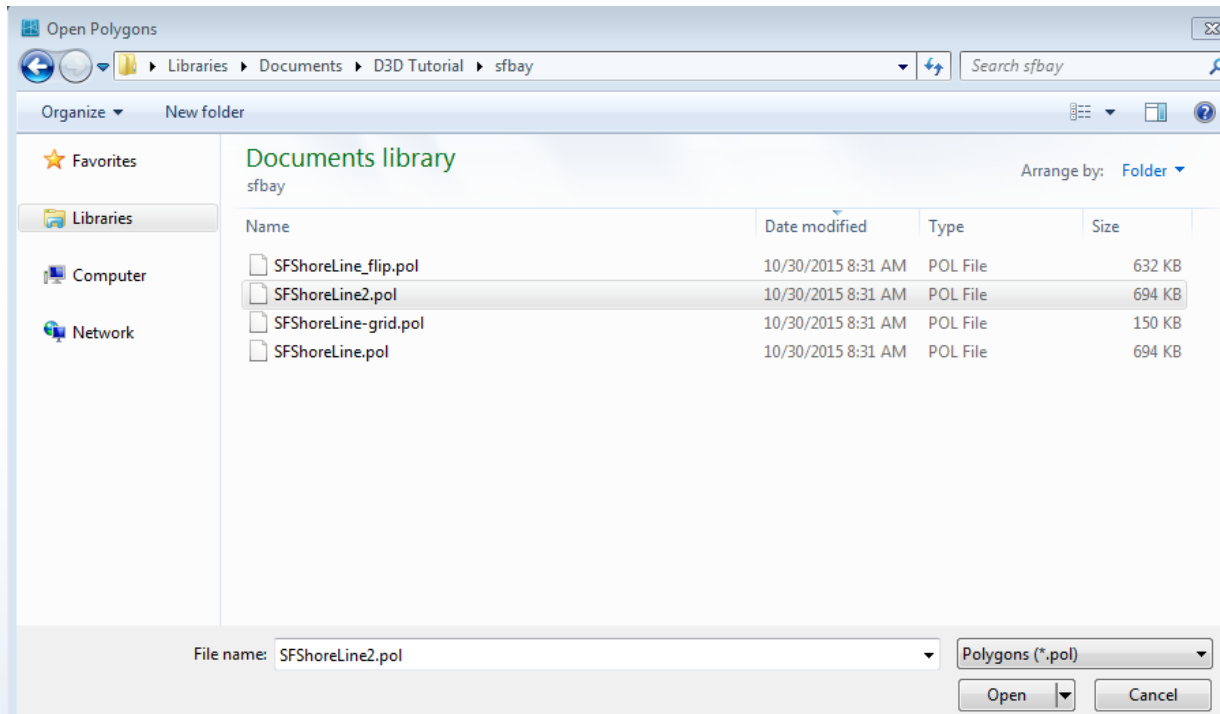
RGFGRID

■ Select File → Attribute Files → Open Polygon



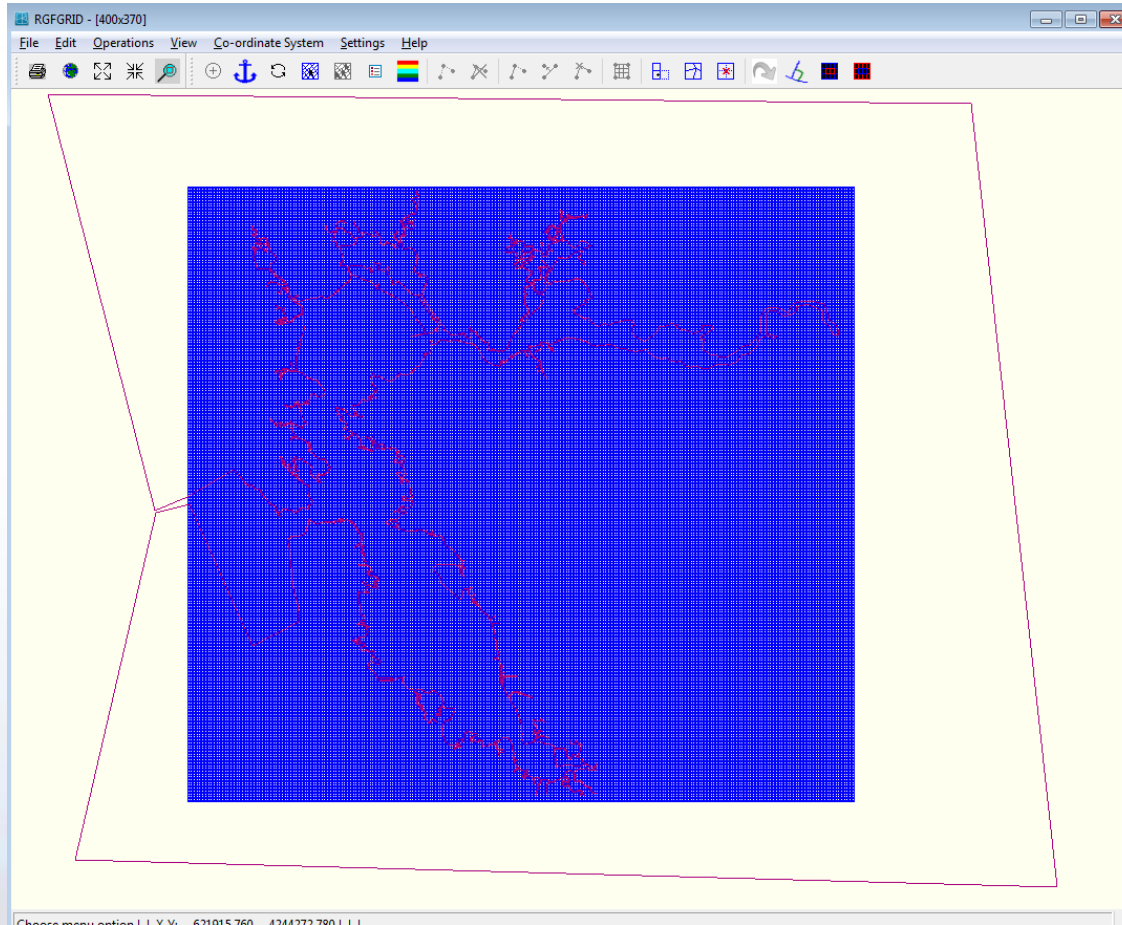
RGFGRID

- Select “SFShoreline2.pol”

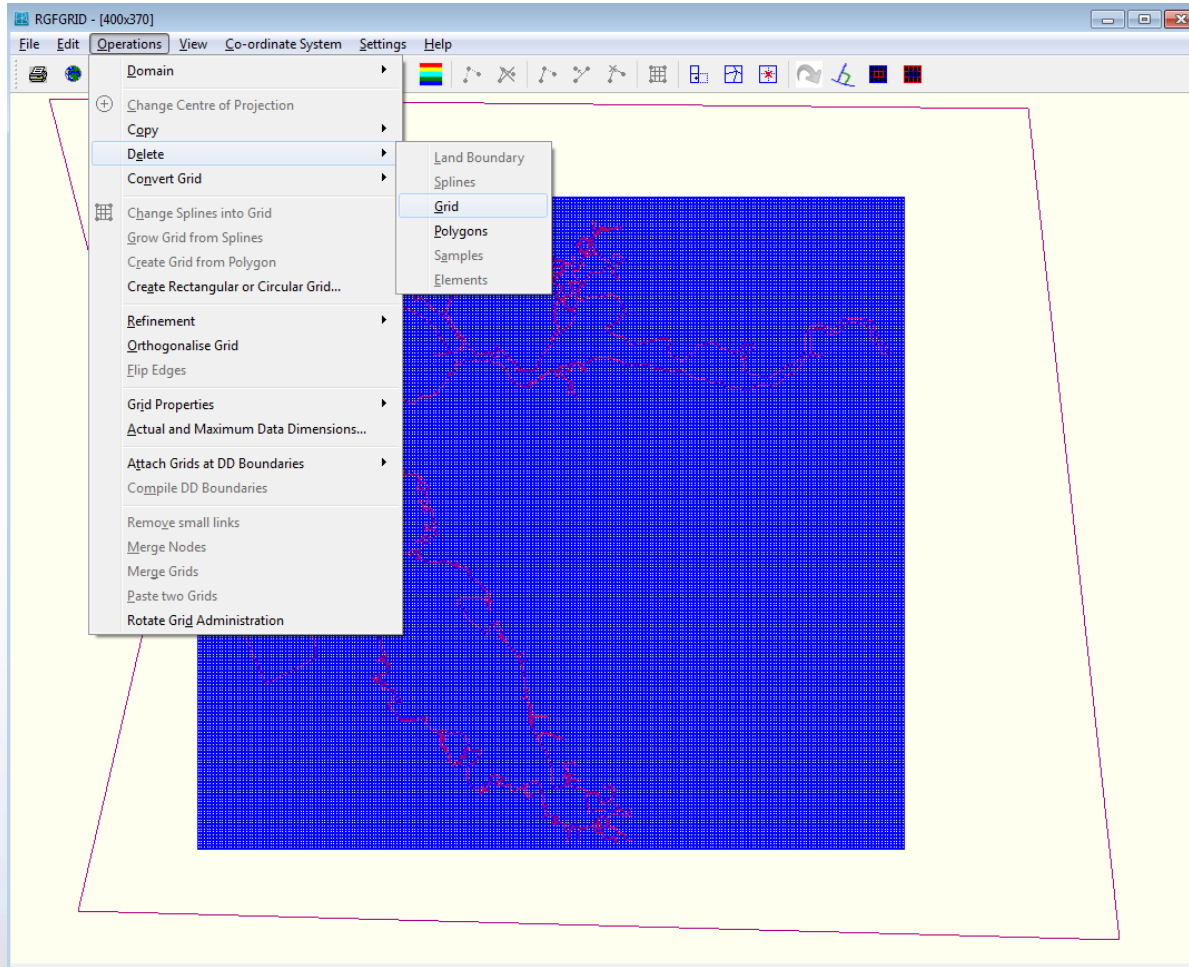


RGFGRID

■ This is the result.



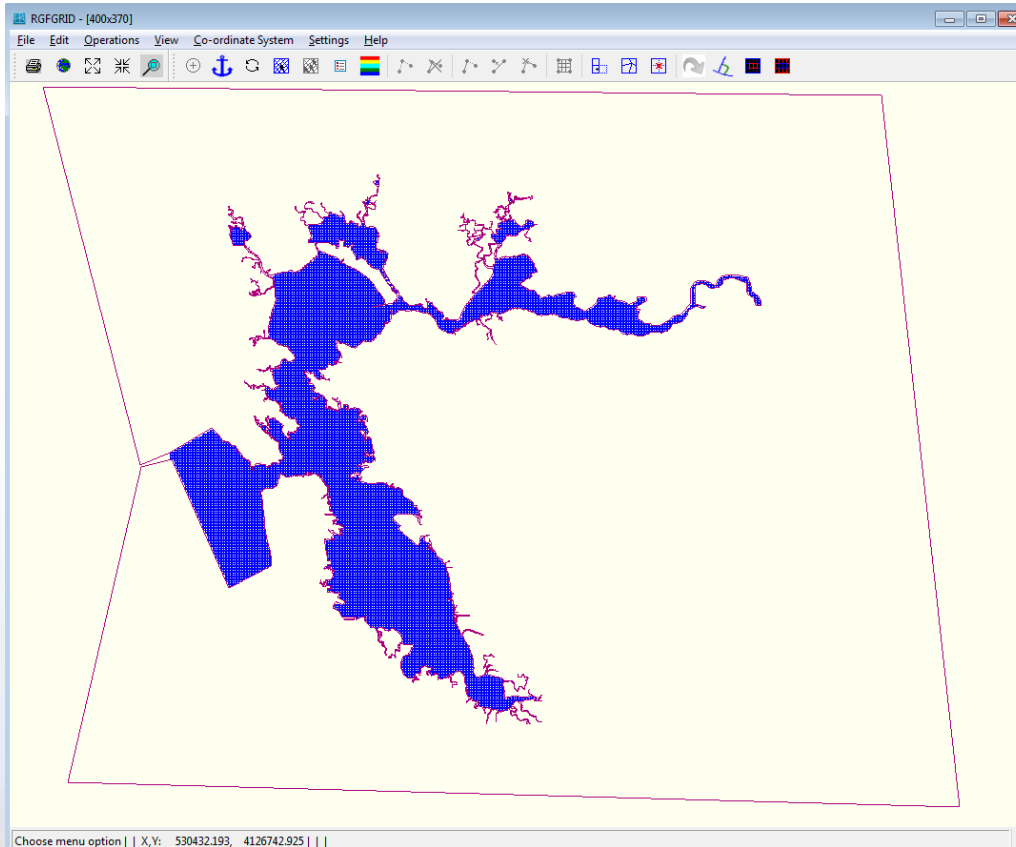
RGFGRID



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

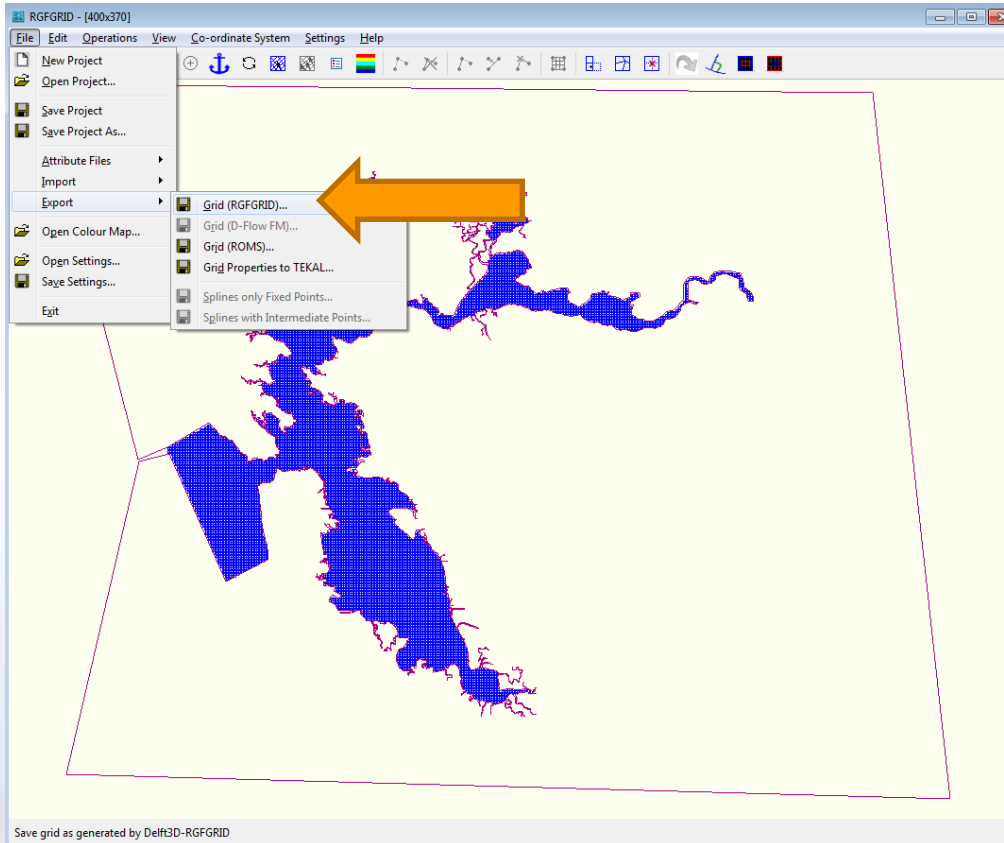
RGFGRID

■ This is the result.



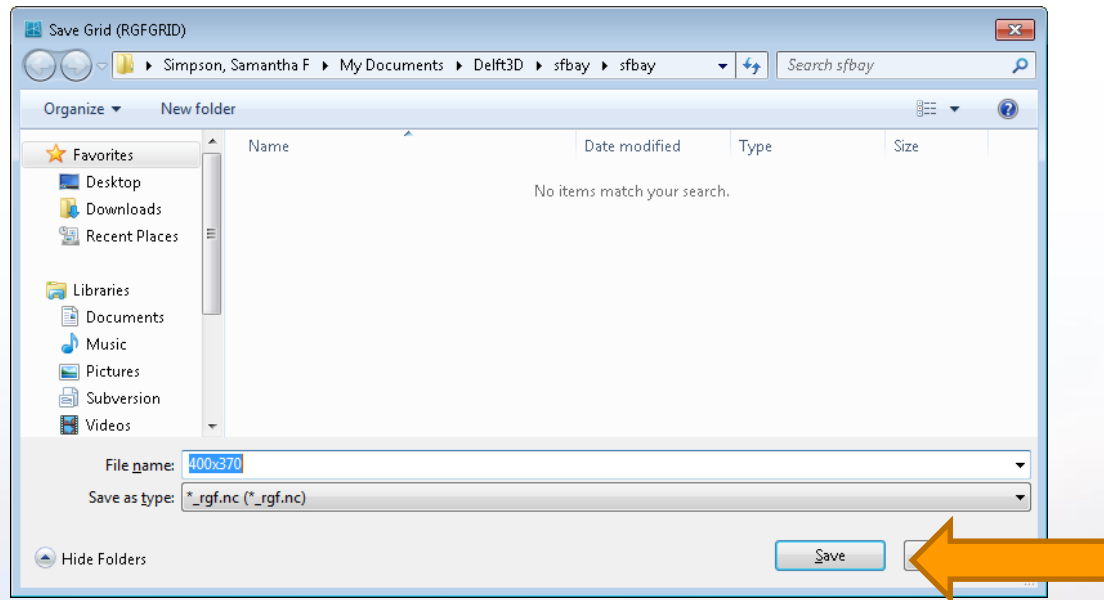
RGFGRID

■ Select File → Export → RGFGRID



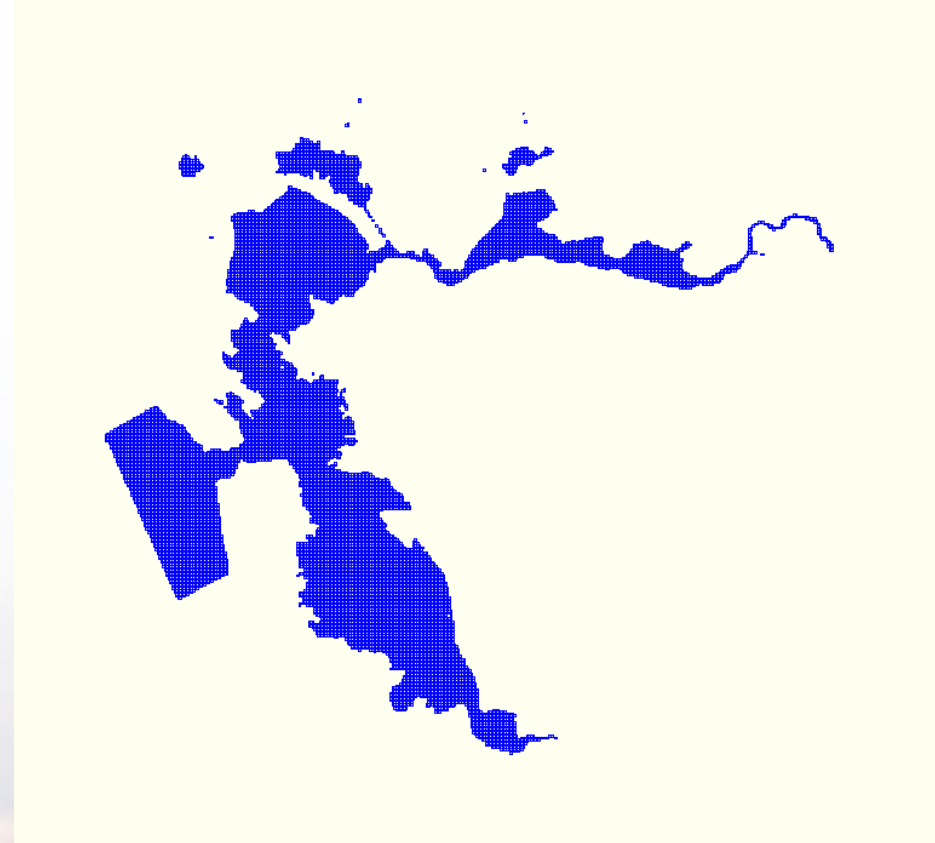
RGFGRID

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



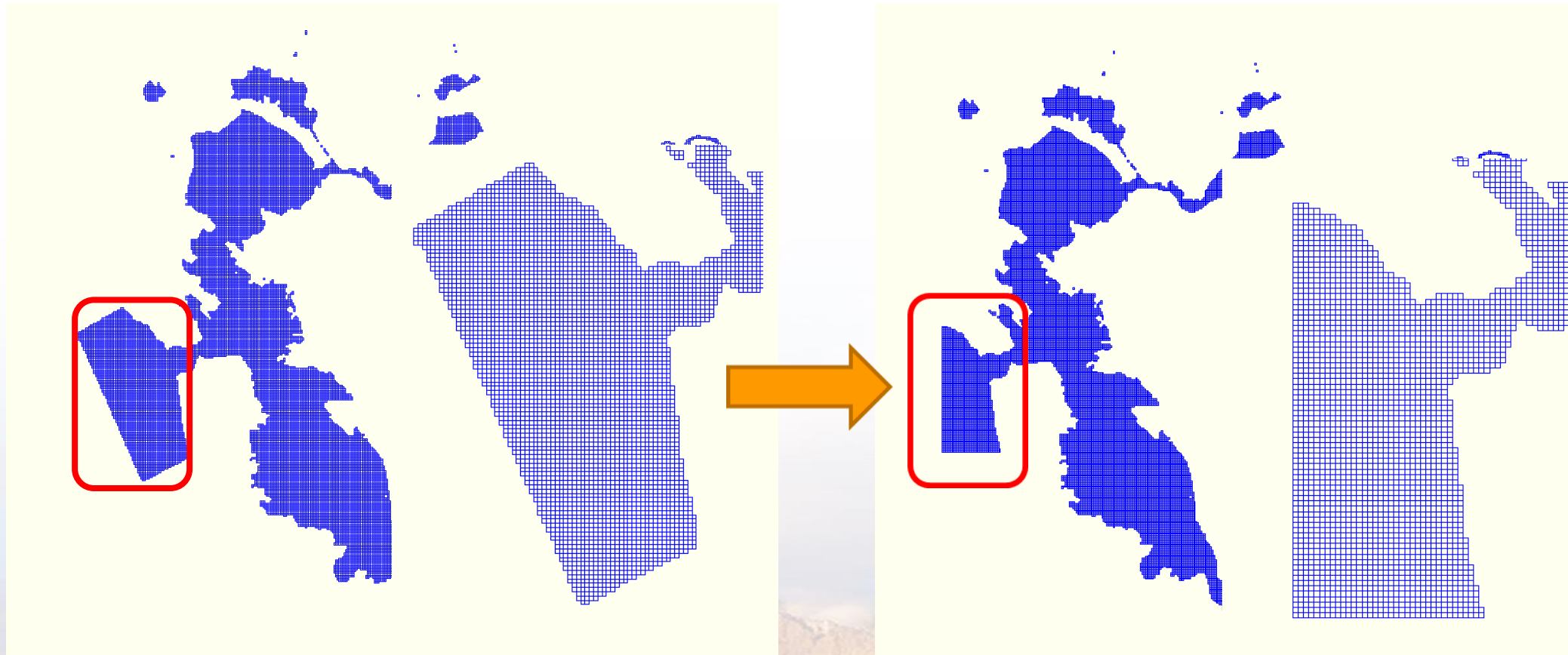
Editing the Grid

- 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.



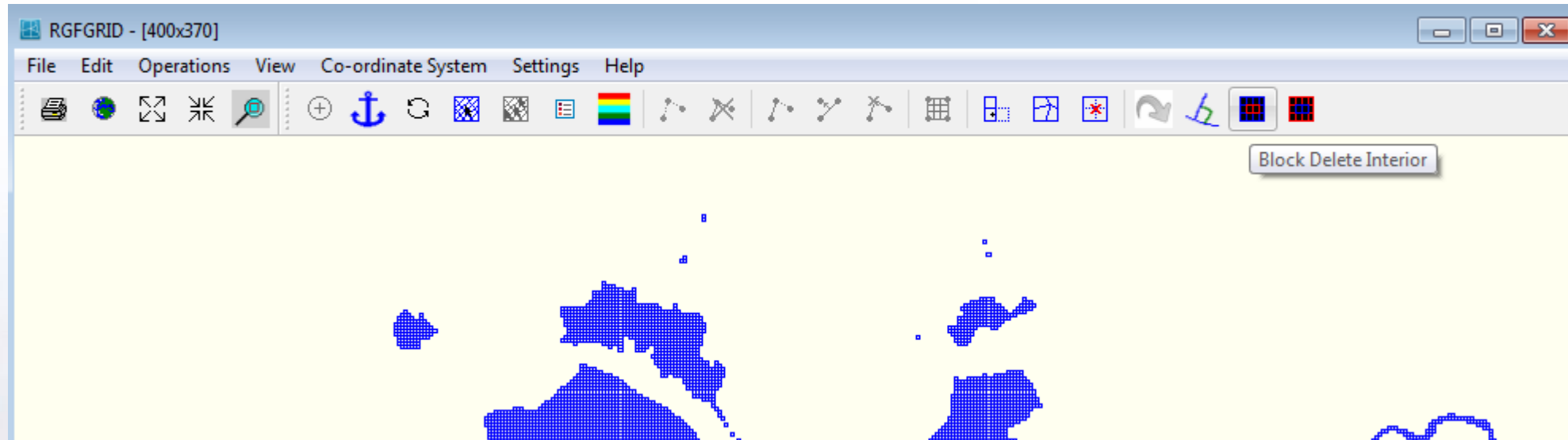
Editing the Grid

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

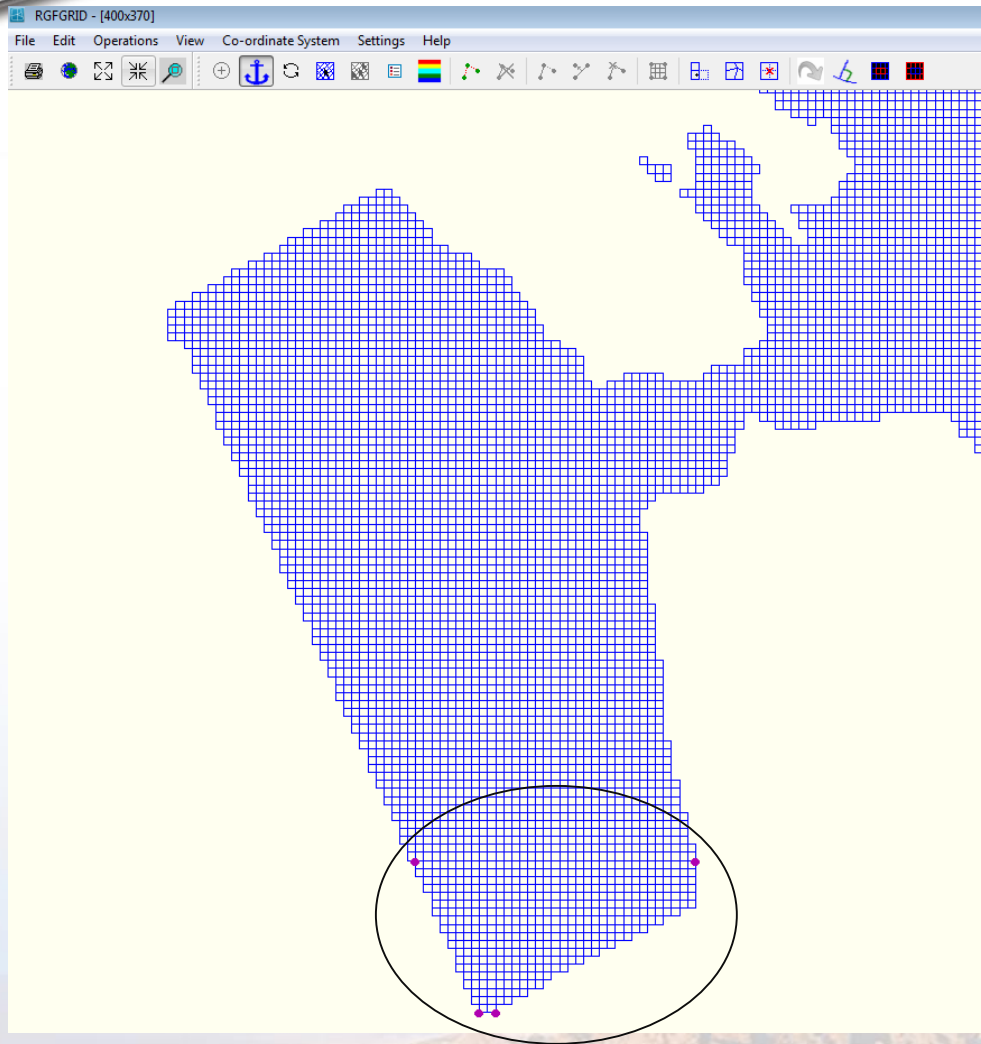


Creating a Linear Boundary

- 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

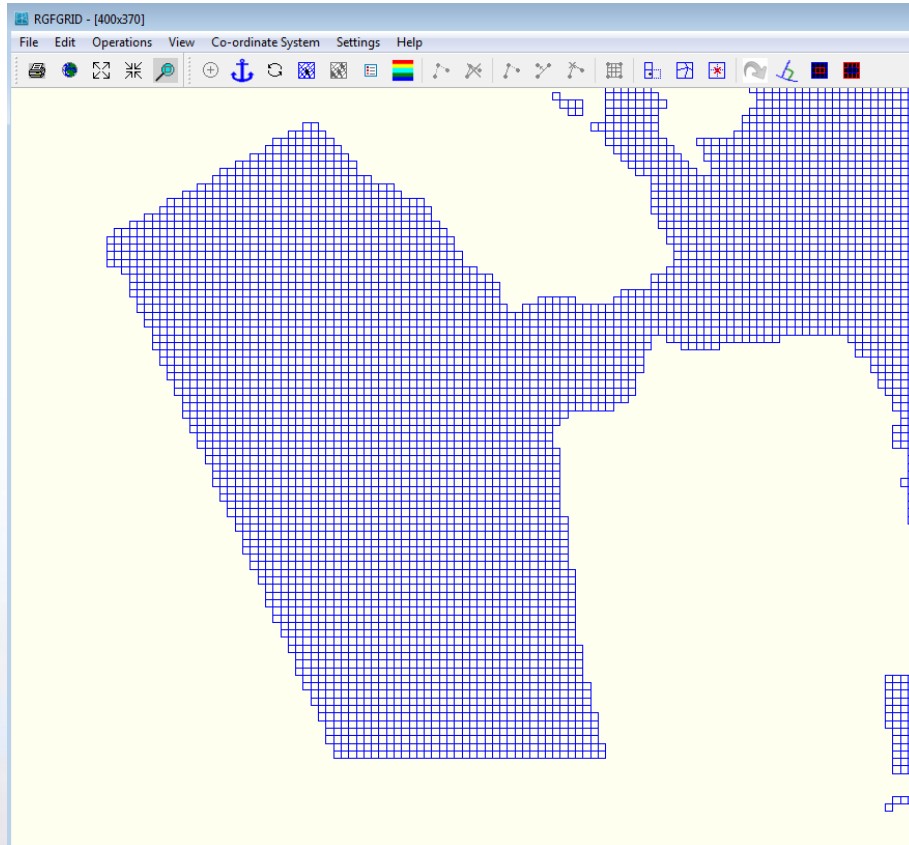


Creating a Linear Boundary



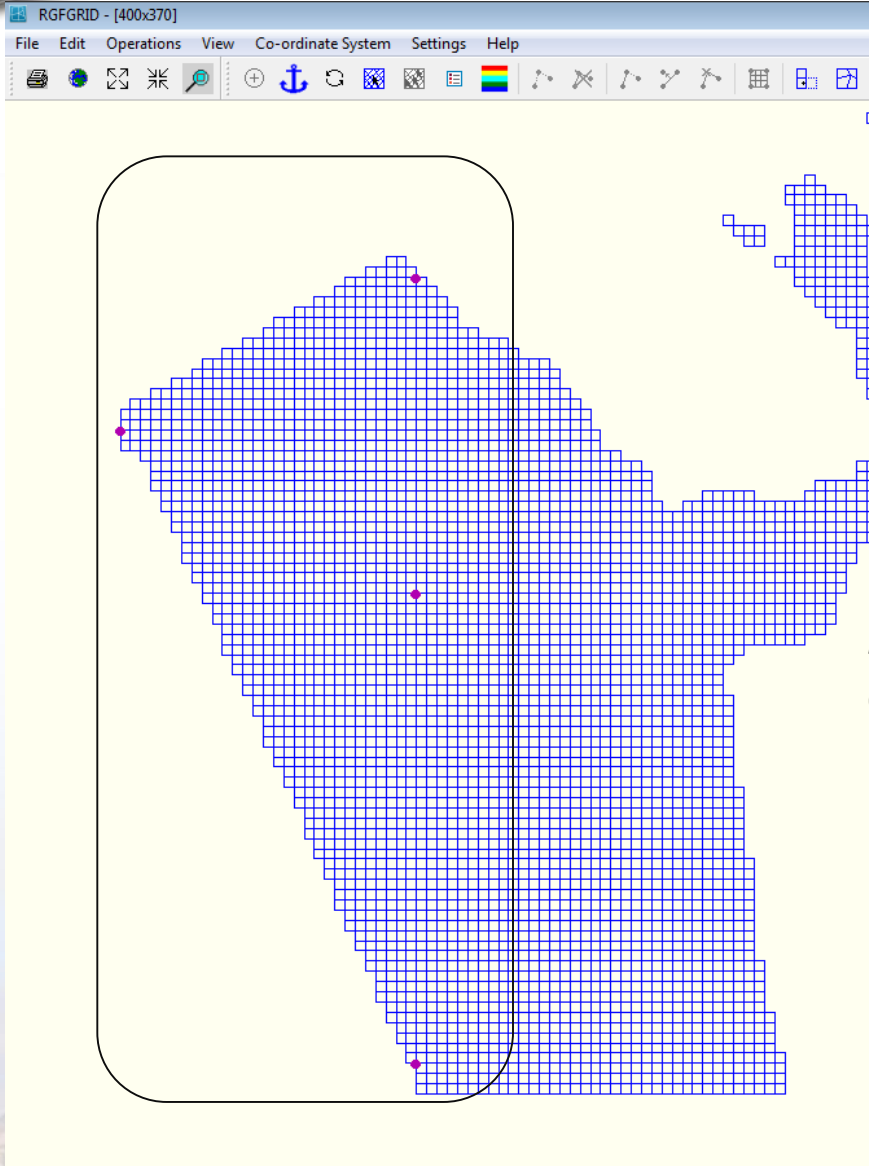
- Create a block by selecting four points on the grid.
- Have top points be on the same line, and have bottom along the bottom corner of the grid.
- Right click to delete the grid points within the selected grid points.

Creating a Linear Boundary

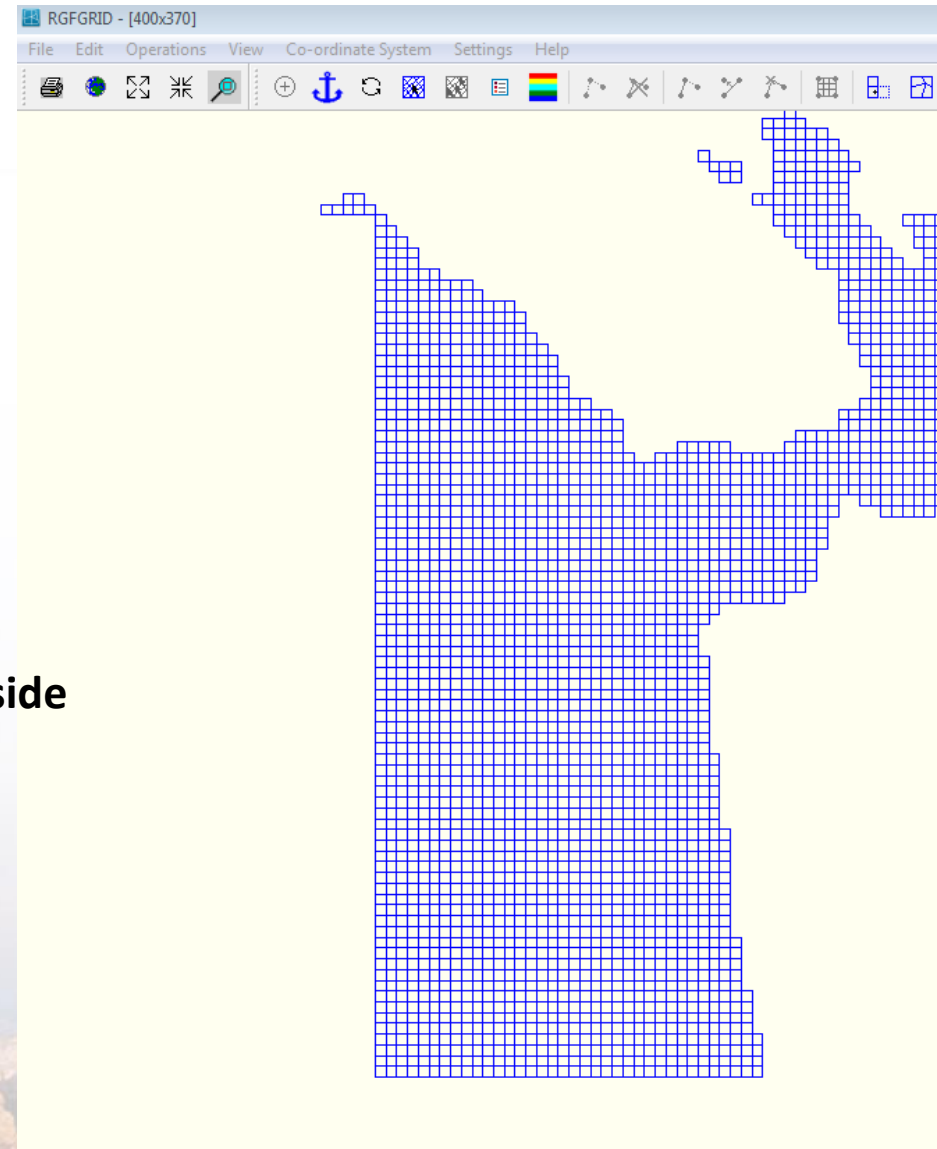


- The bottom should now be smooth.
- Repeat process for left side and top.

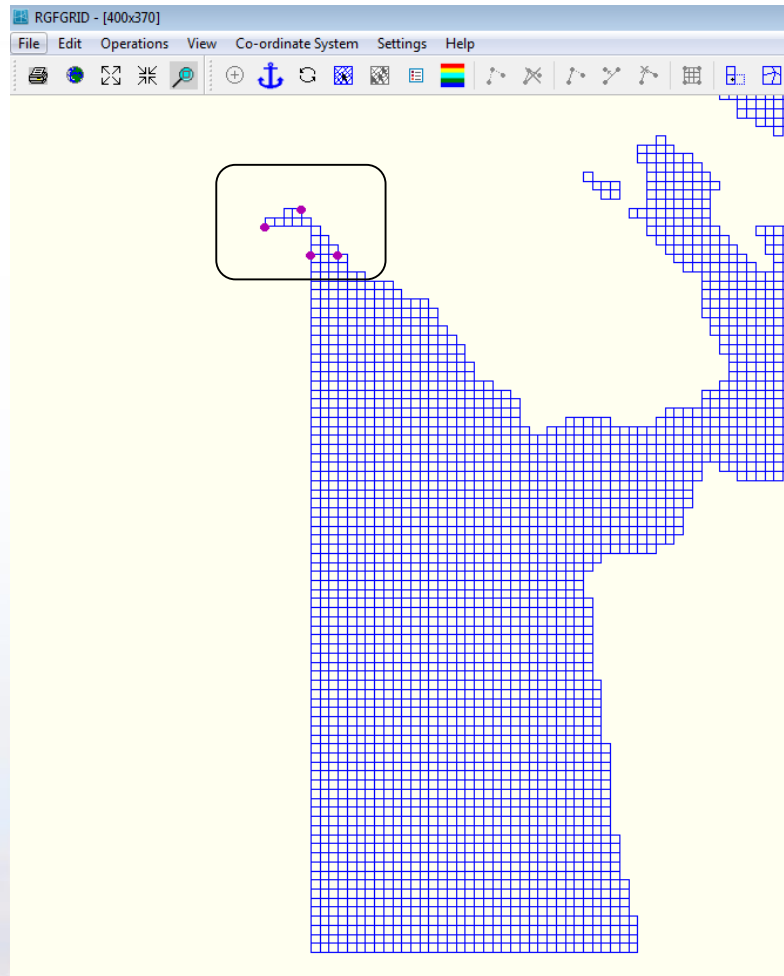
Delete Left cells



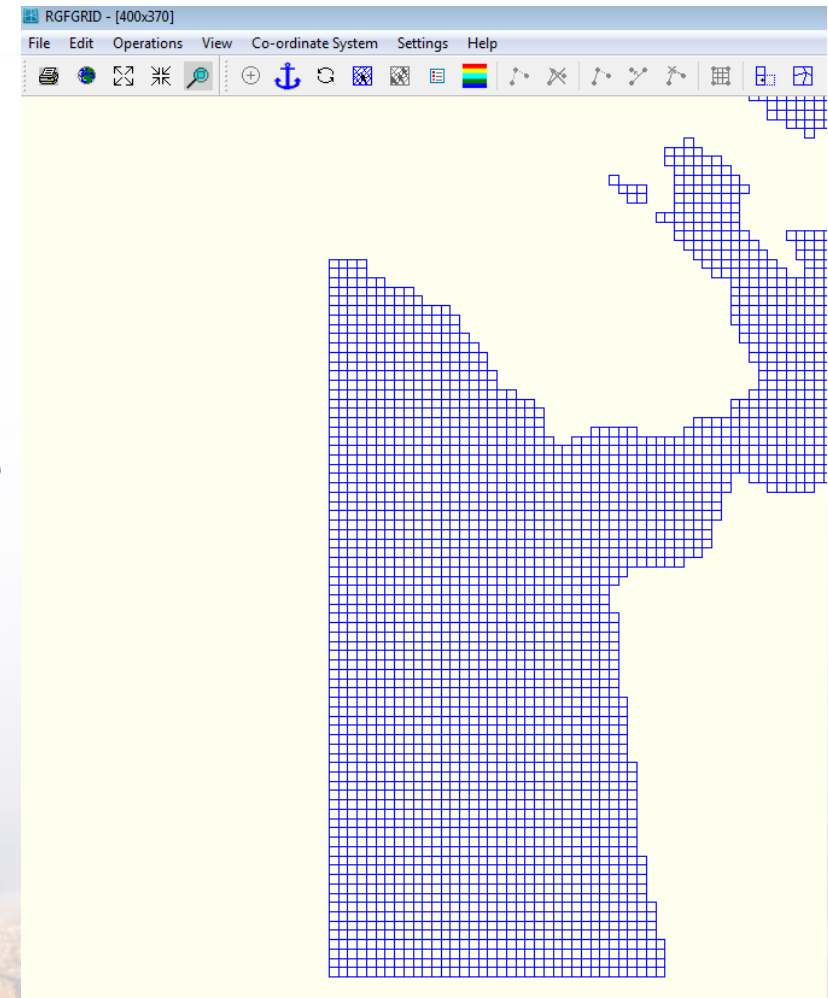
**Right click to
delete cells inside
blocked area.**



Delete Cells on top

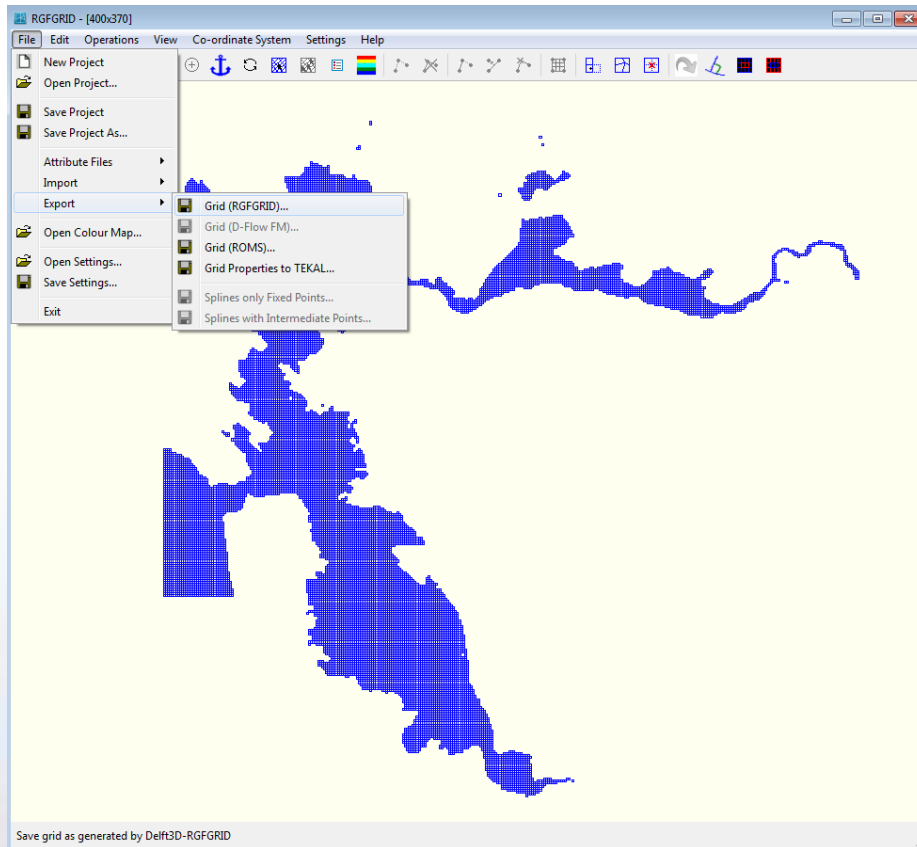


**Right click to
delete cells inside
blocked area.**

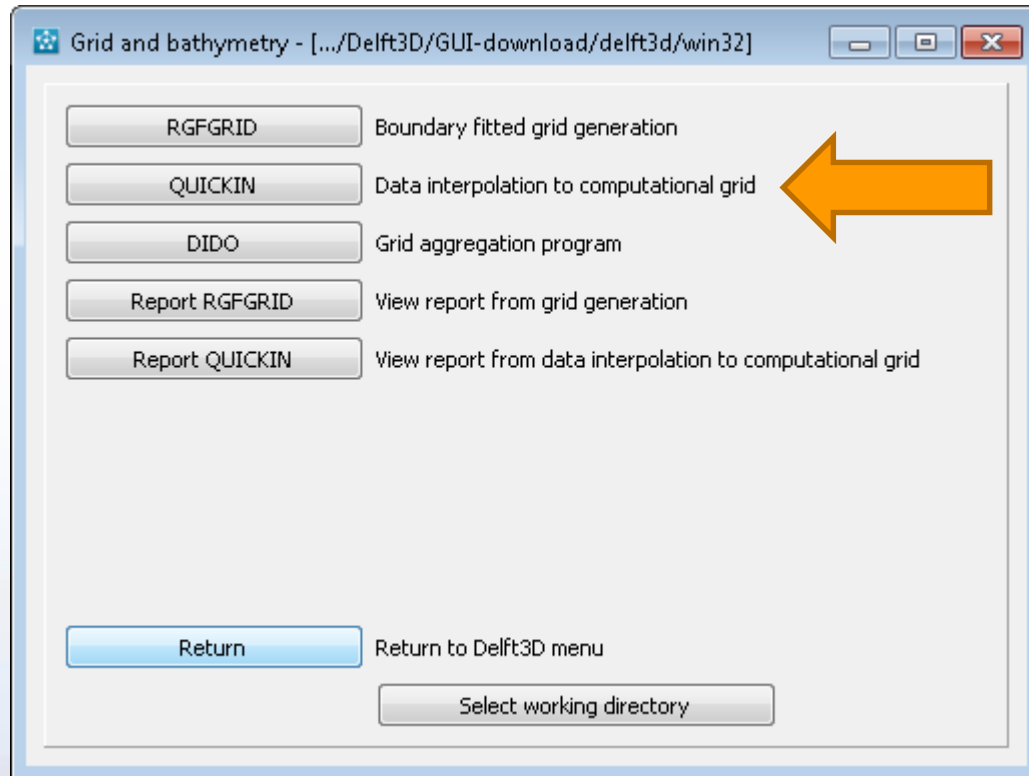


Saving Your Grid

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

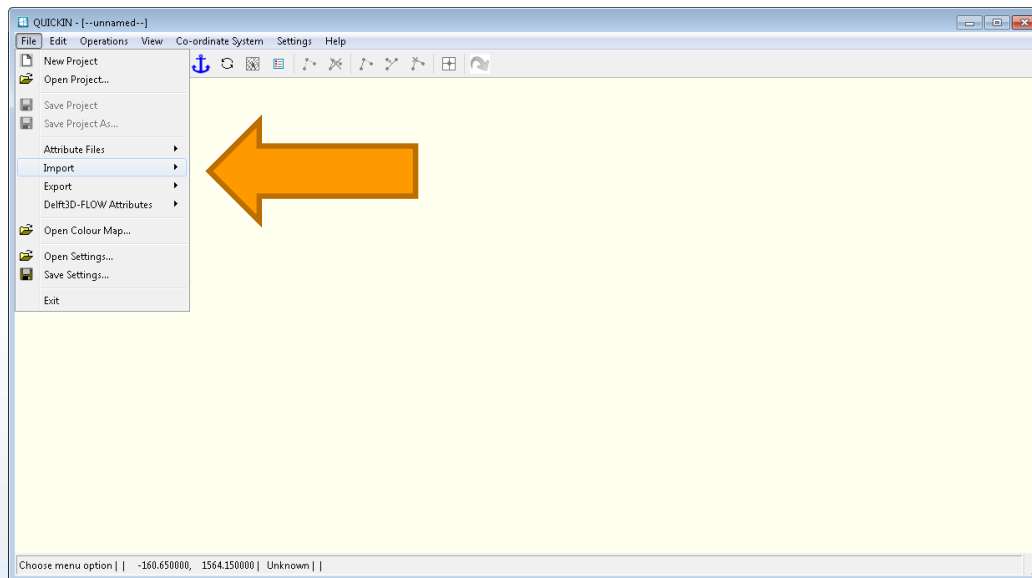


QUICKIN



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

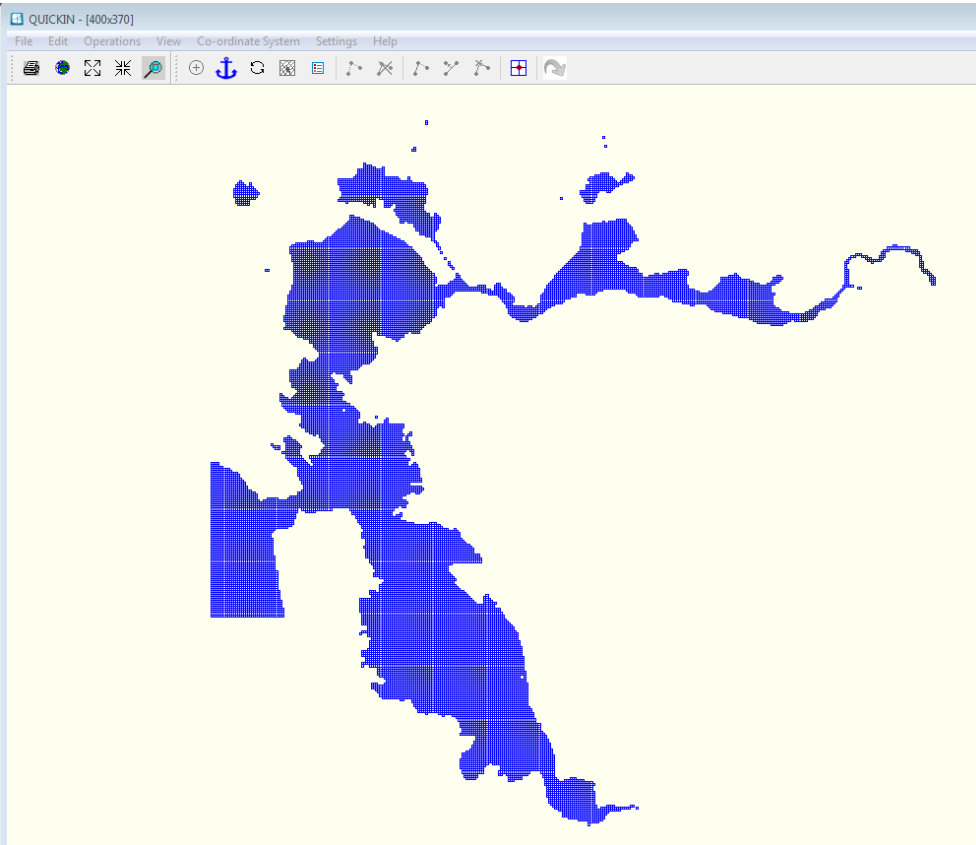
QUICKIN



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

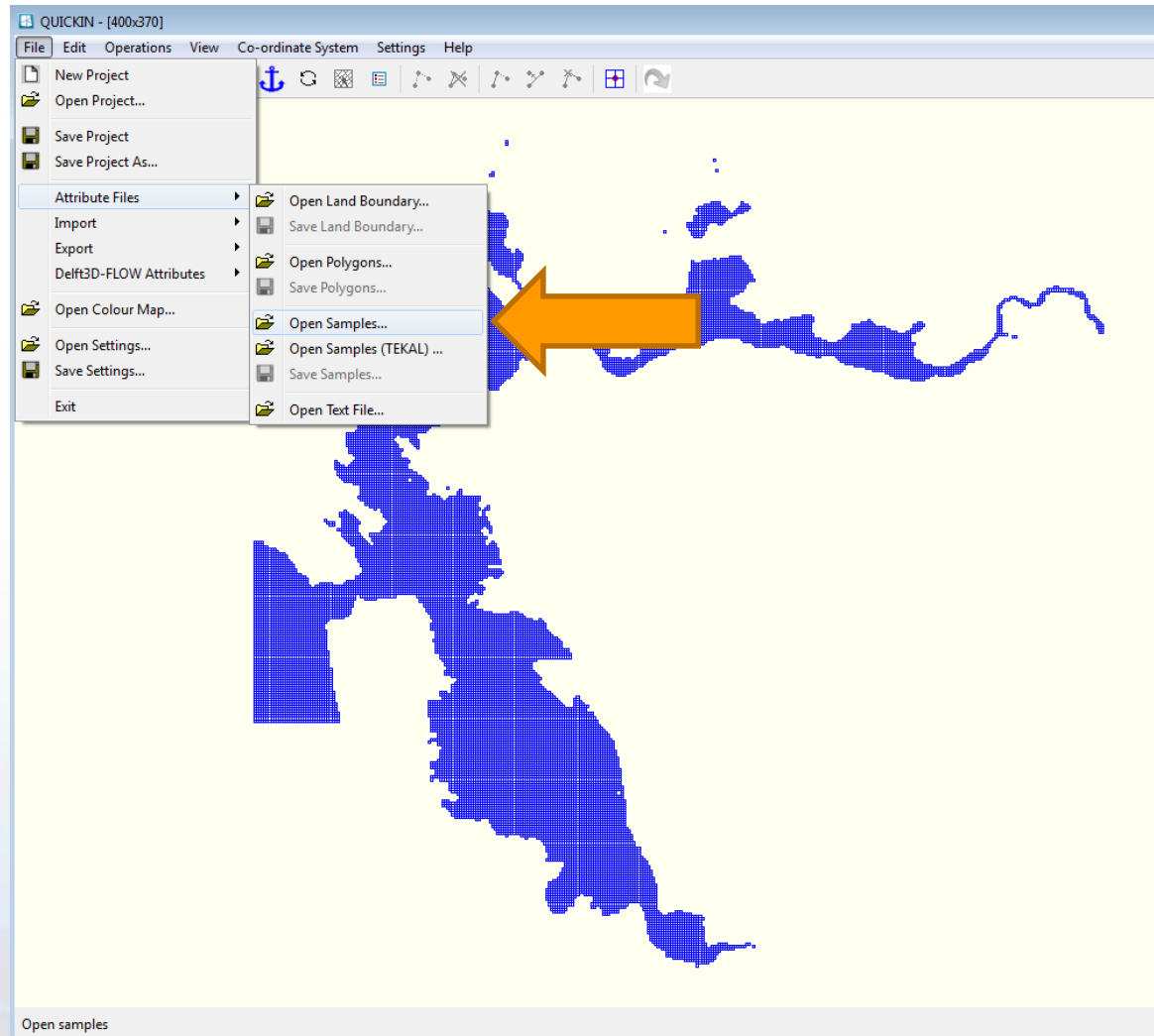
QUICKIN

- This is what you will see.

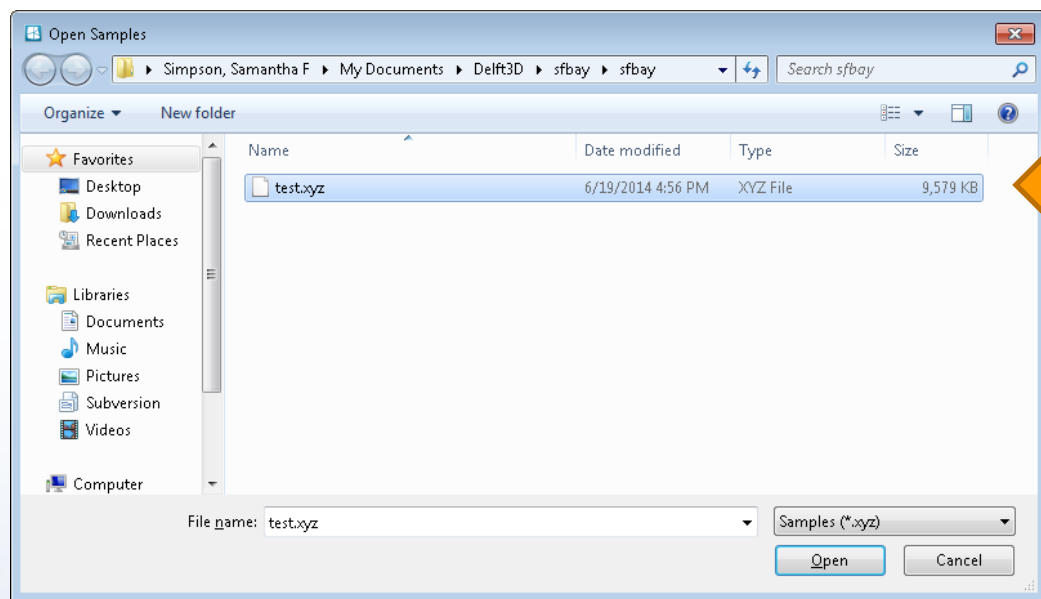


QUICKIN

- Select File → Attribute Files → Open Samples



Adding a Sample File

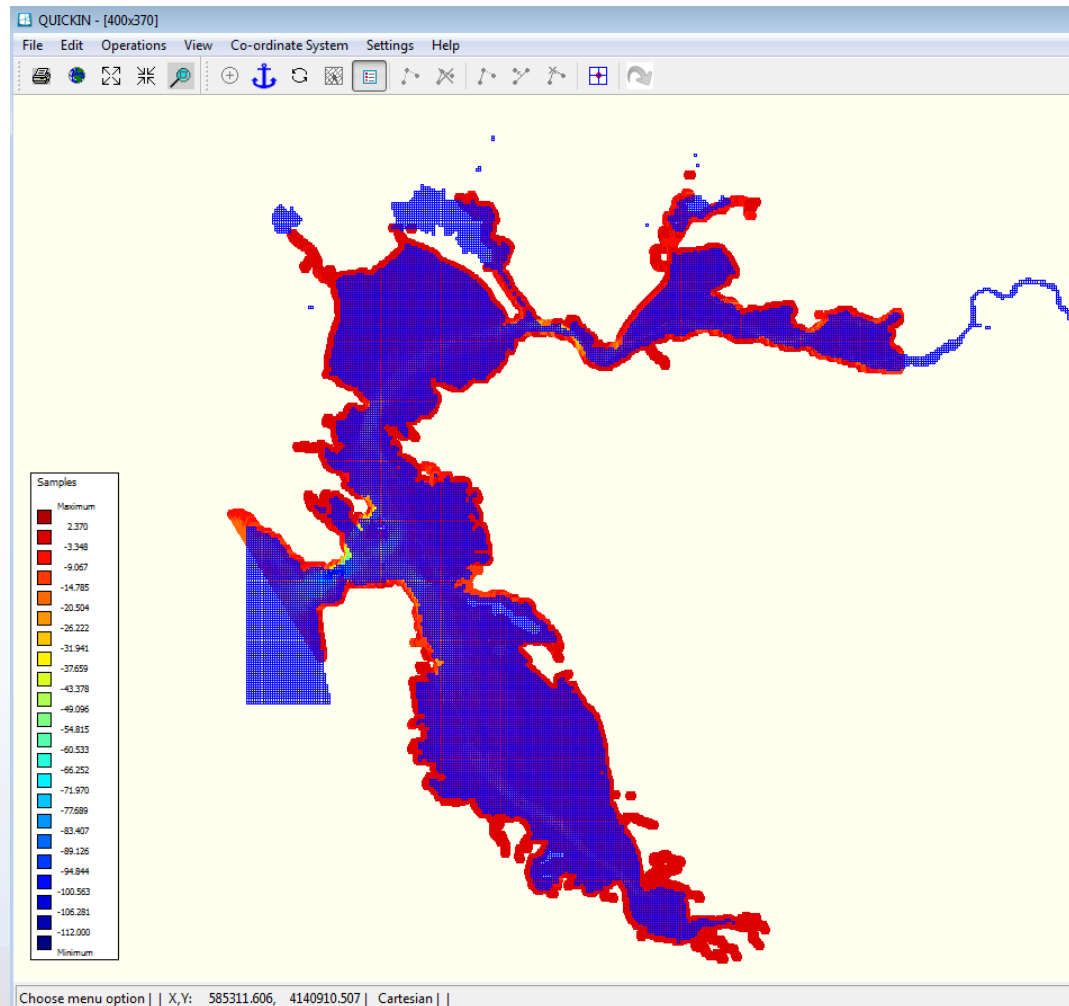


- Select test.xyz

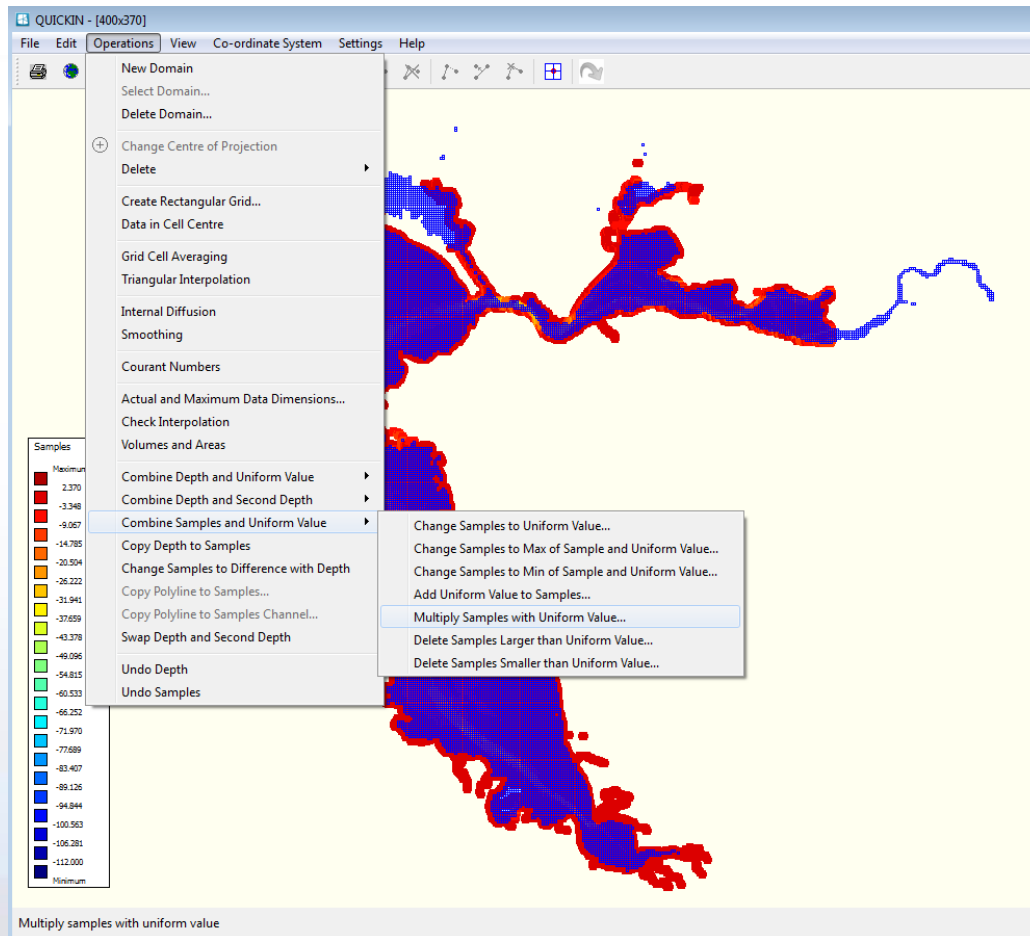
- test.xyz contains data samples at each (x,y) coordinate on the grid.

Adding a Sample File

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

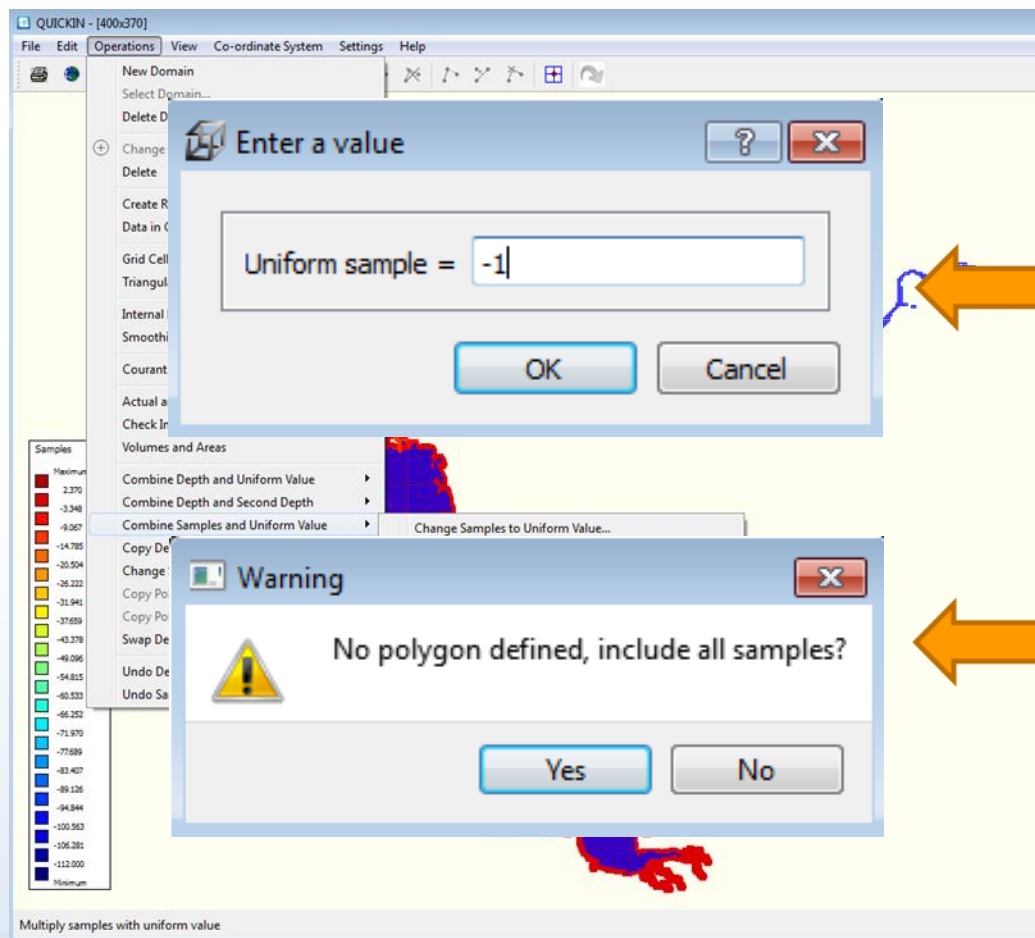


Adding a Sample File



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

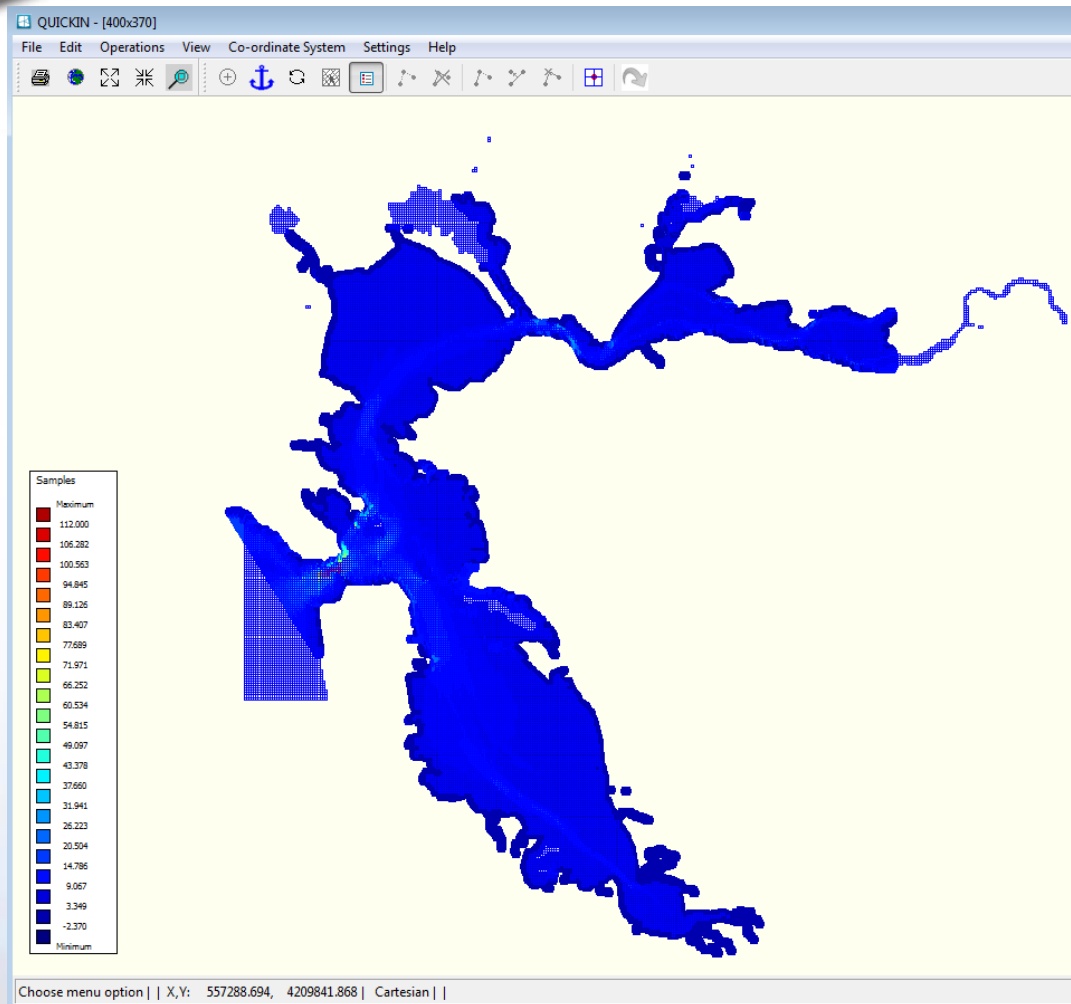
Adding a Sample File



■ Change from -999 to -1, click OK.

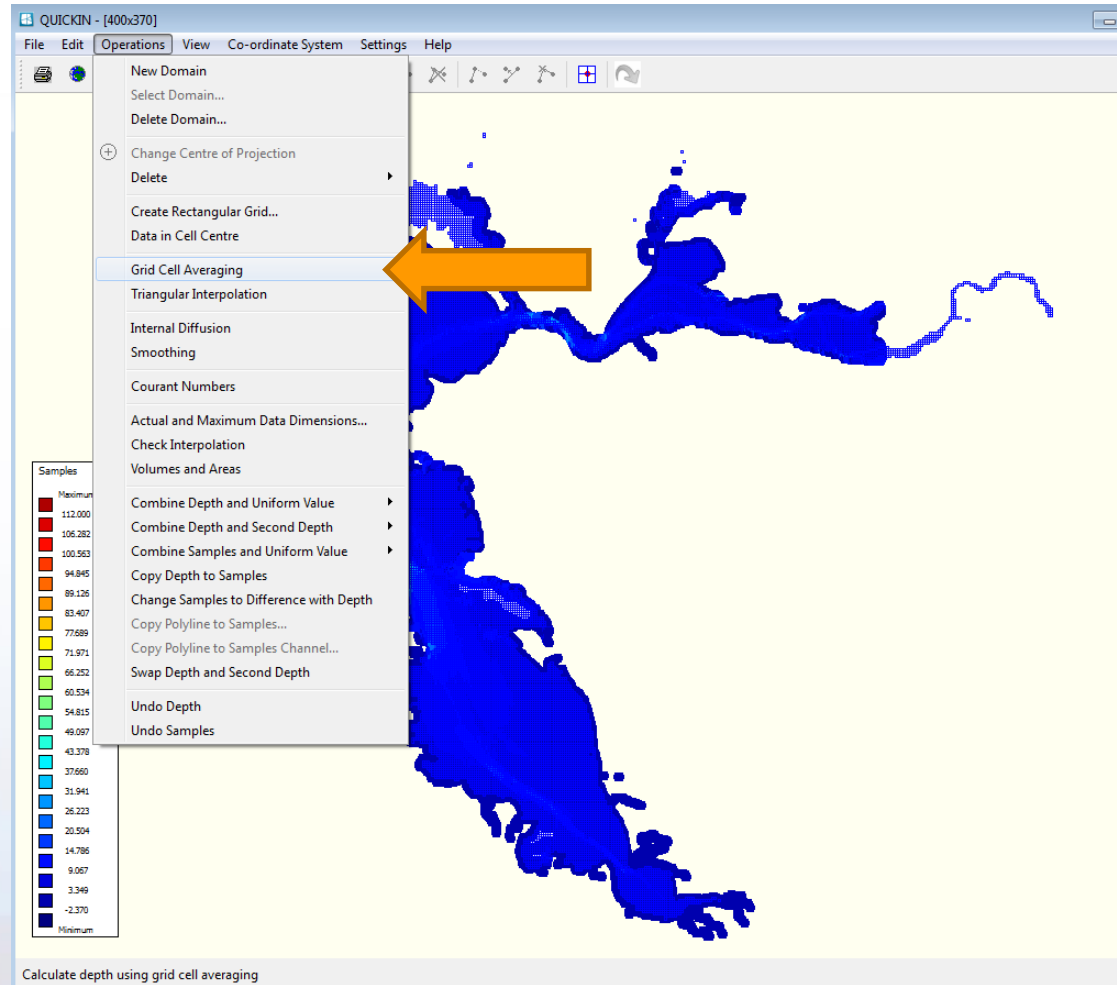
■ A warning window will pop up.
Click Yes.

QUICKIN



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

Grid Cell Averaging

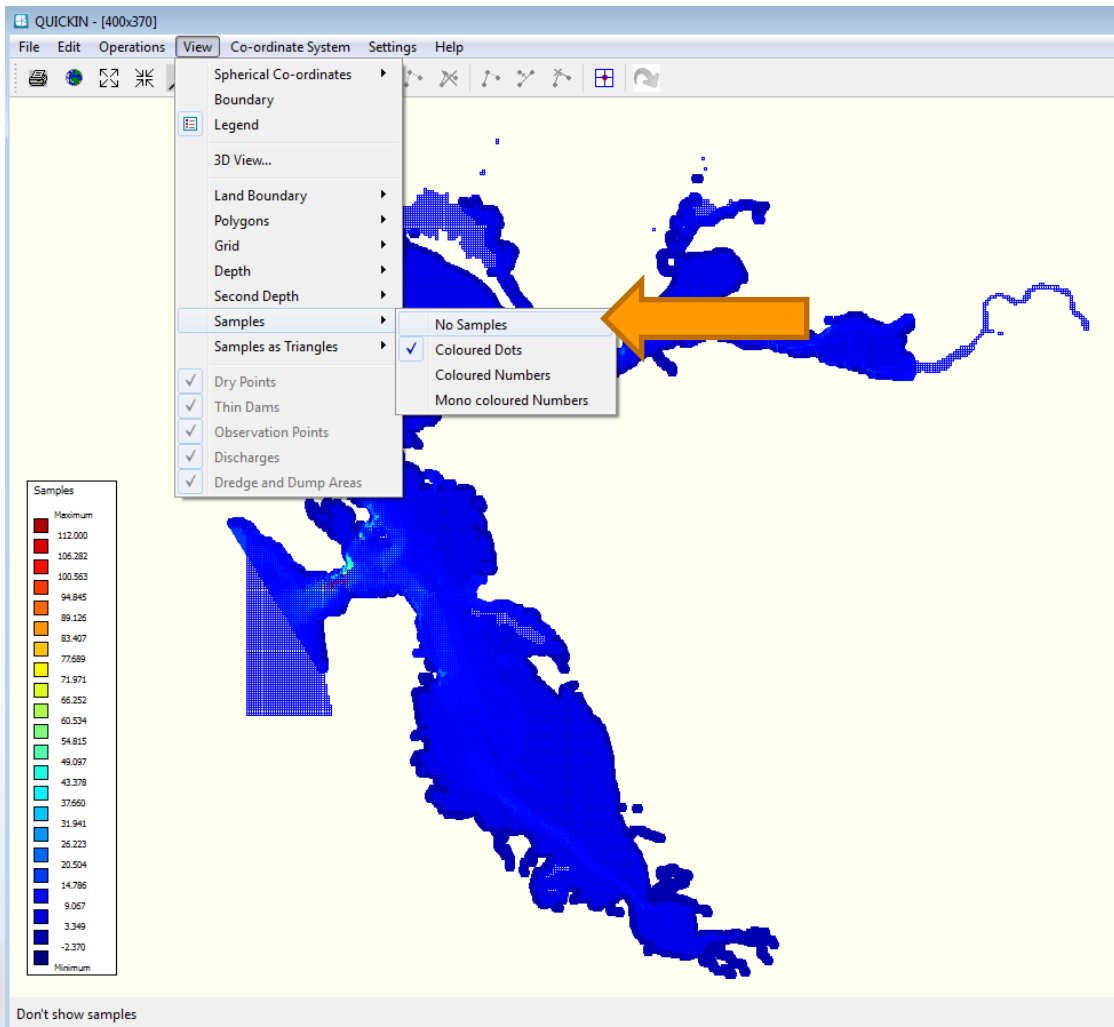


- Select Operations → Grid Cell Averaging

- This will interpolate the samples on to the grid.

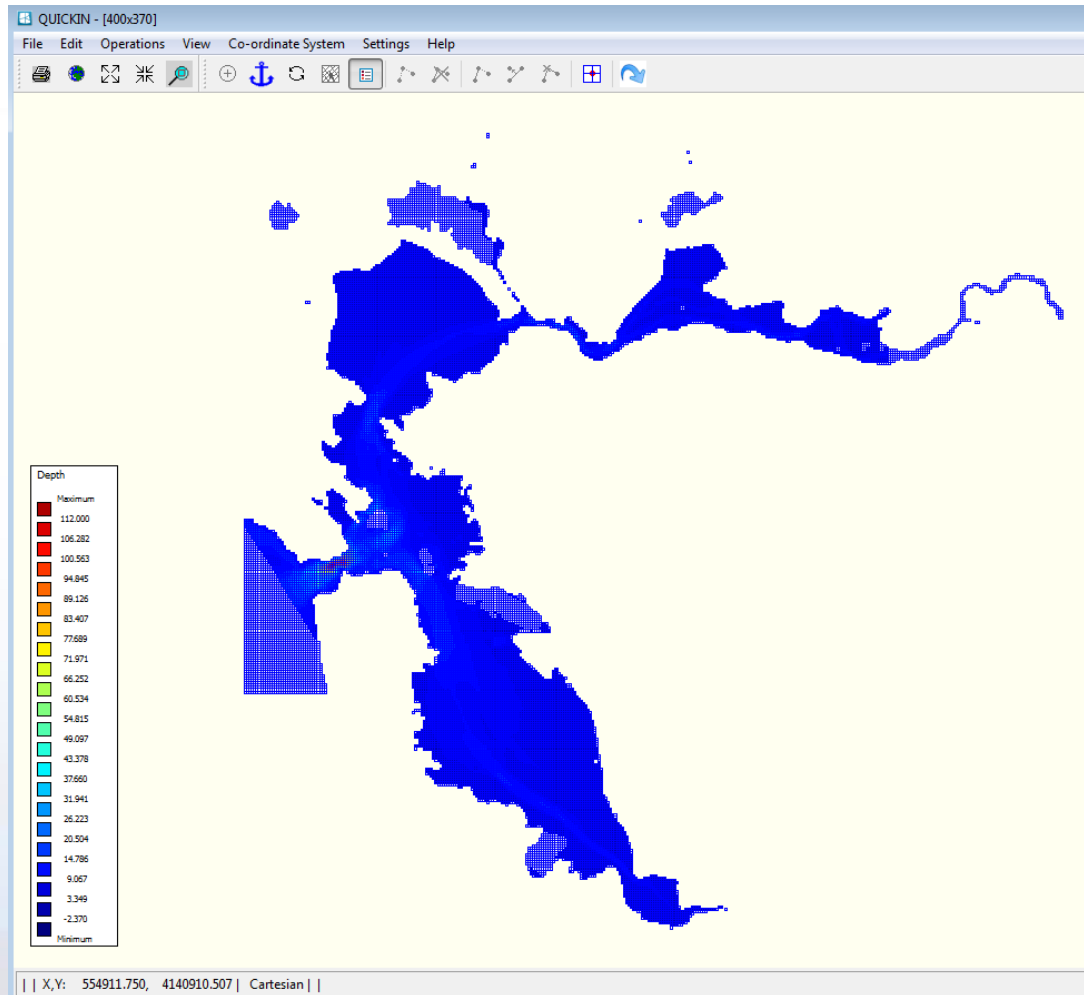
Sample Removal

- Select View → Samples → No Samples

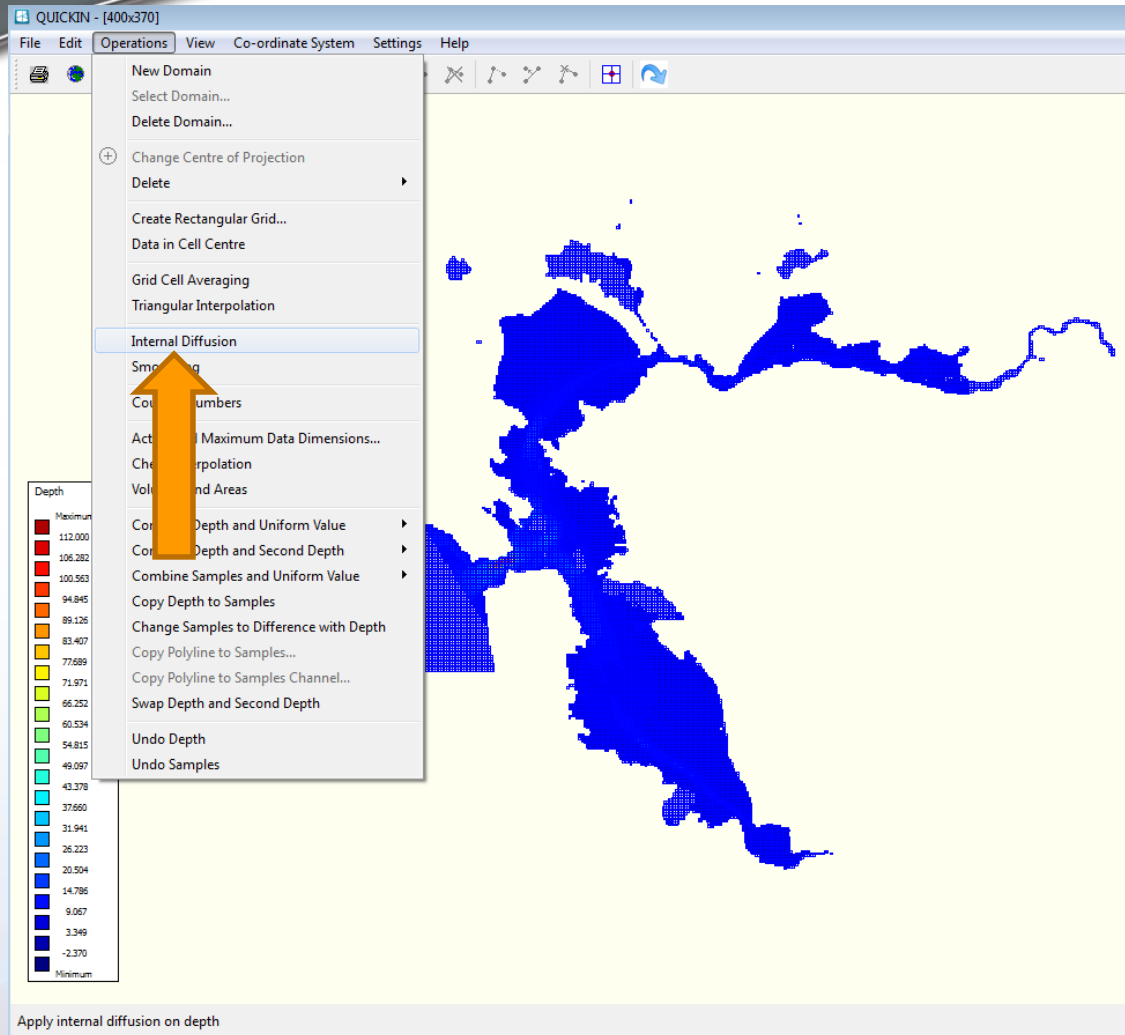


Sample Removal

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

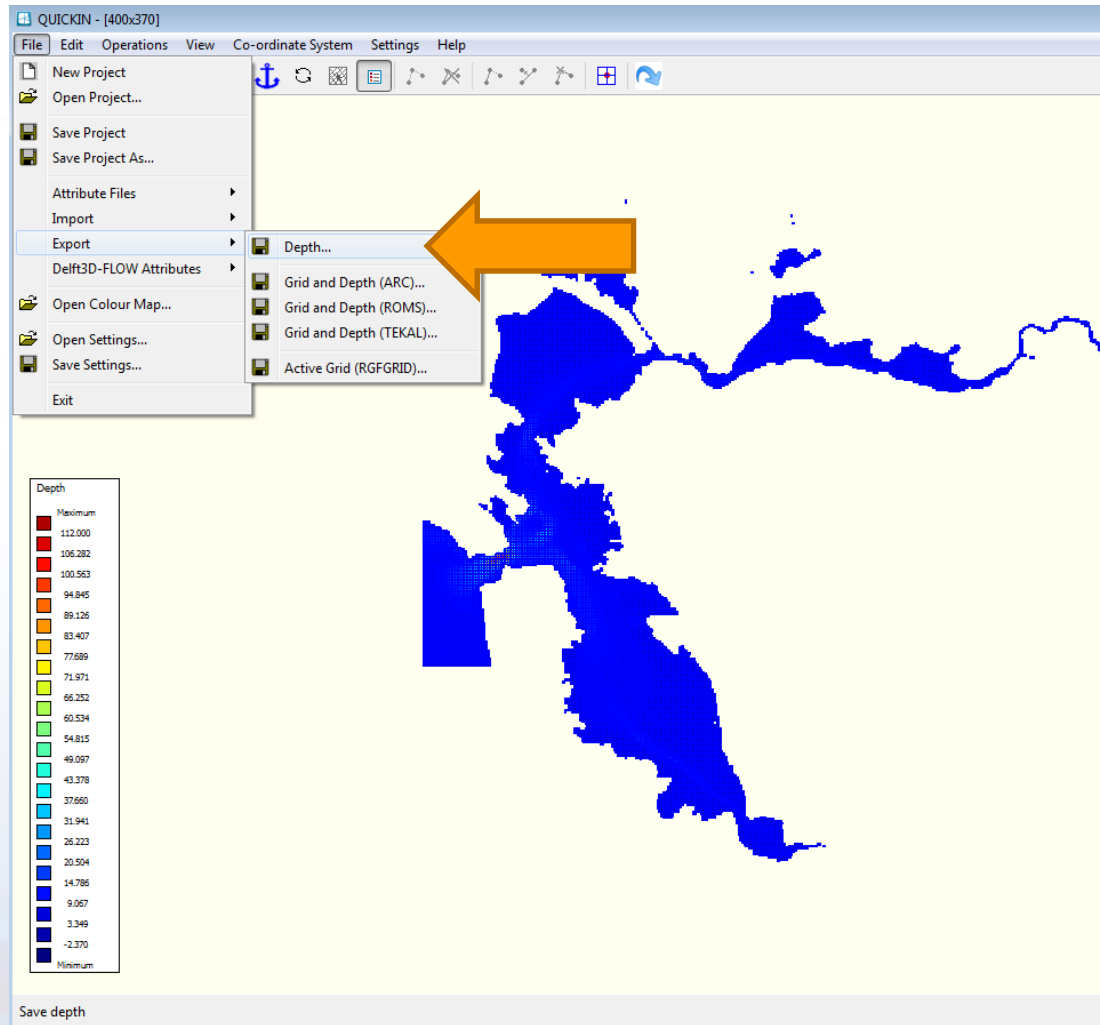


QUICKIN



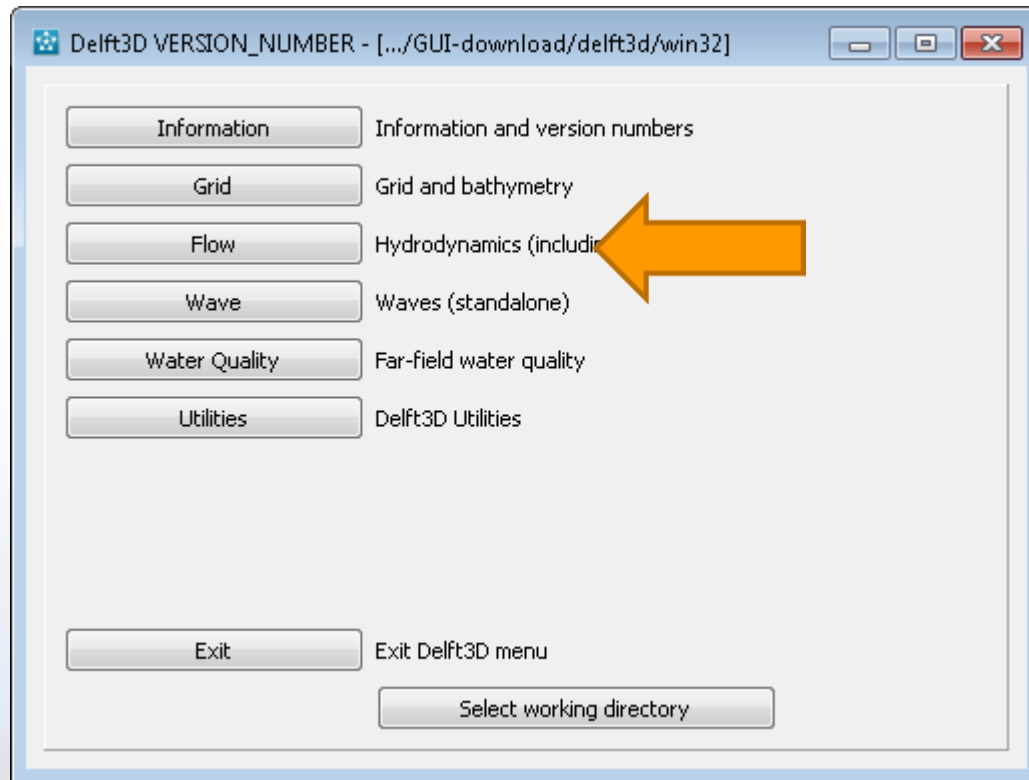
- Select Operations → Internal Diffusion
- This will extrapolate the depths to the remaining unpopulated cells.

QUICKIN



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

Flow

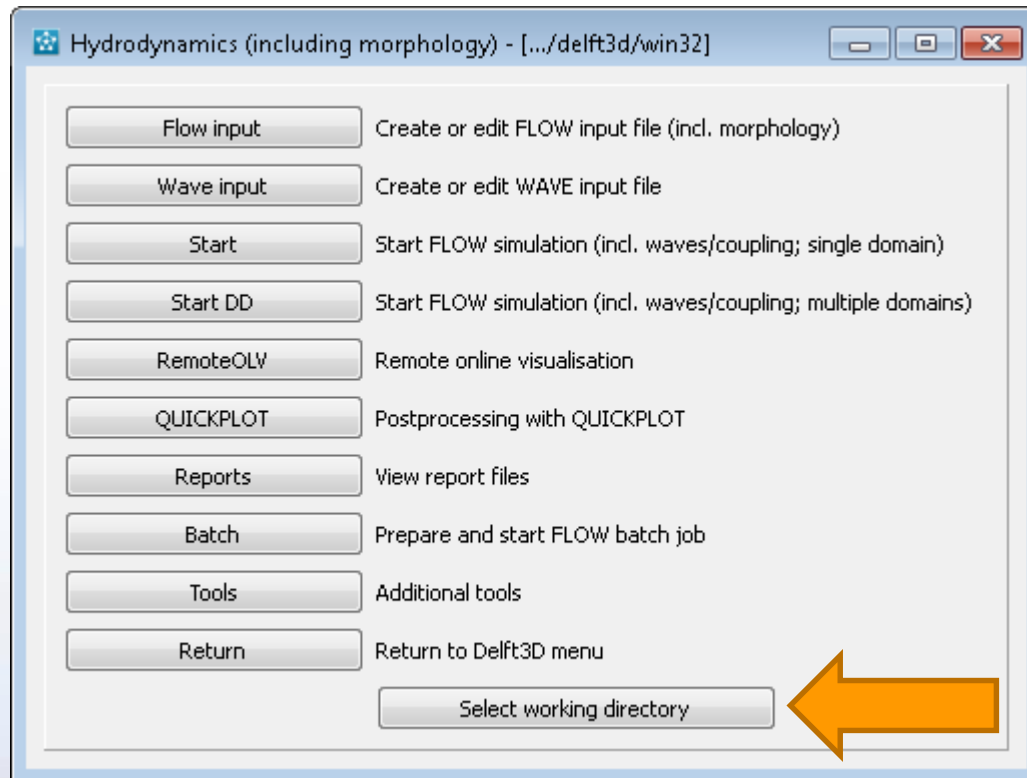


- Return to the main Delft 3D menu.

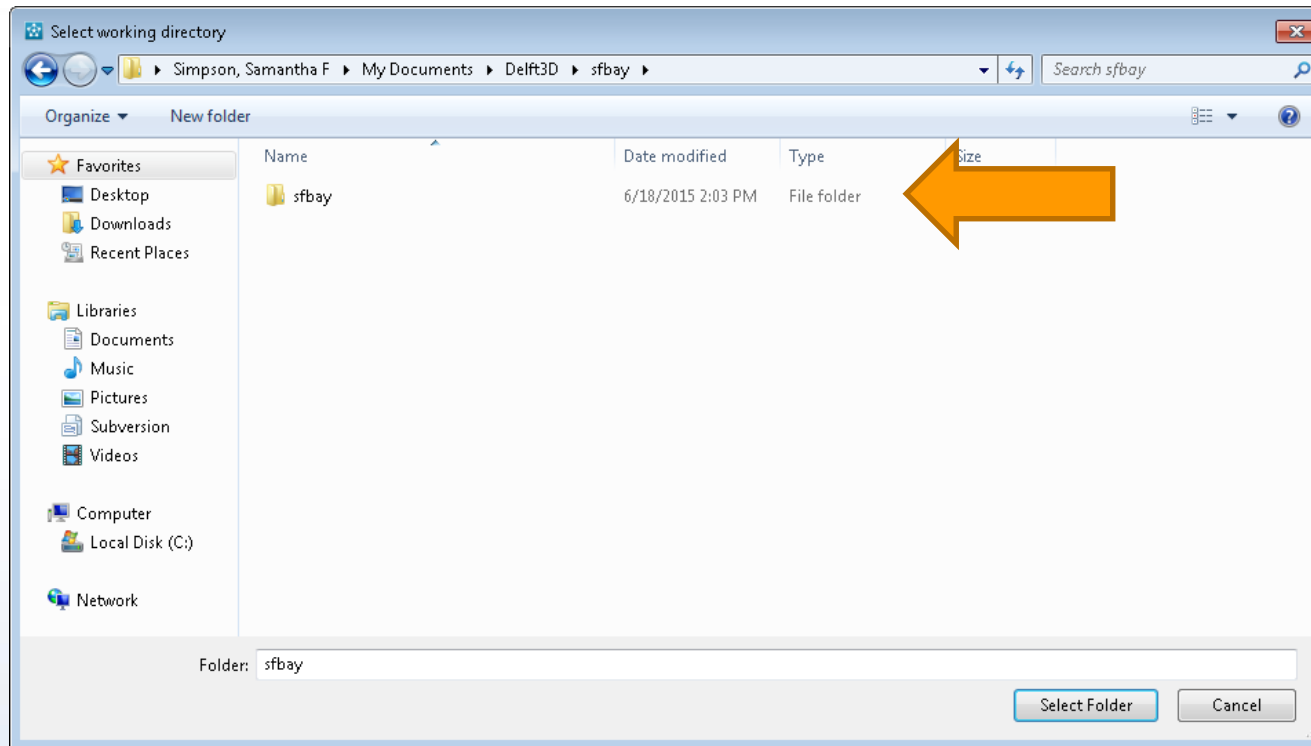
- Select Flow

Flow

■ Select Working Directory

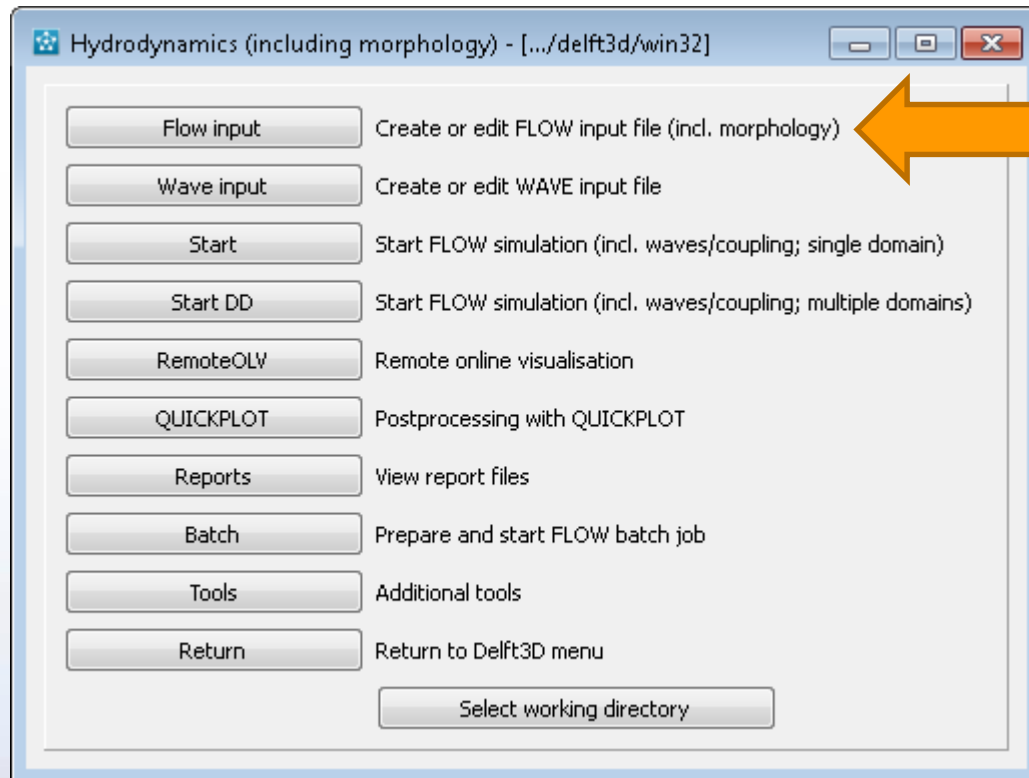


Flow



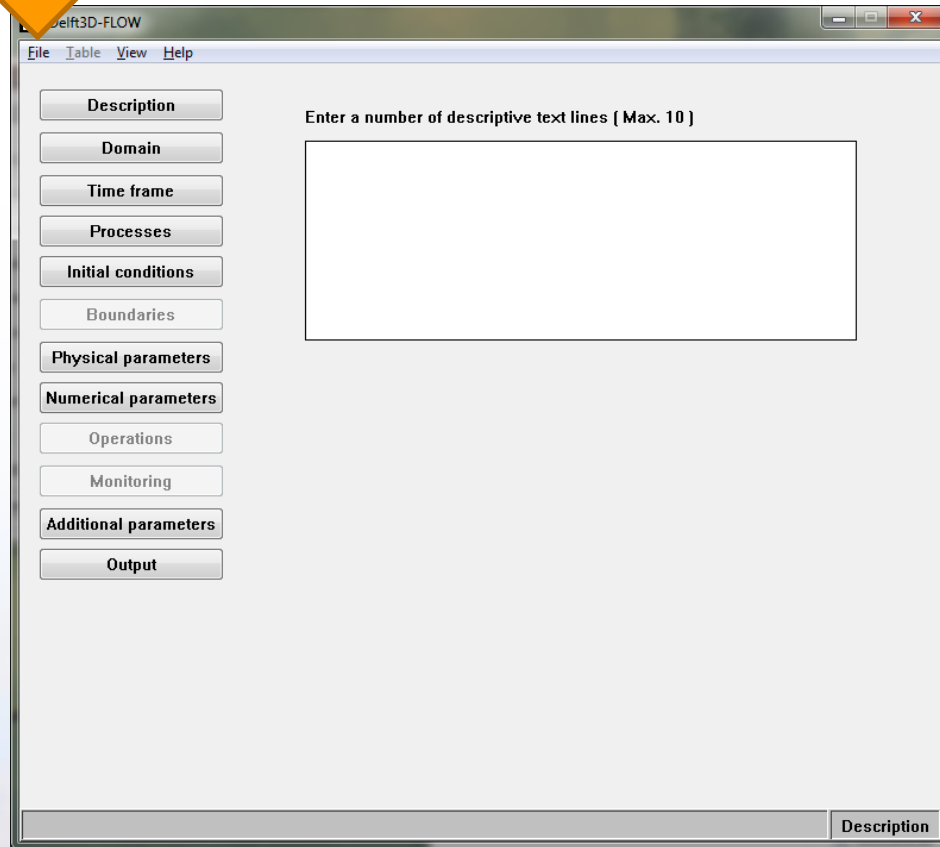
- Choose the folder currently being worked in.

Flow



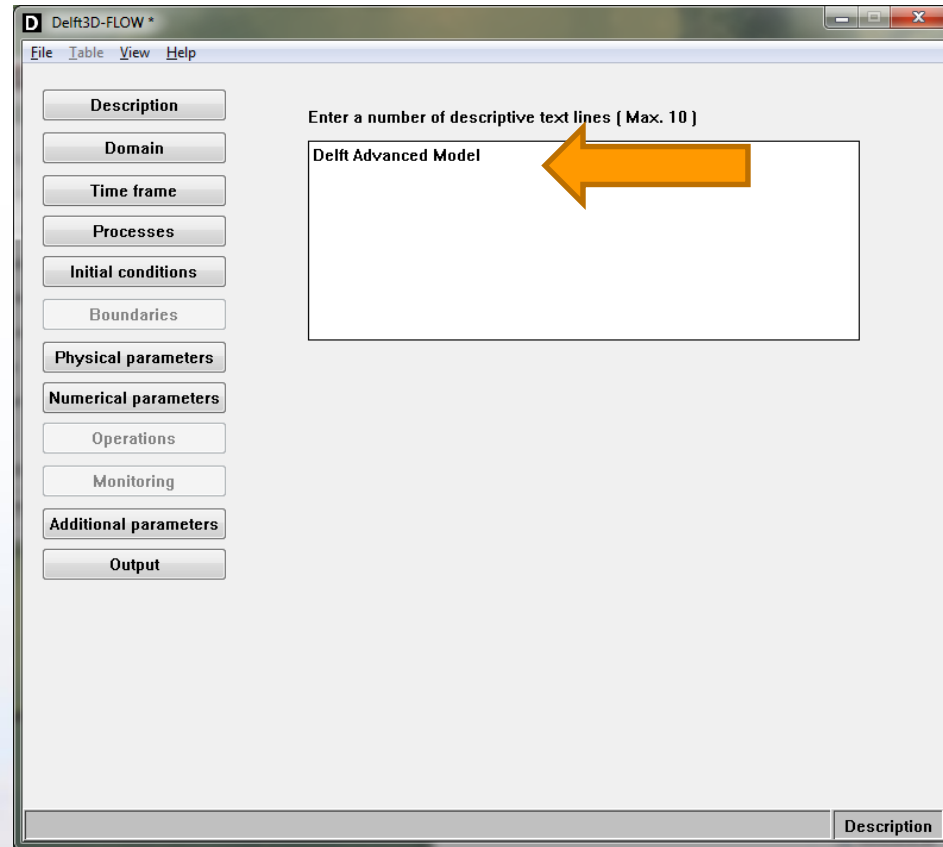
- Return to the main menu and select Flow Input

Delft3D-FLOW



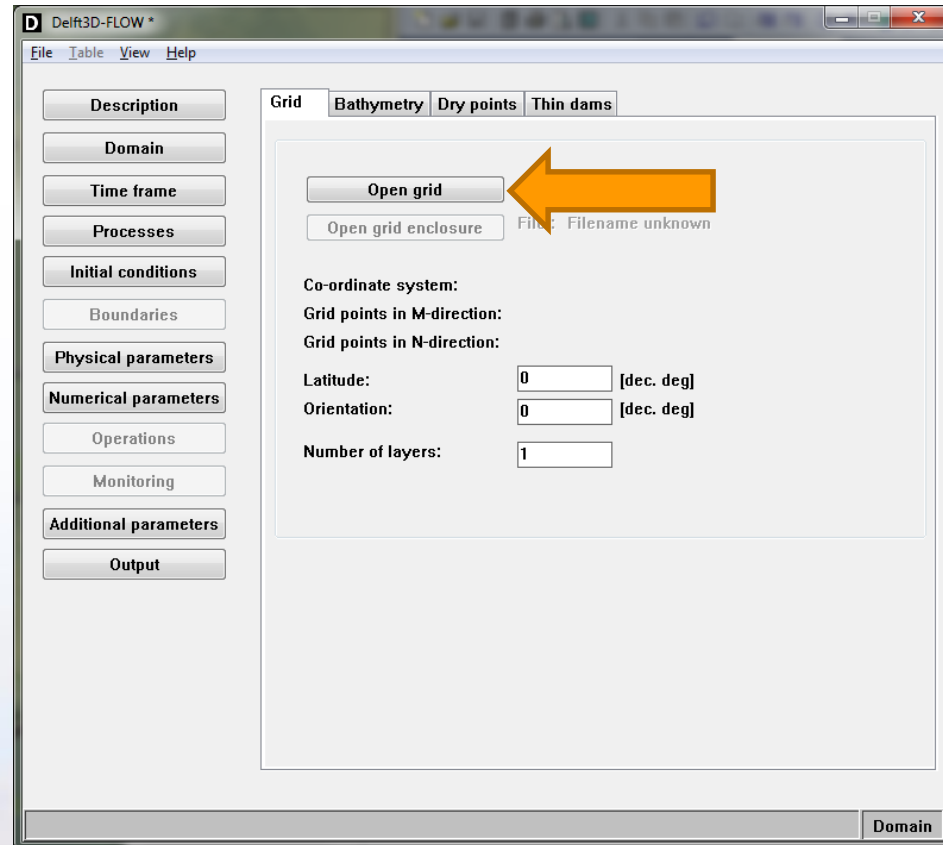
■ Select File → New

Delft3D-FLOW



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

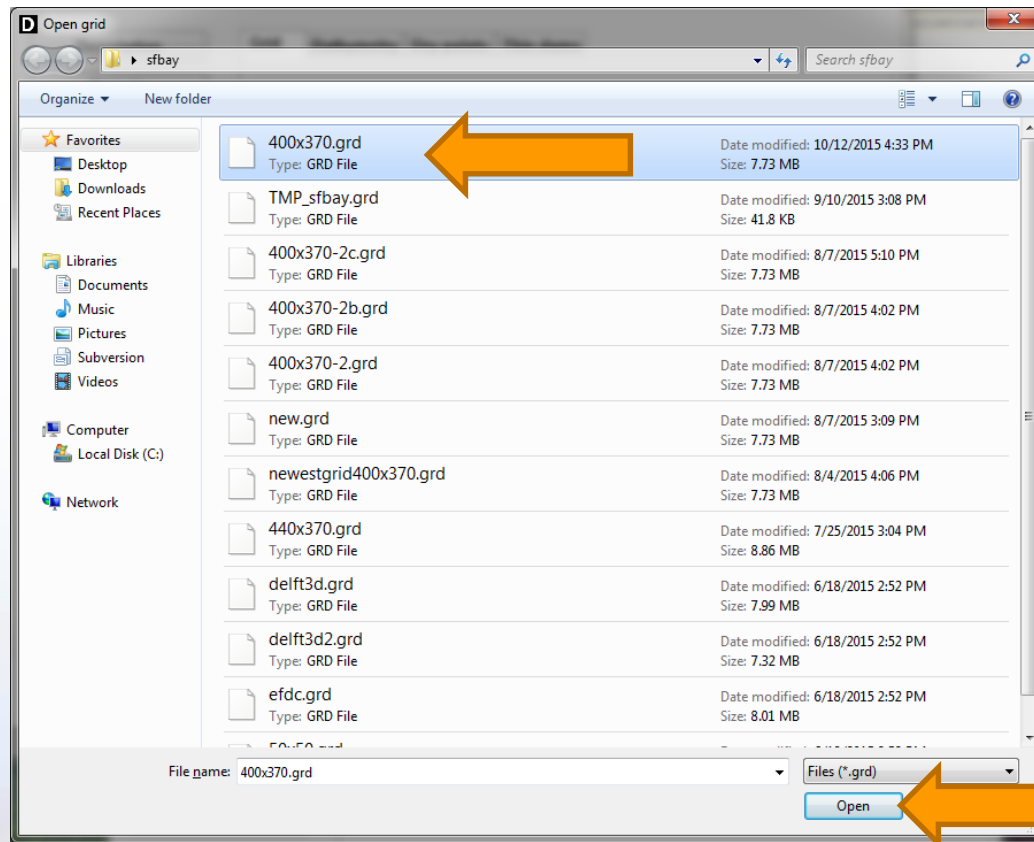
Delft3D-FLOW



- Under Domain → Grid, select Open Grid

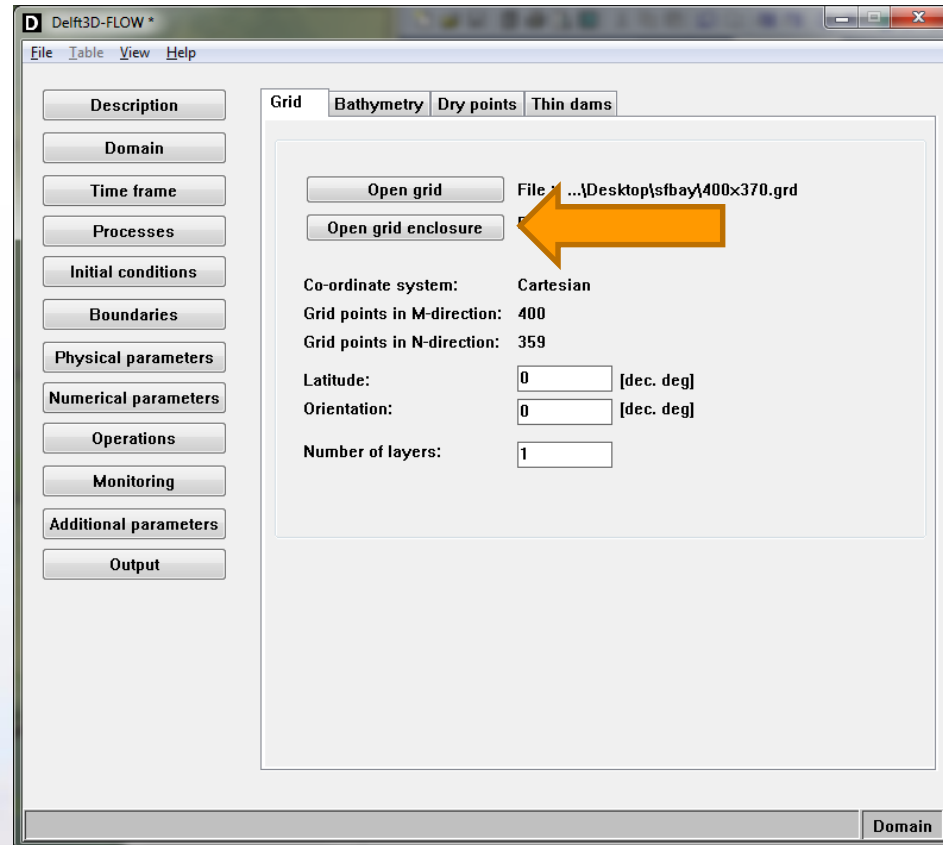
Delft3D-FLOW

■ Select 400x370.grd



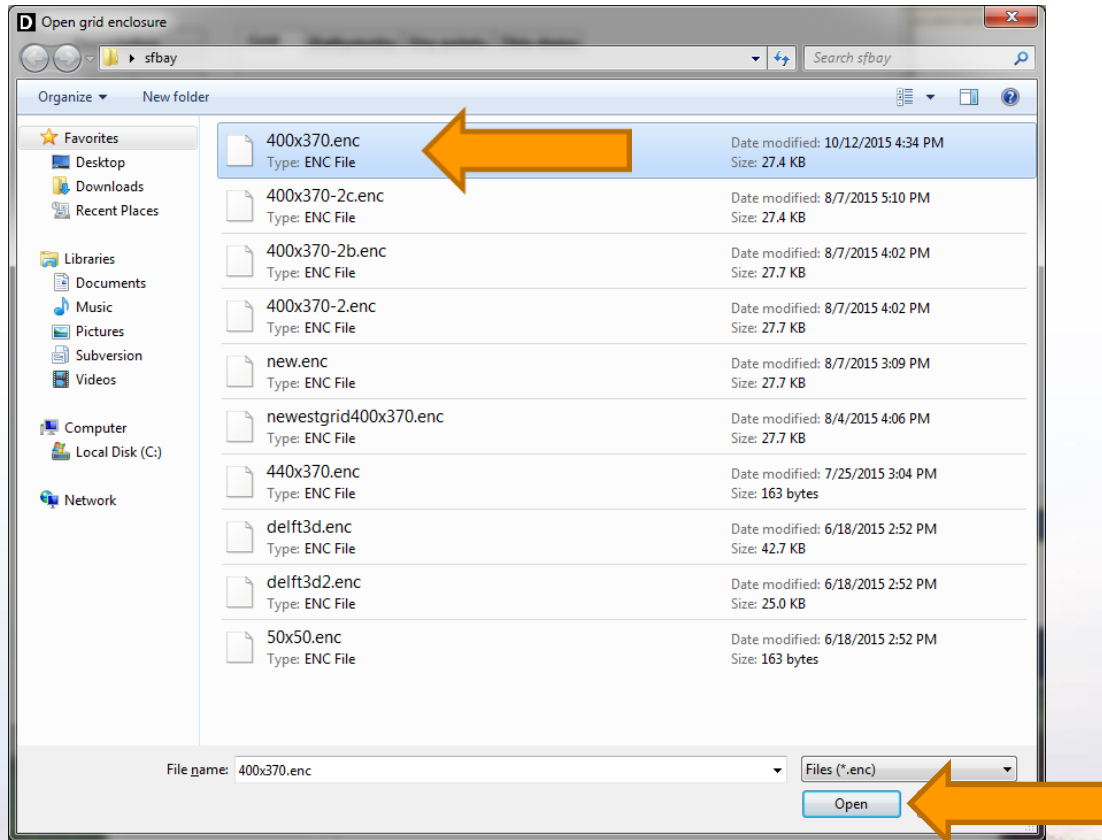
Delft3D-FLOW

- Select Open grid enclosure

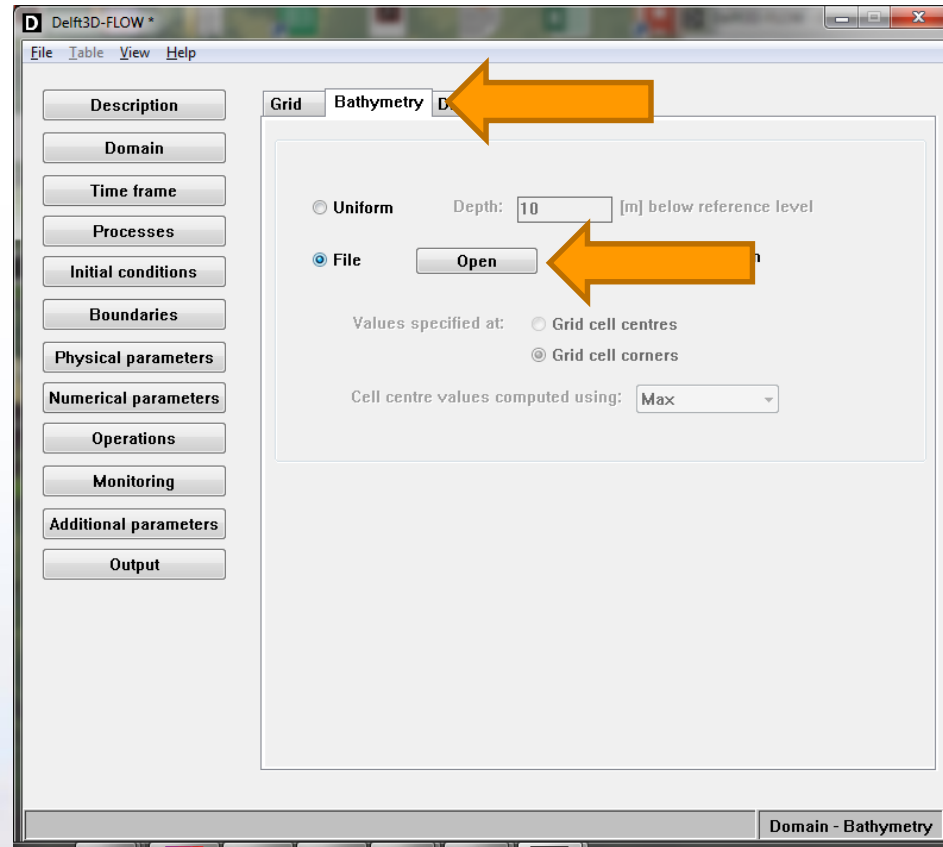


Delft3D-FLOW

■ Select 400X370.enc



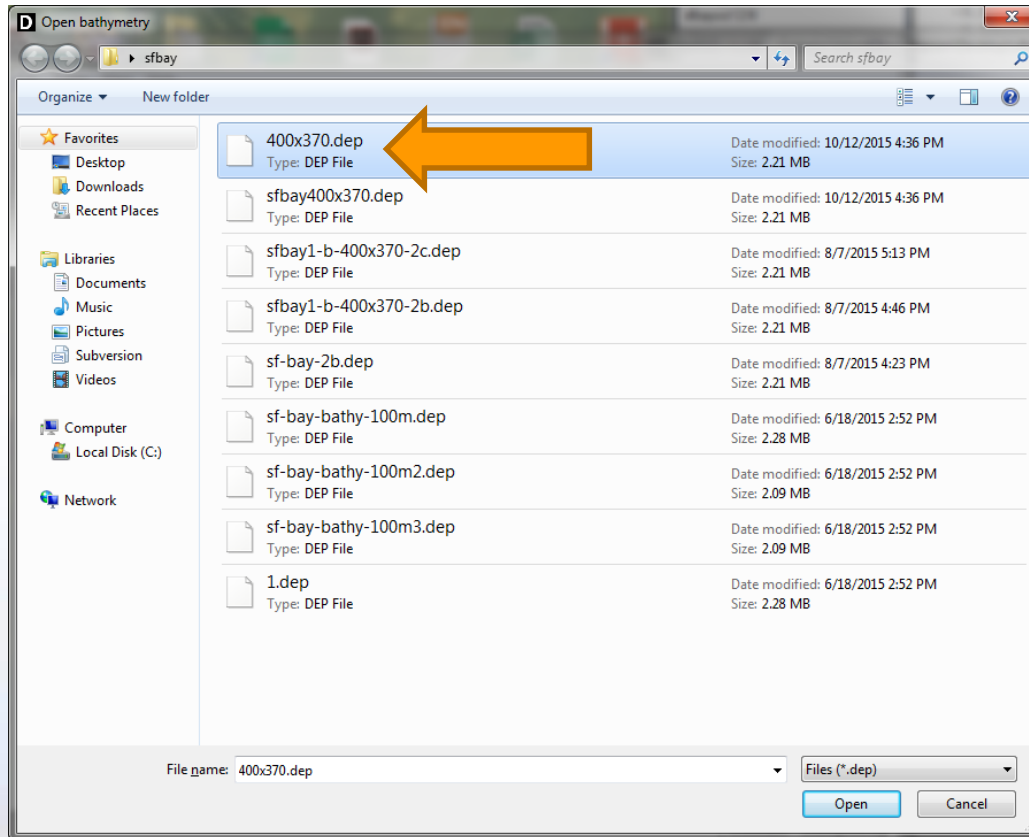
Delft3D-FLOW



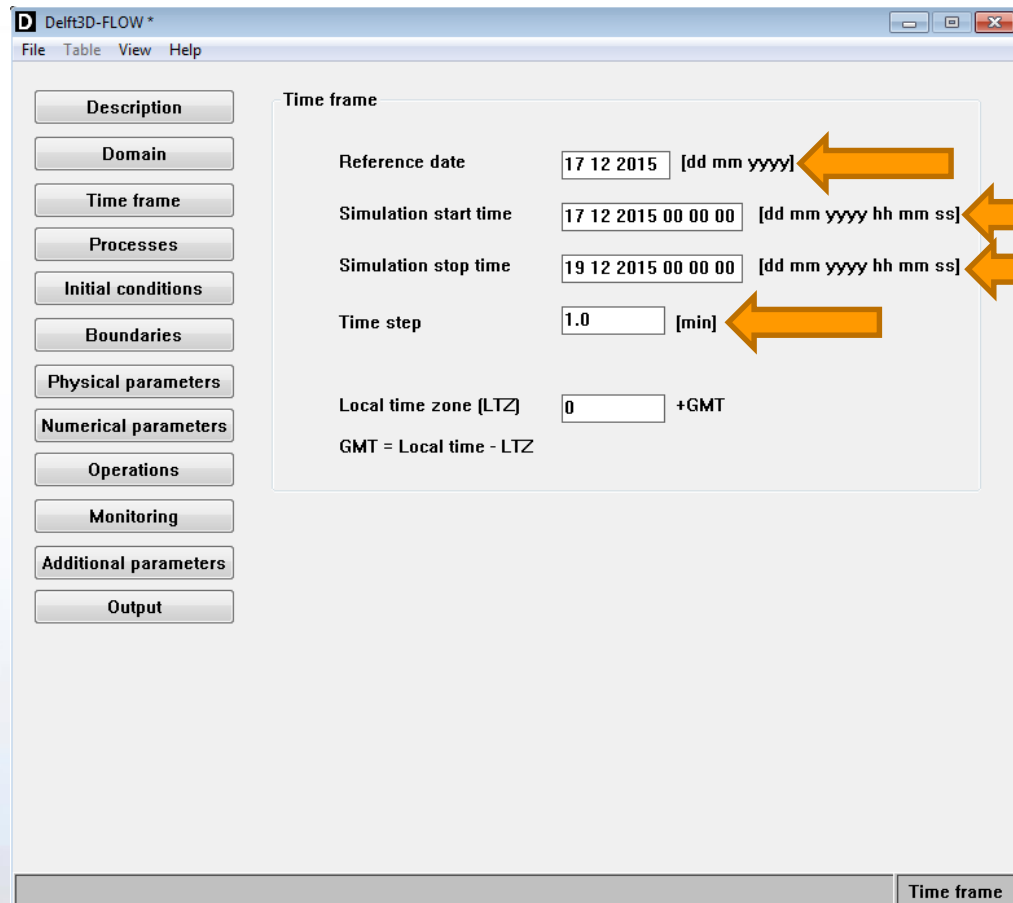
- Select Bathymetry and then Select Open

Delft3D-FLOW

- Select the 400x370.dep depth file



Delft3D-FLOW



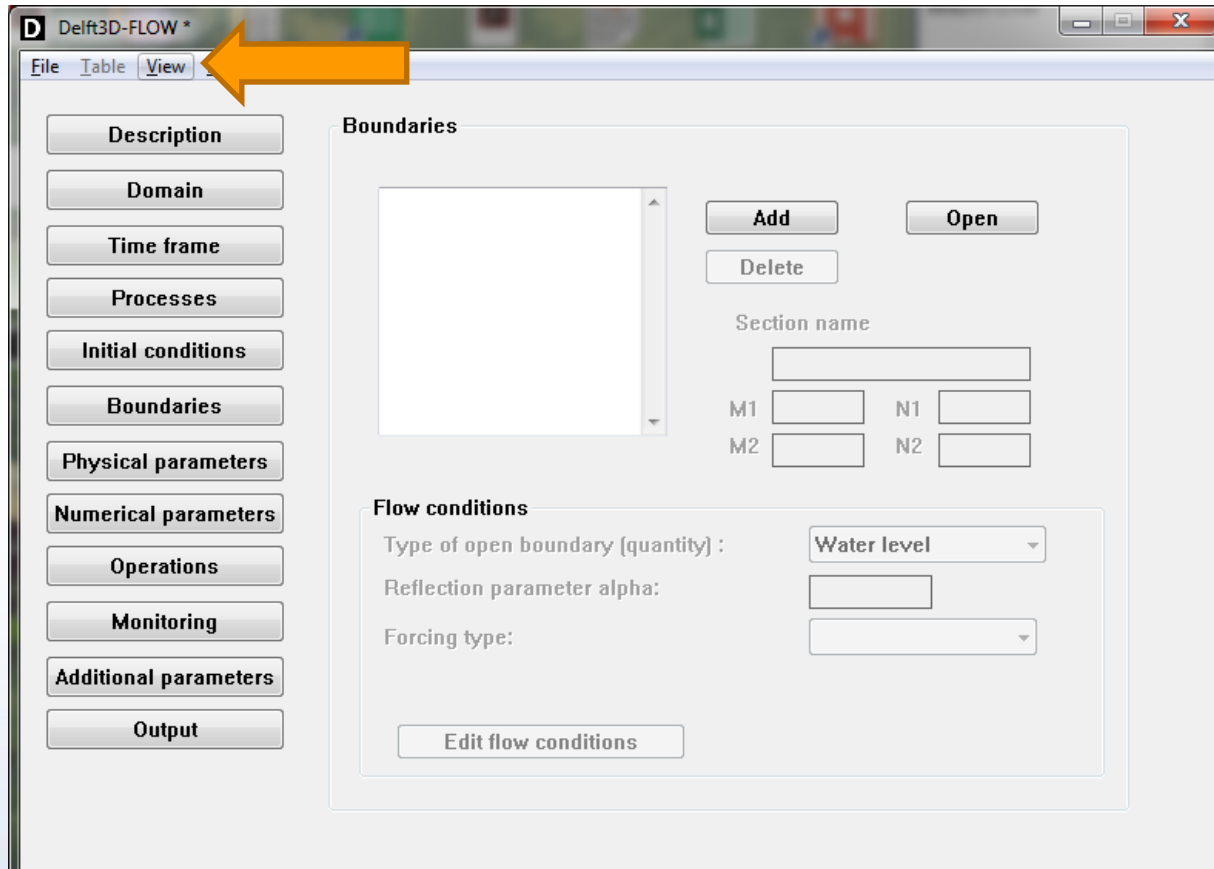
The screenshot shows the Delft3D-FLOW software window with the 'Time frame' tab selected. The left sidebar contains buttons for Description, Domain, Time frame, Processes, Initial conditions, Boundaries, Physical parameters, Numerical parameters, Operations, Monitoring, Additional parameters, and Output. The main area displays the following settings:

Parameter	Value	Format
Reference date	17 12 2015	[dd mm yyyy]
Simulation start time	17 12 2015 00 00 00	[dd mm yyyy hh mm ss]
Simulation stop time	19 12 2015 00 00 00	[dd mm yyyy hh mm ss]
Time step	1.0	[min]
Local time zone (LTZ)	0	+GMT

Below the table, it states: GMT = Local time - LTZ.

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

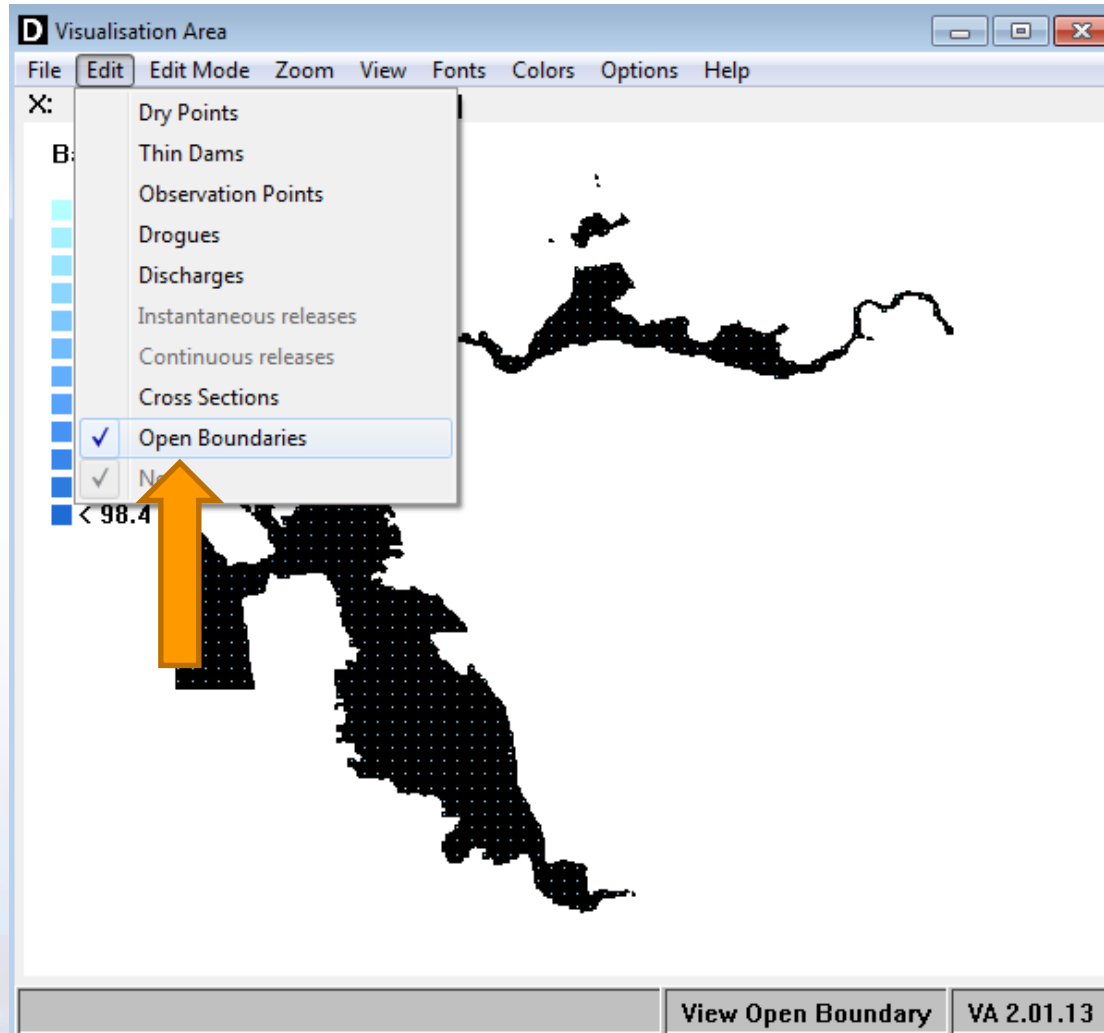
Delft3D-FLOW



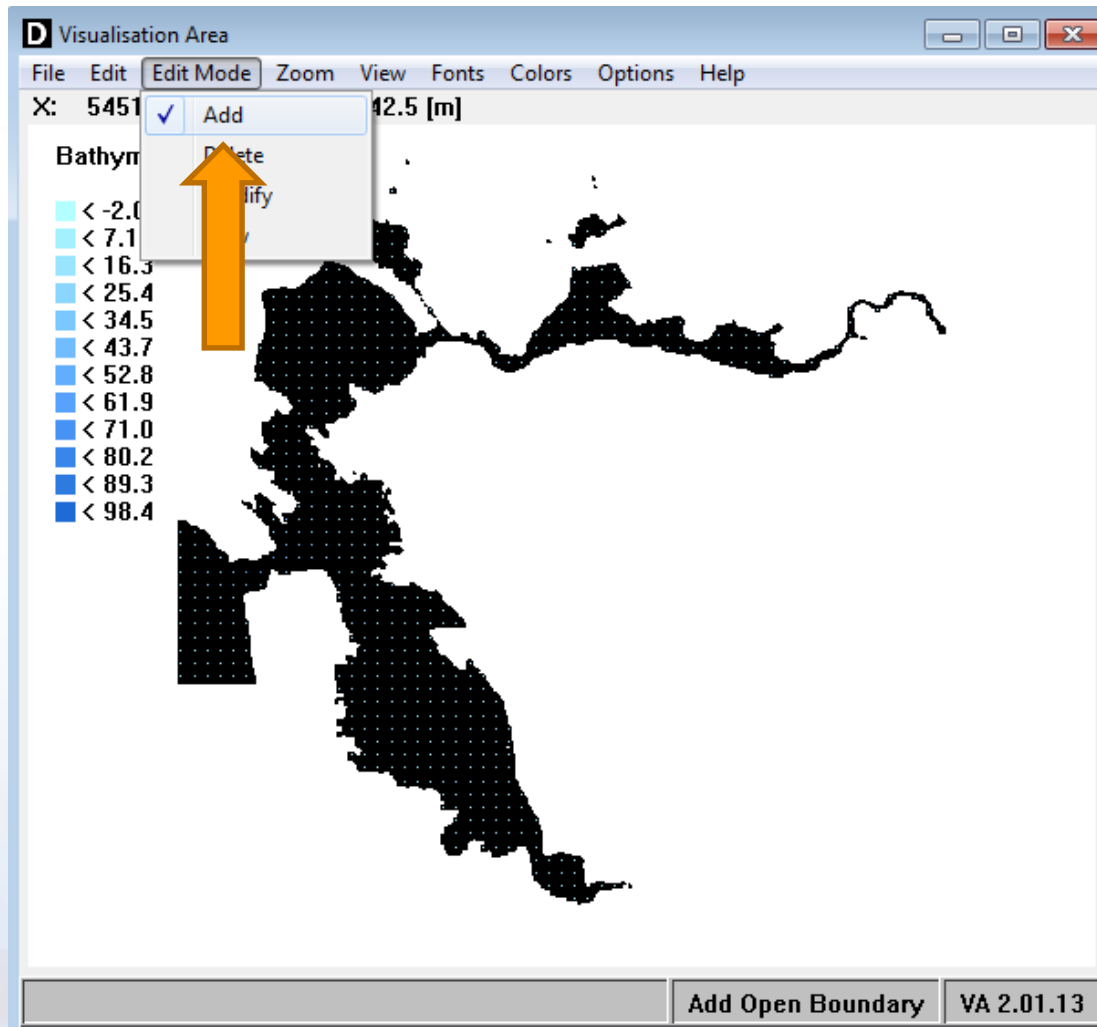
- Select the Boundaries Tab
- Then, select View → Visualization Area

Visualization Area

- Select Edit → Open Boundaries

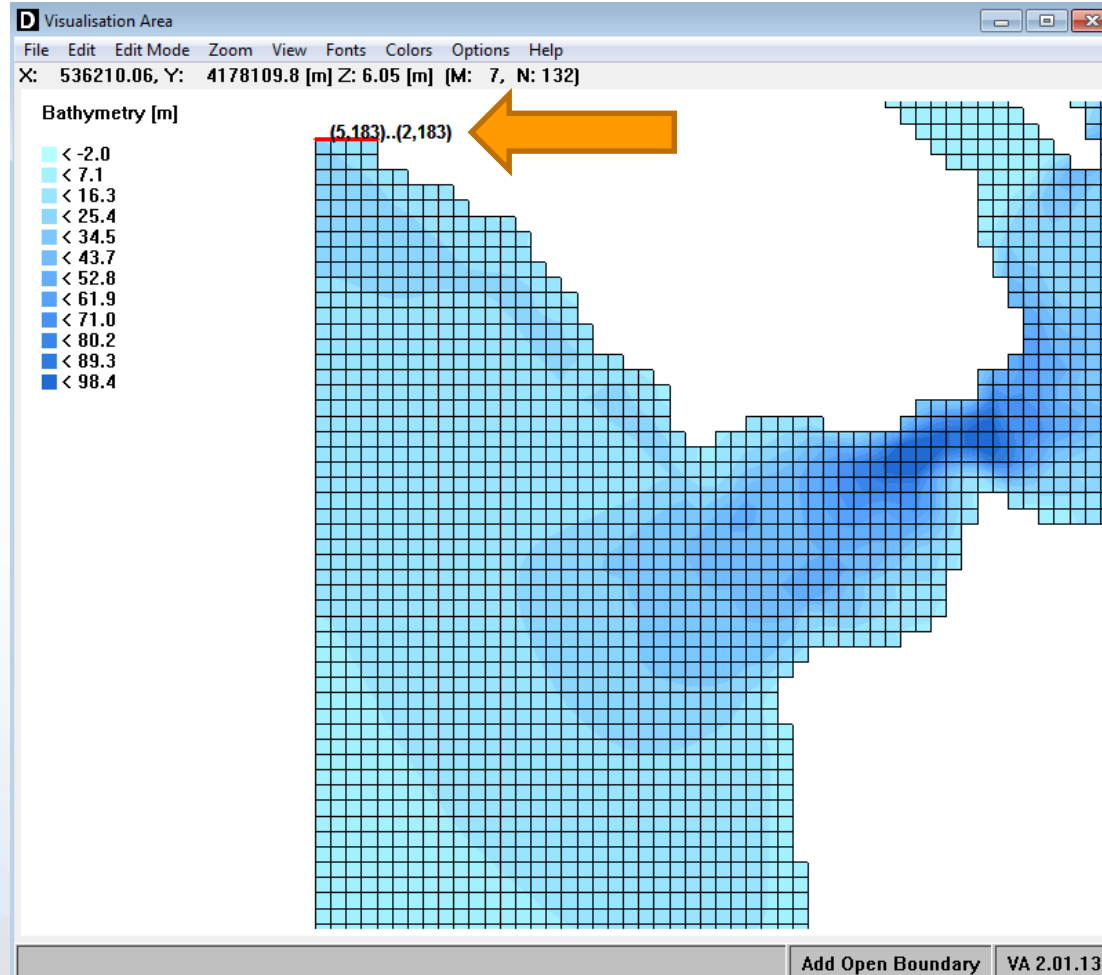


Visualization Area



- Select Edit Mode → Add

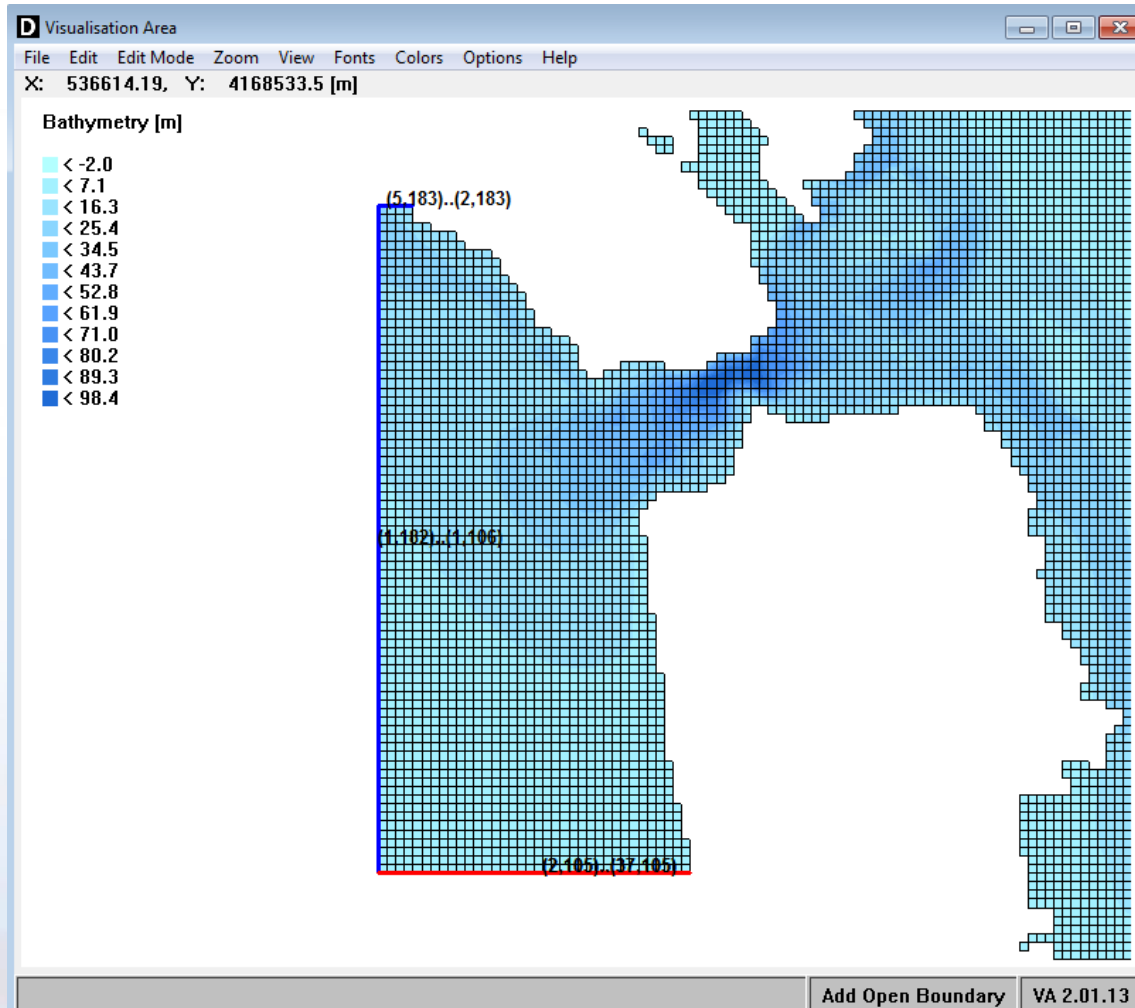
Visualization Area



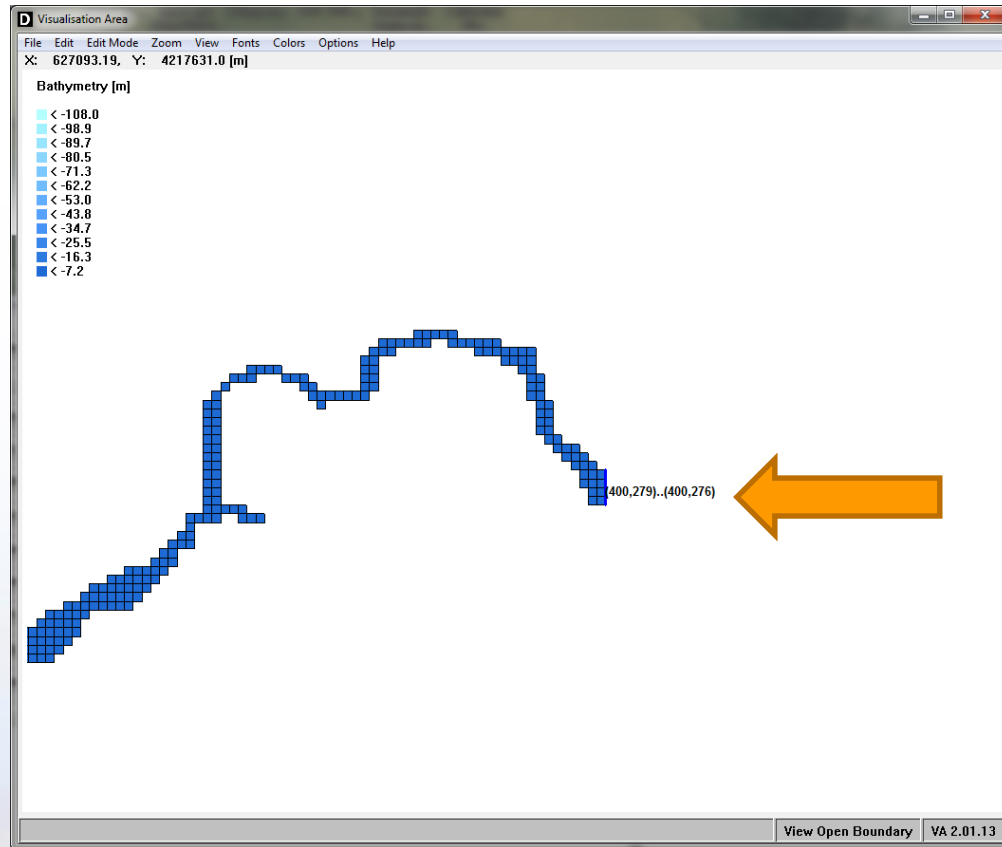
- Using the mouse, drag and add boundaries along the edge of the grid.

Visualization Area

- The left side should look like this.



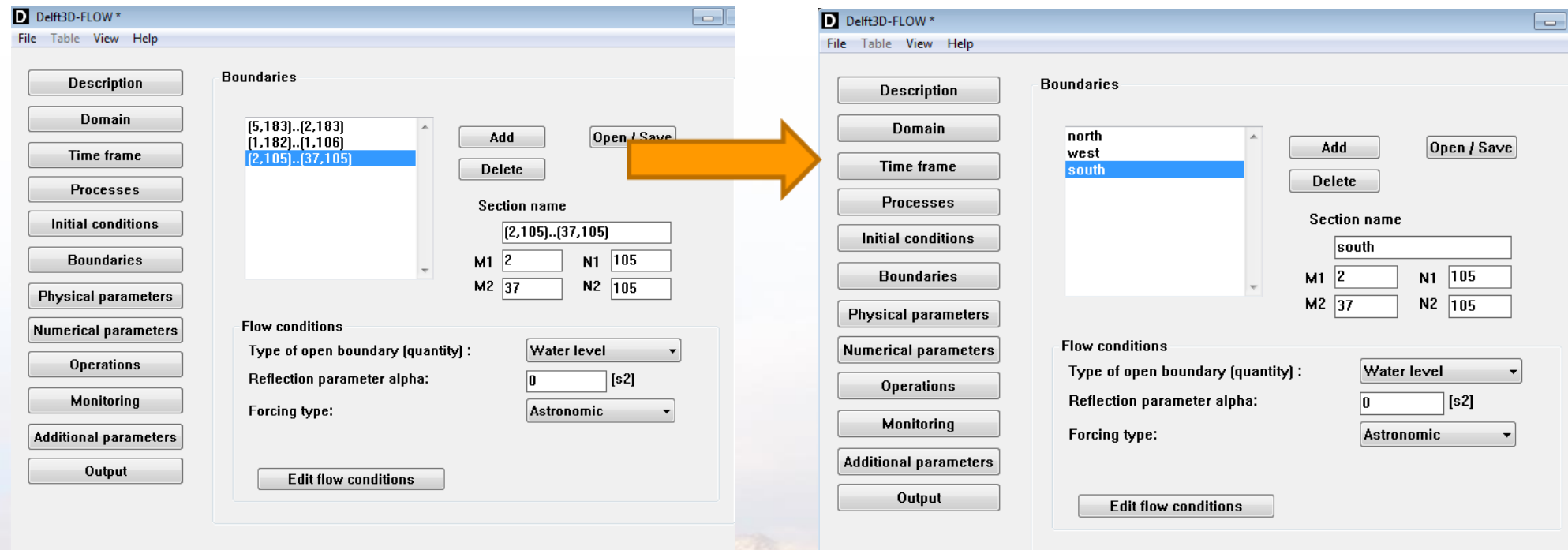
Visualization Area



- 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

Delft3D-FLOW *

File Table View Help

Boundaries

north
west
south

Add Open / Save
Delete

Section name
south

M1 2 N1 105
M2 37 N2 105

Flow conditions

Type of open boundary (quantity) : Water level

Reflection parameter alpha: 0 [s2]

Forcing type: Time-series

Edit flow conditions

Boundaries

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

Editing Boundary Flow Conditions

Delft3D-FLOW *

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Boundaries

north
west
south

Add

Delete

Open / Save

Section name

north

M1 5 N1 183

M2 2 N2 183

Flow conditions

Type of open boundary (quantity): Water level

Reflection parameter alpha: 0 [s2]

Forcing type: Time-series

Edit flow conditions

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

Editing Boundary Flow Conditions

D Boundaries : Flow Conditions

Table

Boundary: north
Quantity: Water level
Forcing type: Time-series

Time dd mm yyyy hh mm ss	Begin [m]	End [m]
17 12 2015 00 00 00	-0.5	-0.5
17 12 2015 08 00 00	-1.5	-1.5
17 12 2015 16 00 00	1	1
18 12 2015 08 00 00	-0.5	-0.5
18 12 2015 16 00 00	1	1
19 12 2015 00 00 00	0	0

Close

- 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

Delft3D-FLOW *

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Boundaries

north
west
south

Add

Delete

Open / Save

Section name

north

M1 5 N1 183

M2 2 N2 183

Flow conditions

Type of open boundary (quantity) : Water level

Reflection parameter alpha: 0 [s2]

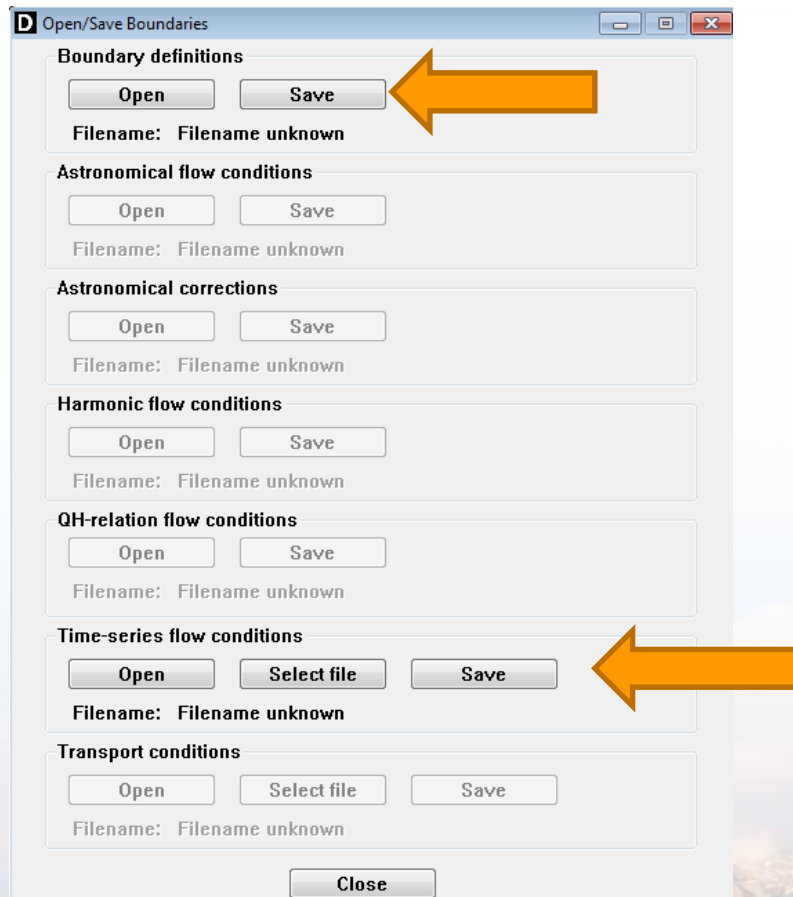
Forcing type: Time-series

Edit flow conditions

Boundaries

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

Editing Boundary Flow Conditions



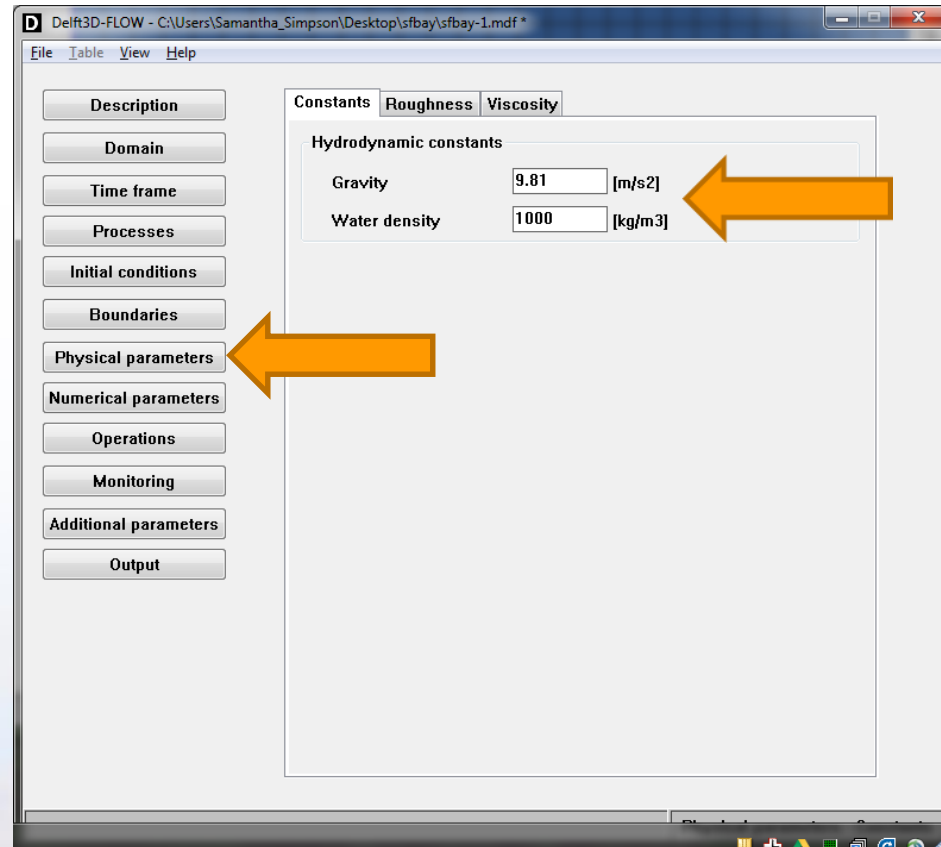
The screenshot shows a software dialog box titled "Open/Save Boundaries". It contains several sections, each with "Open" and "Save" buttons and a "Filename: Filename unknown" label. The sections are:

- Boundary definitions**: The "Save" button is highlighted with an orange arrow.
- Astronomical flow conditions**: Contains "Open" and "Save" buttons.
- Astronomical corrections**: Contains "Open" and "Save" buttons.
- Harmonic flow conditions**: Contains "Open" and "Save" buttons.
- QH-relation flow conditions**: Contains "Open" and "Save" buttons.
- Time-series flow conditions**: Contains "Open", "Select file", and "Save" buttons. The "Save" button is highlighted with an orange arrow.
- Transport conditions**: Contains "Open", "Select file", and "Save" buttons.

A "Close" button is located at the bottom center of the dialog.

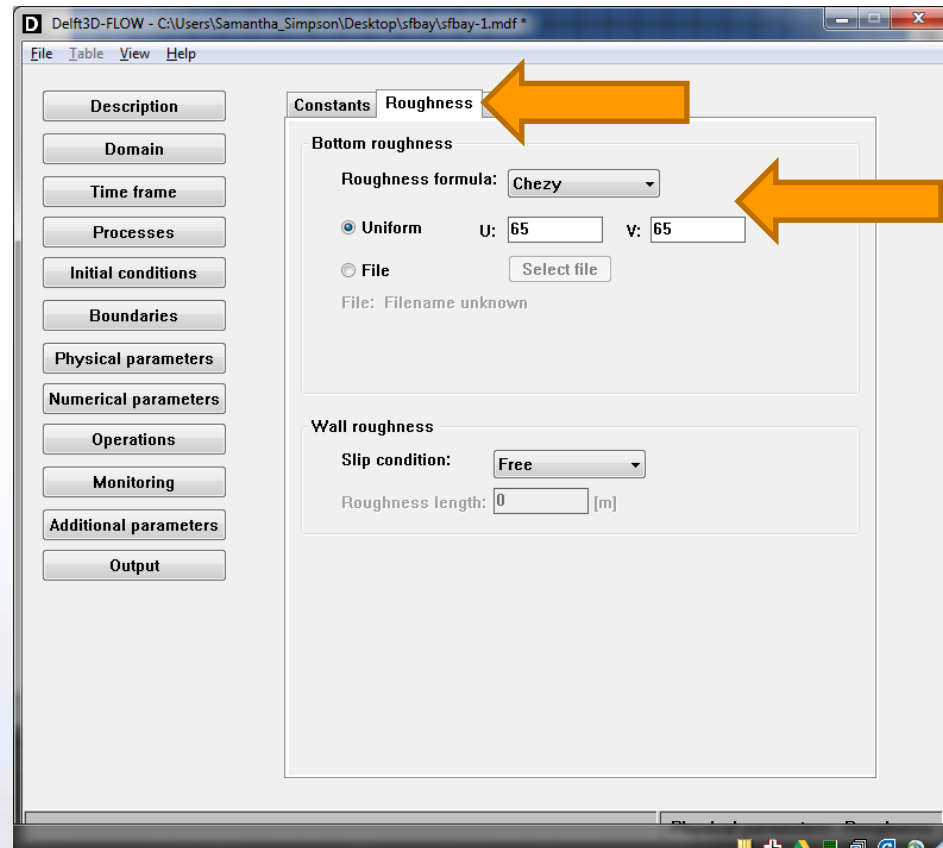
- Save Boundary definition as **bnds.bnd**
- Save Time-series flow conditions as **boundaries.bct**
- You have now created a .bnd and a .bct file.

Physical Parameters



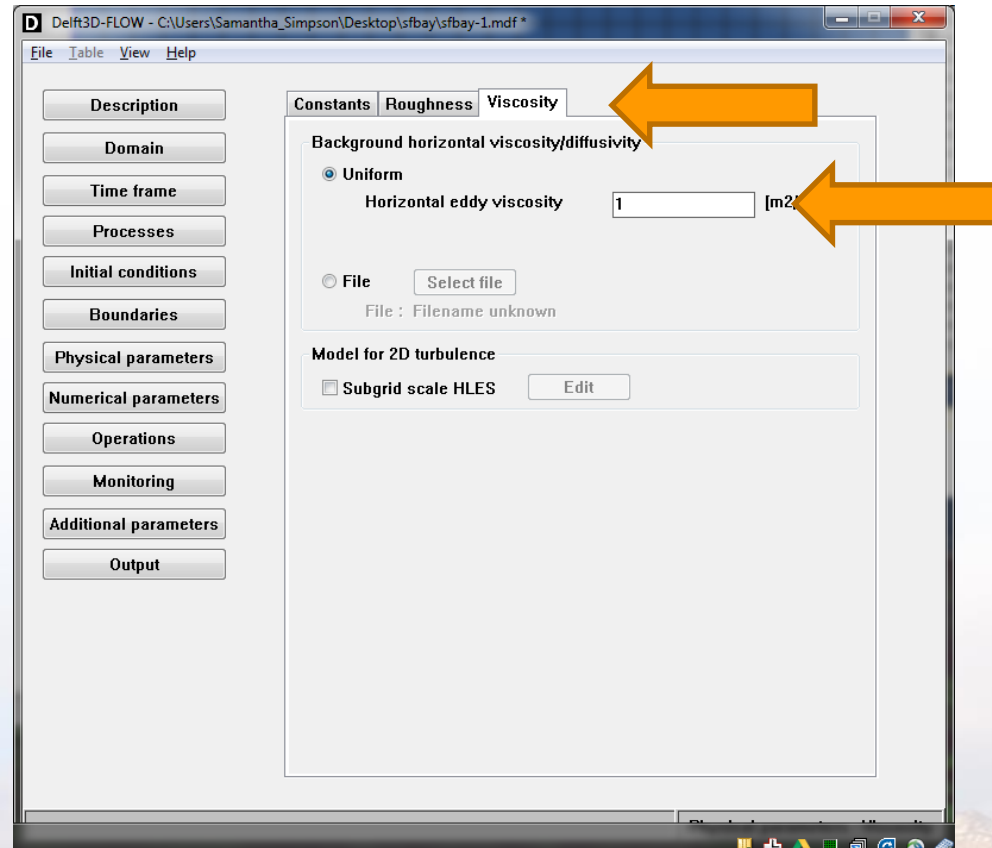
- Then under Physical parameters → Constants , add the gravity and water density

Physical Parameters



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

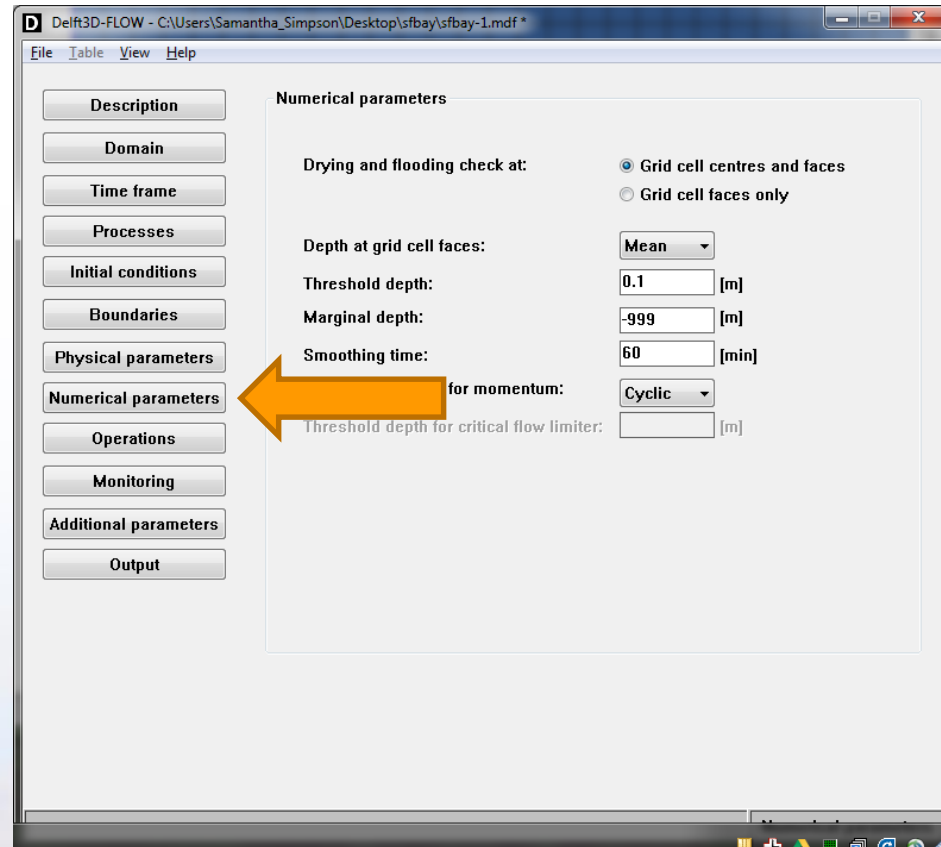
Physical Parameters



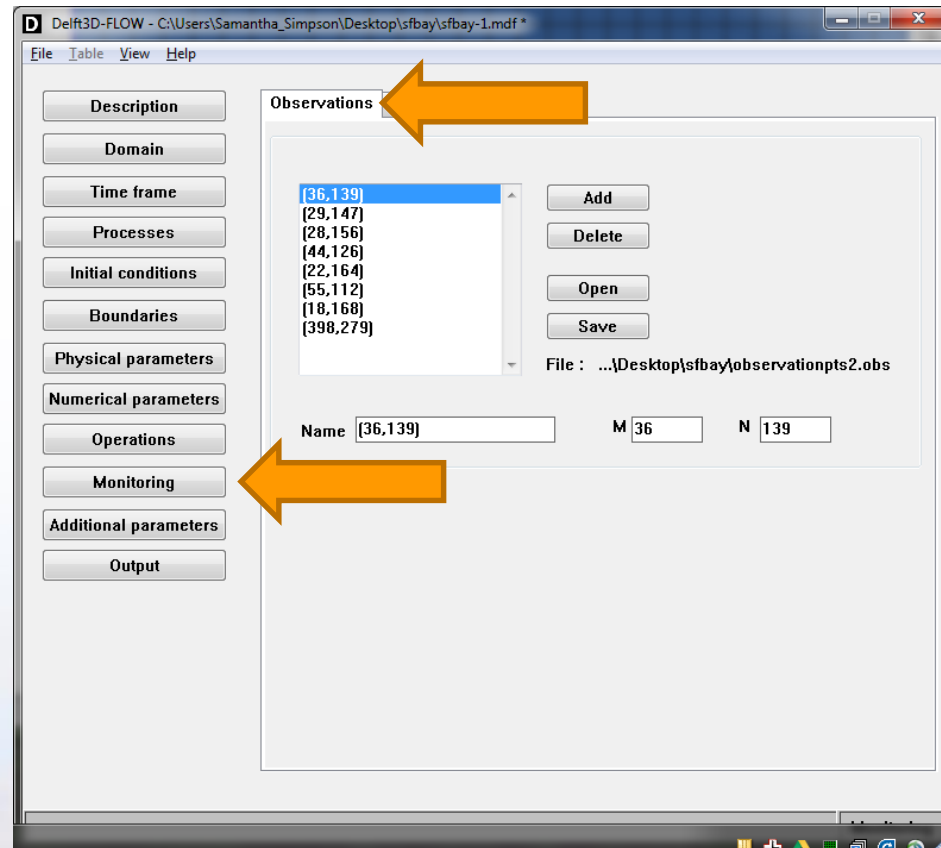
- Under Viscosity, select Uniform Horizontal Eddy Viscosity

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:

The screenshot shows the Delft3D-FLOW software window. On the left is a vertical sidebar with buttons for: Description, Domain, Time frame, Processes, Initial conditions, Boundaries, Physical parameters, Numerical parameters, Operations, Monitoring, Additional parameters, and Output. The 'Output' button is highlighted with an orange arrow. The main window has tabs for 'Storage', 'Print', and 'Details'. The 'Storage' tab is active and contains the following settings:

FLOW simulation times
Start time: 17 12 2015 00 00 00
Stop time: 19 12 2015 00 00 00
Time Step [min]: 1.0

Store map results
Start time: 17 12 2015 00 00 00
Stop time: 19 12 2015 00 00 00
Interval: 10.0 [min]

Store communication file :
Start time: 17 12 2015 00 00 00
Stop time: 19 12 2015 00 00 00
Interval: 0.0 [min]

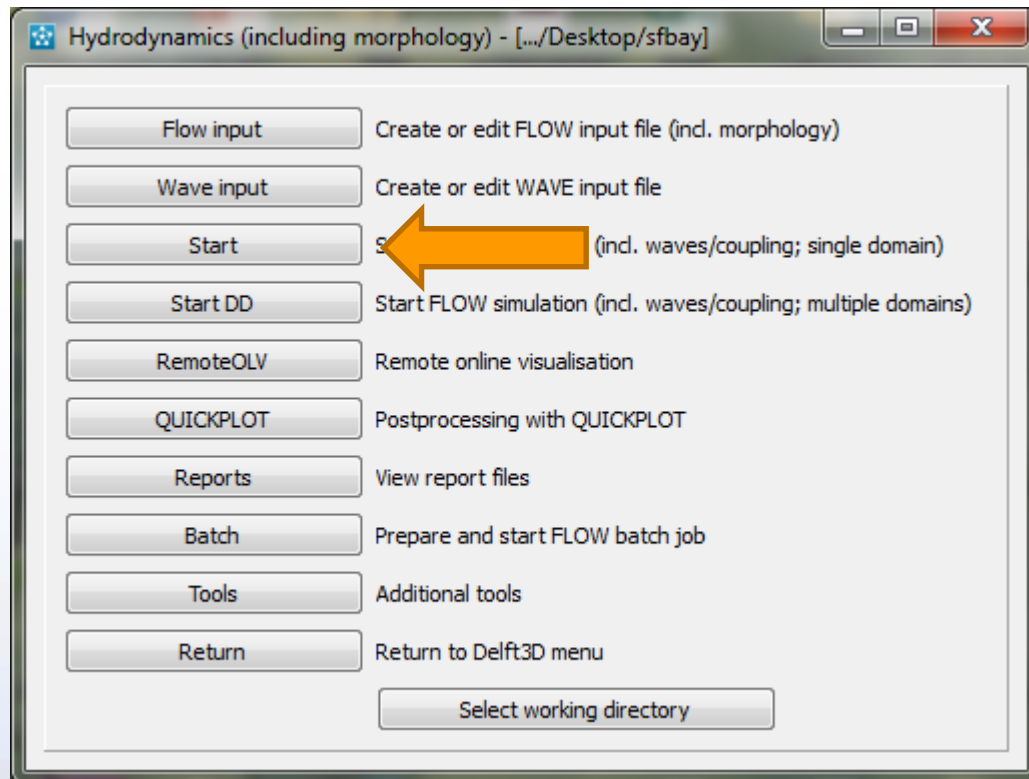
History interval 30 [min] **Restart int.** 144.0 [min]

☐ Fourier analysis ☐ Online visualisation
☐ Export WAQ input

File : Filename unknown [Edit WAQ input >>](#)

At the bottom right of the window, there is an 'Output' button.

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