SAND2013-6187P

SNL Water Power Program

Building an SNL-EFDC Model

Jesse Roberts[1], Scott James[2], and Craig Jones[3]

[1]Sandia National Laboratories

[2]E[x]ponent Inc.

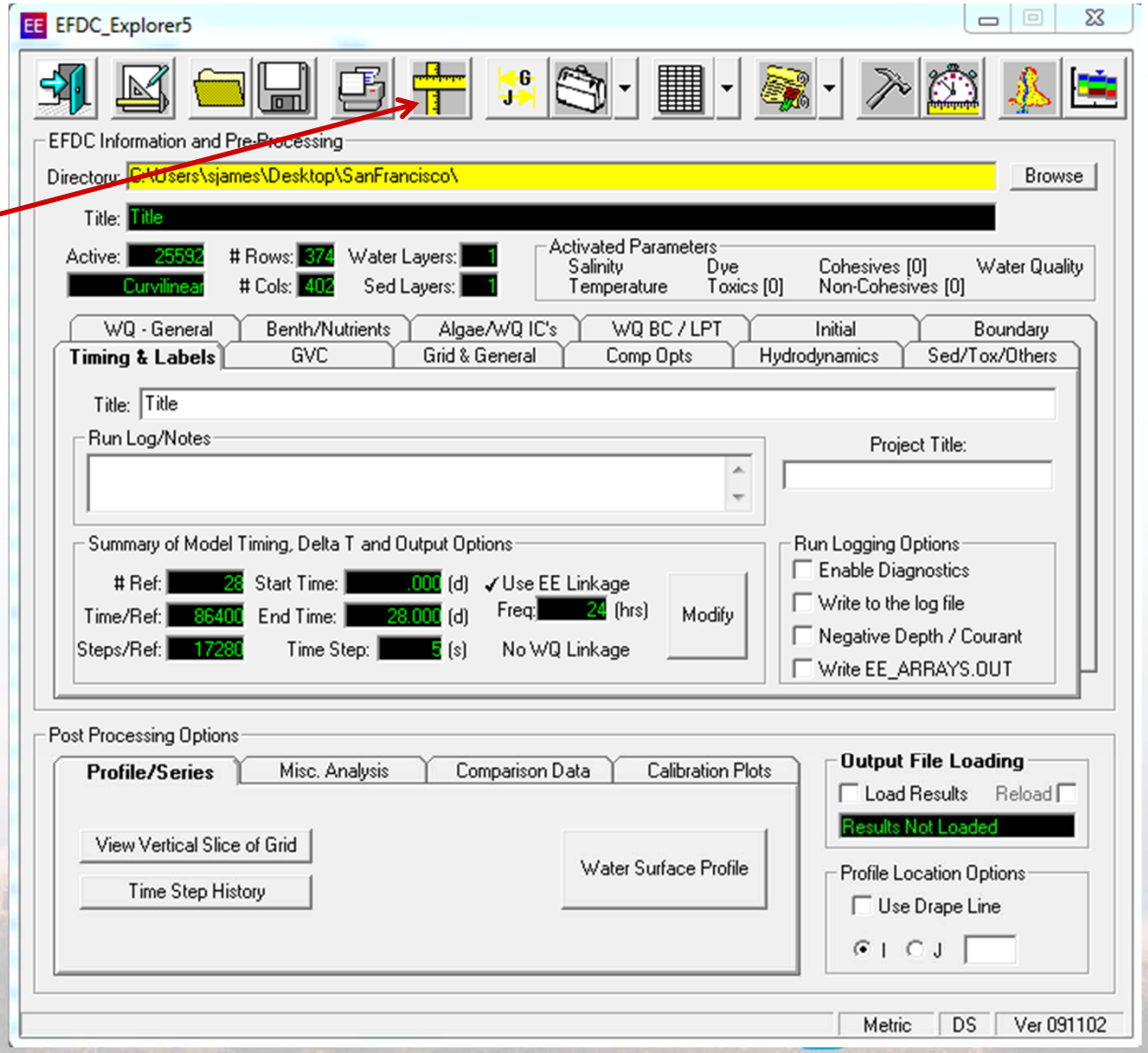
[3]Sea Engineering Inc.

Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

Sandia National Laboratories

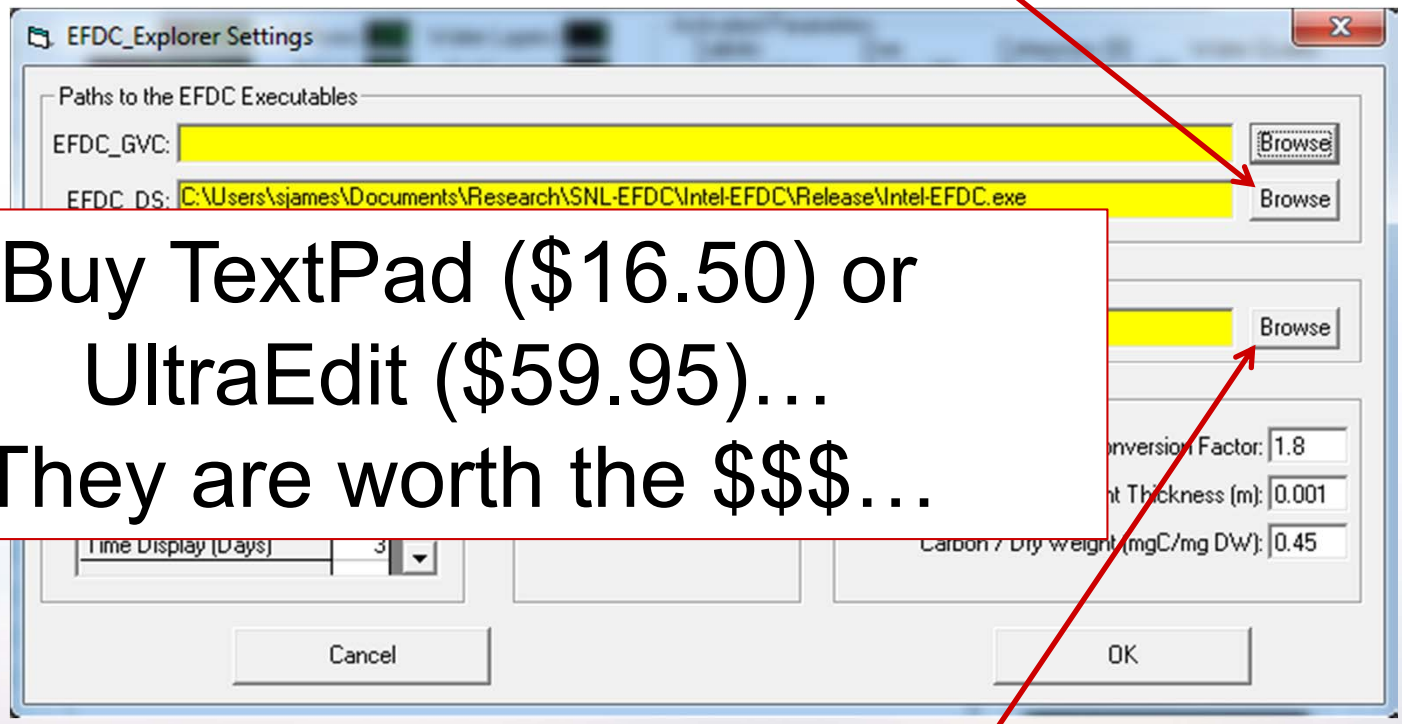
Set Things Up

Click “Settings”



Select the Executable

Select the proper executable (e.g., Intel-EFDC.exe)

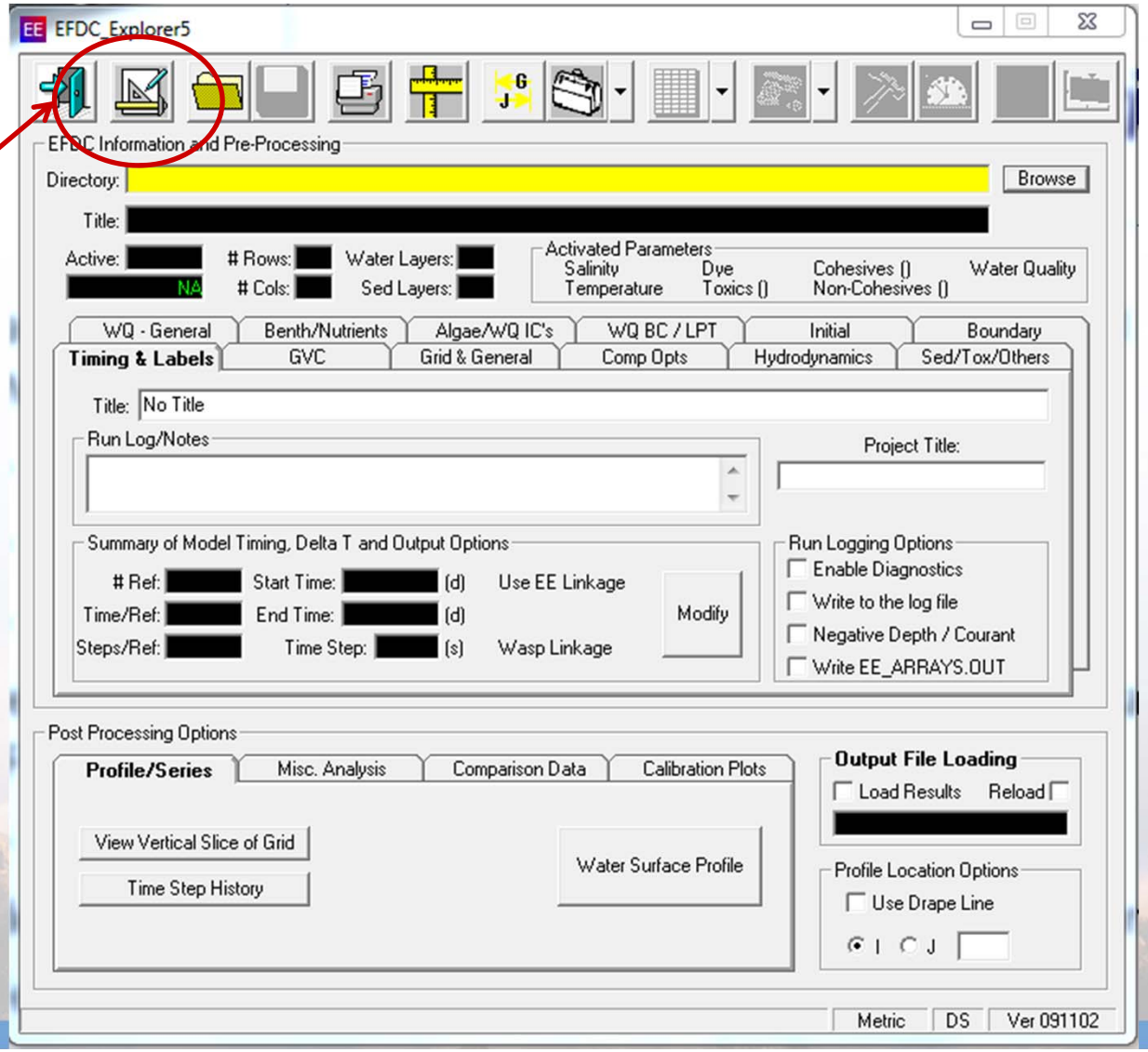


Buy TextPad ($16.50) or UltraEdit ($59.95)…
They are worth the $$$…

Select a “Text Editor” – like Notepad.exe

Building a Grid (Using EE)

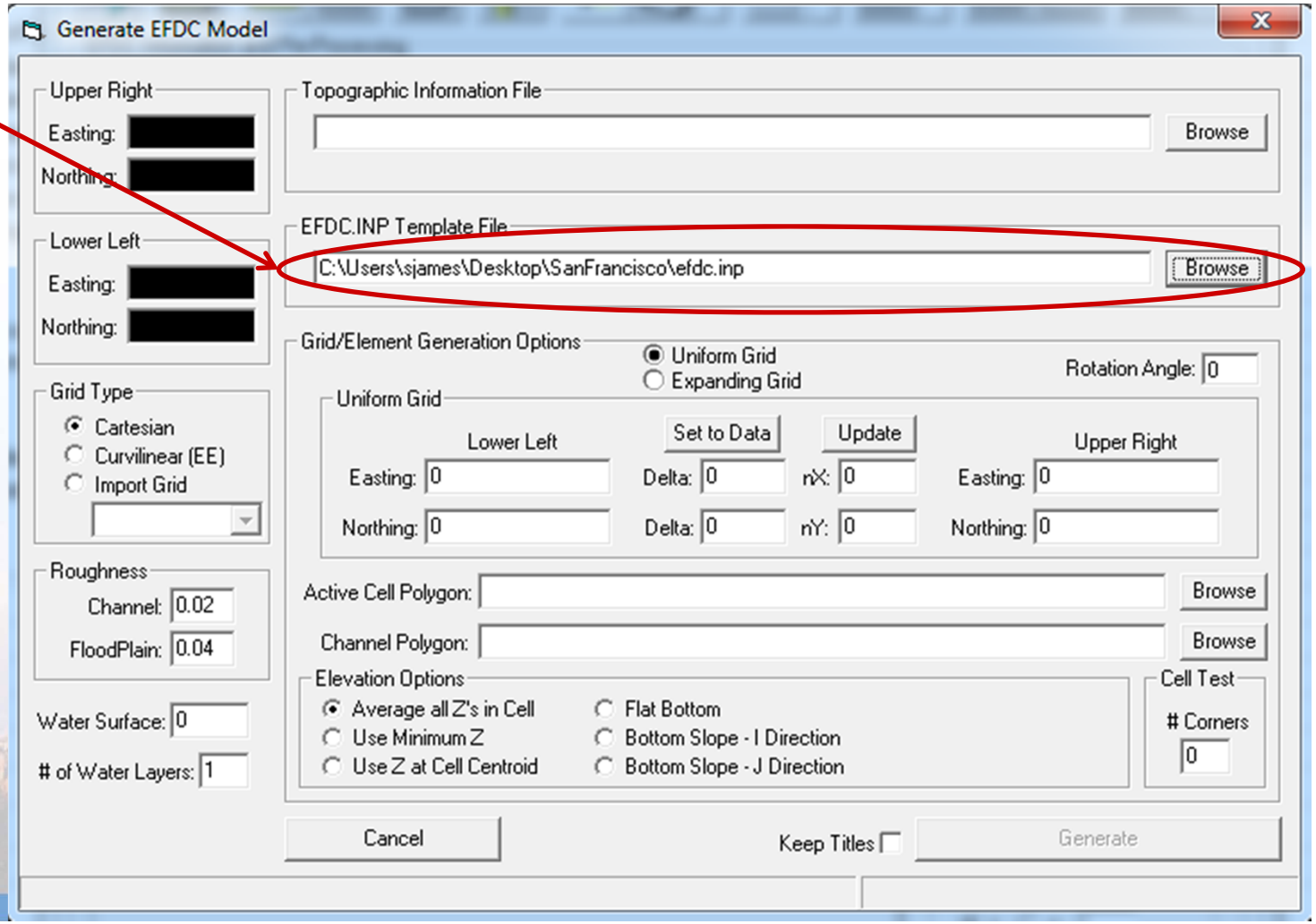
Click
"Generate a New Model"



Building a Grid (Using EE)

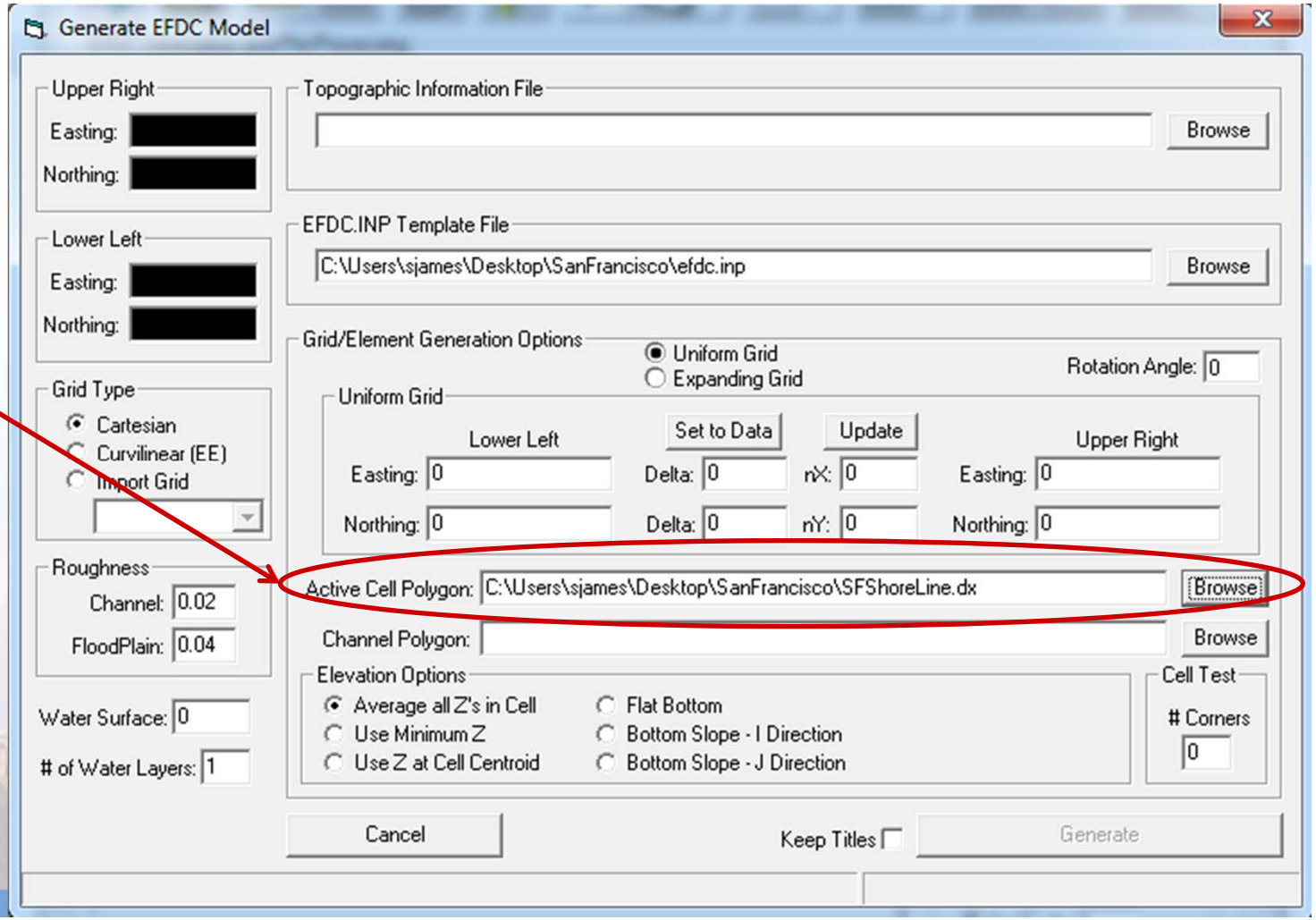
Select an appropriate efdc.inp template file

Look for the "SanFrancisco" folder in the "Models" directory



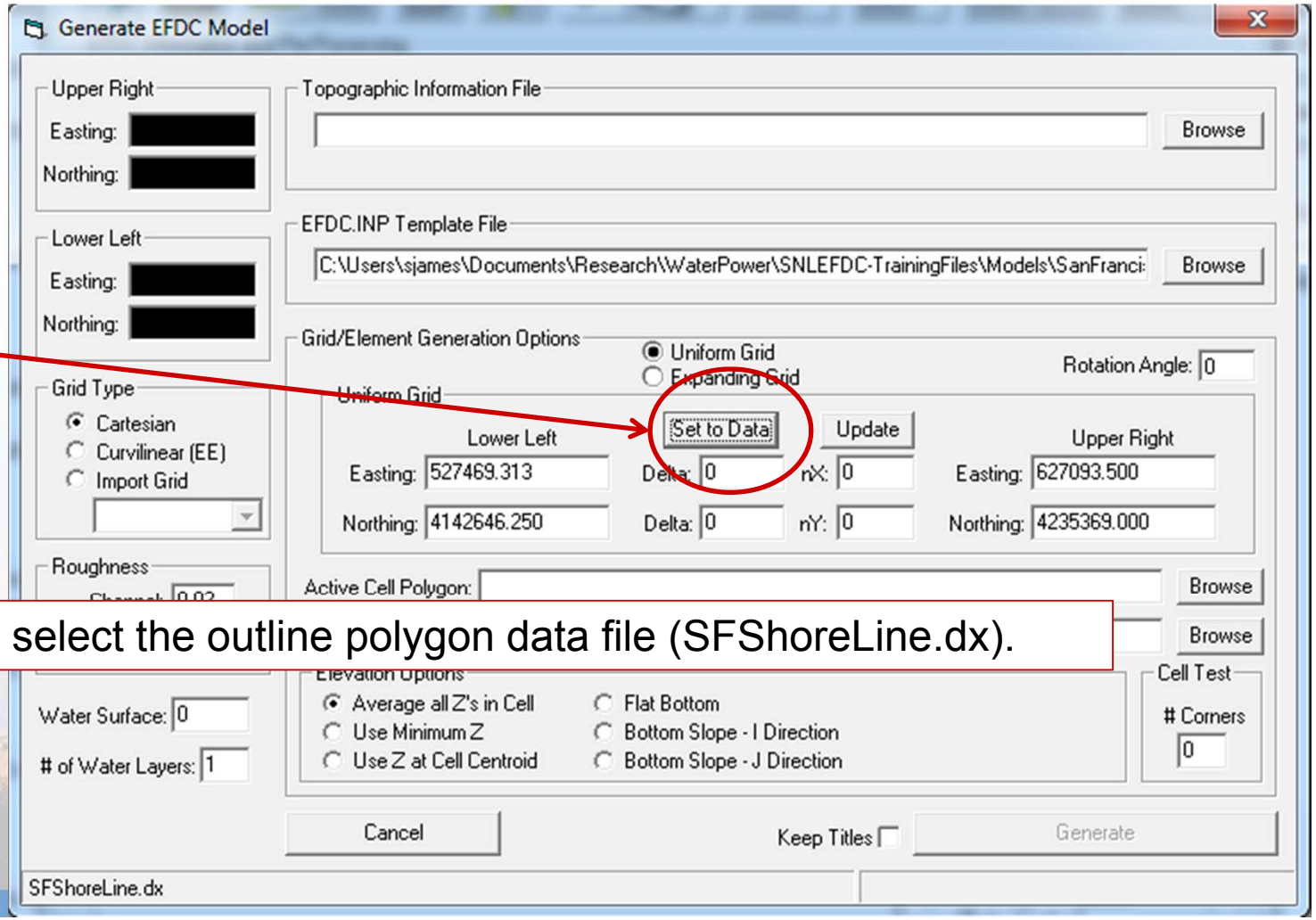
Building a Grid (Using EE)

Select the shoreline data file



Building a Grid (Using EE)

Click
"Set to Data"



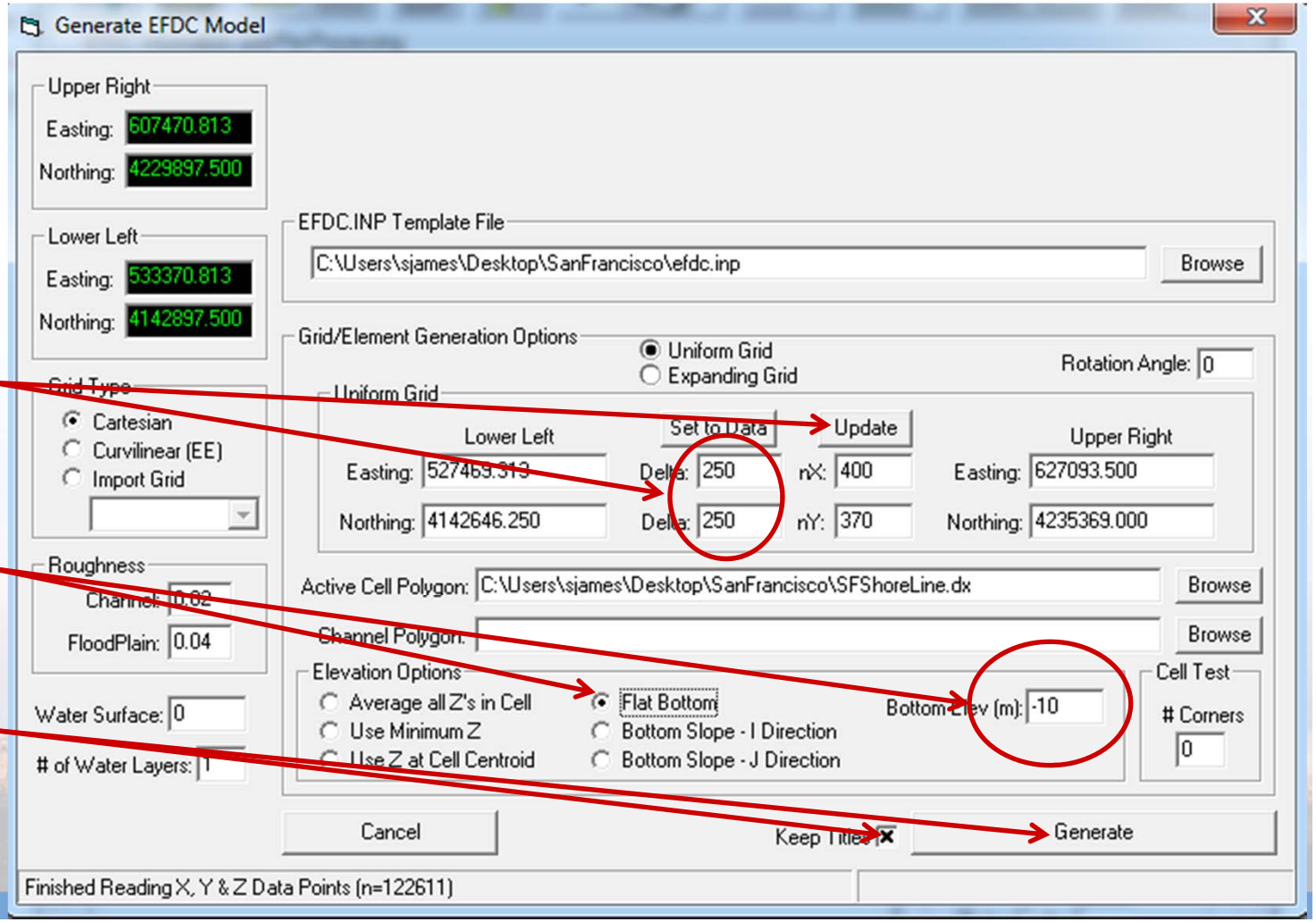
Then select the outline polygon data file (SFShoreLine.dx).

Building a Grid (Using EE)

Select grid-cell sizes and "Update"

Select a flat bottom and enter a value (−10 m)

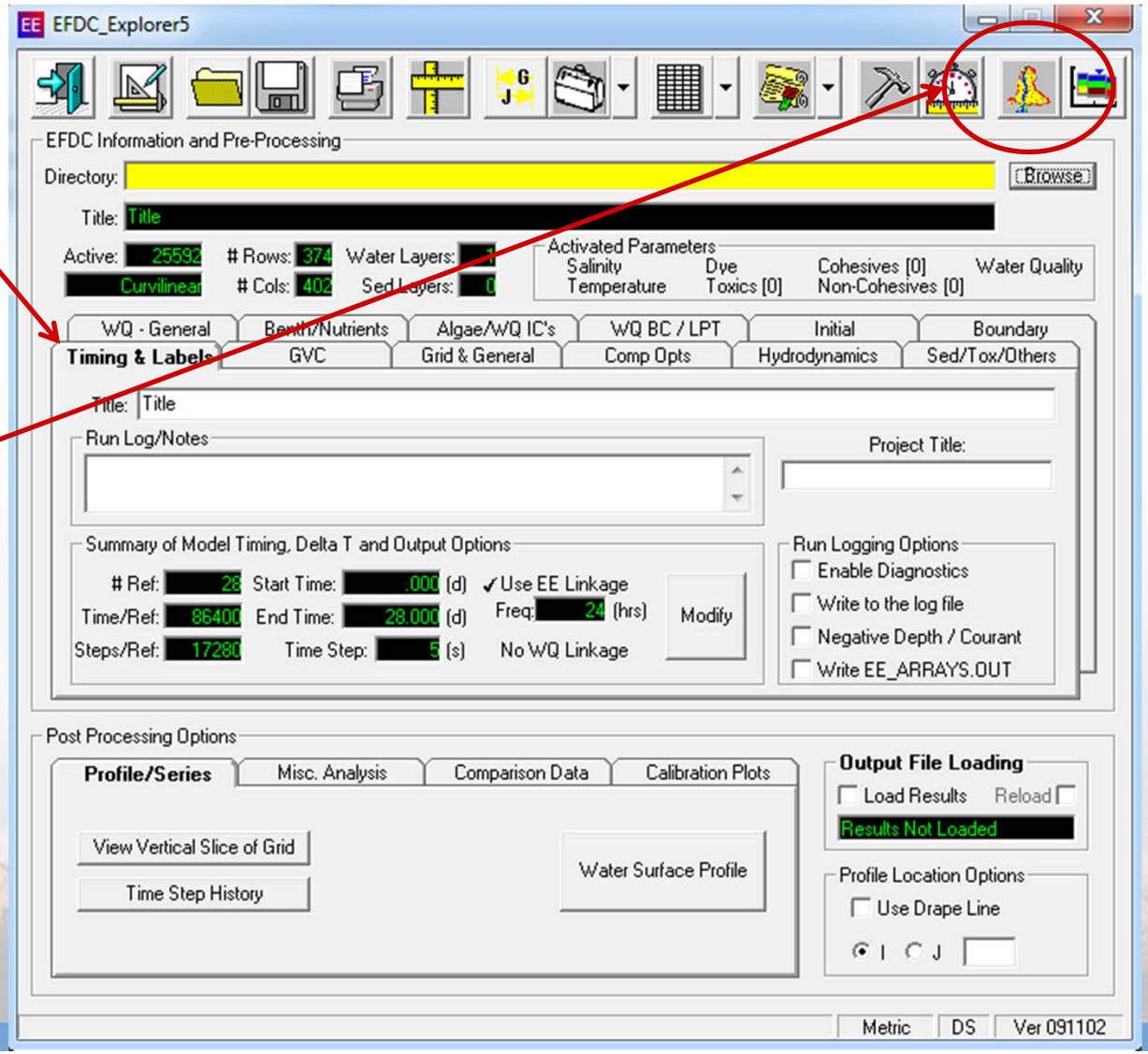
Keep titles and generate



Viewing the Grid (Using EE)

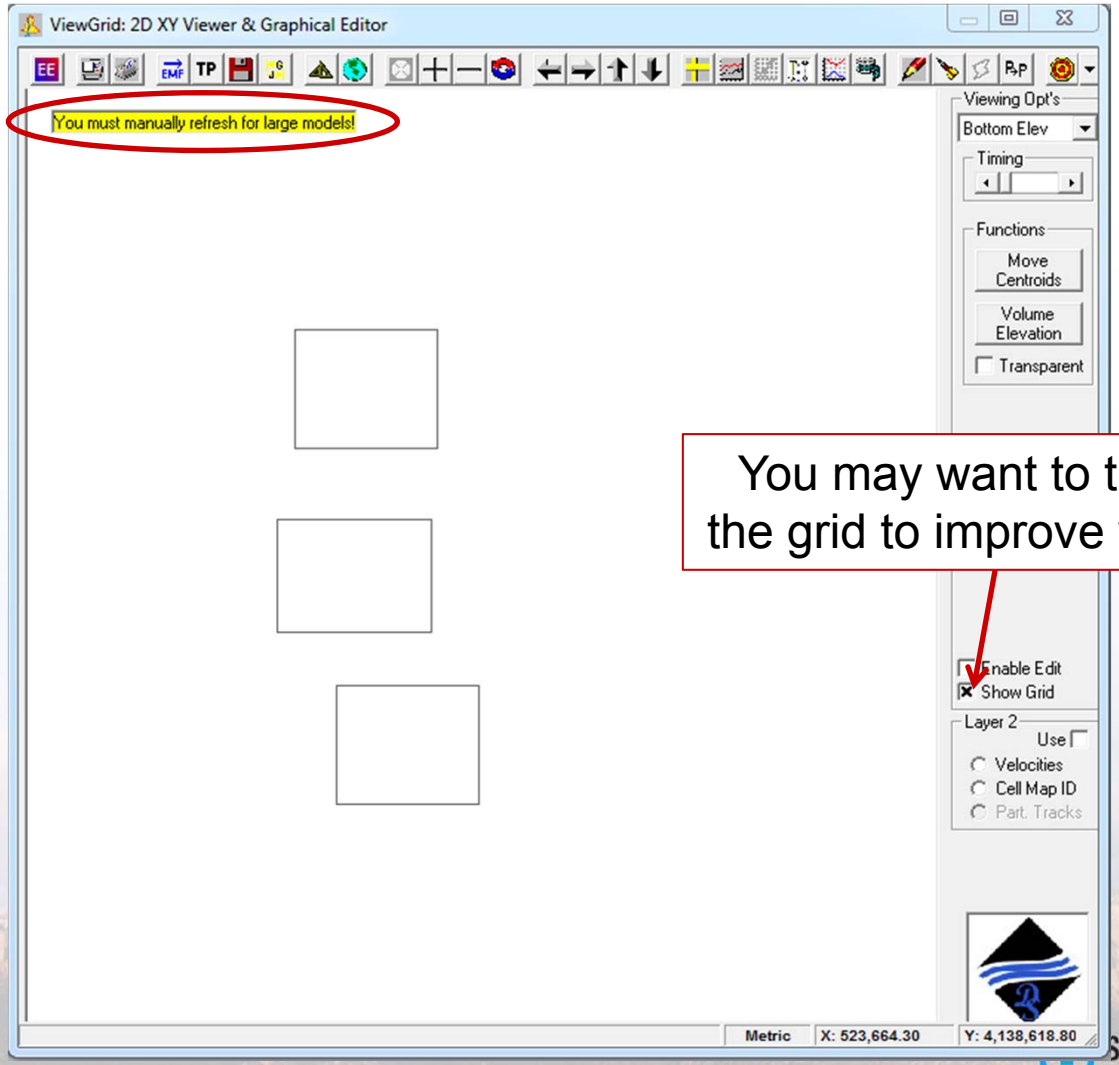
Note the 12 available tabs

Click "View Grid"



Laboratories

Viewing the Grid (Using EE)



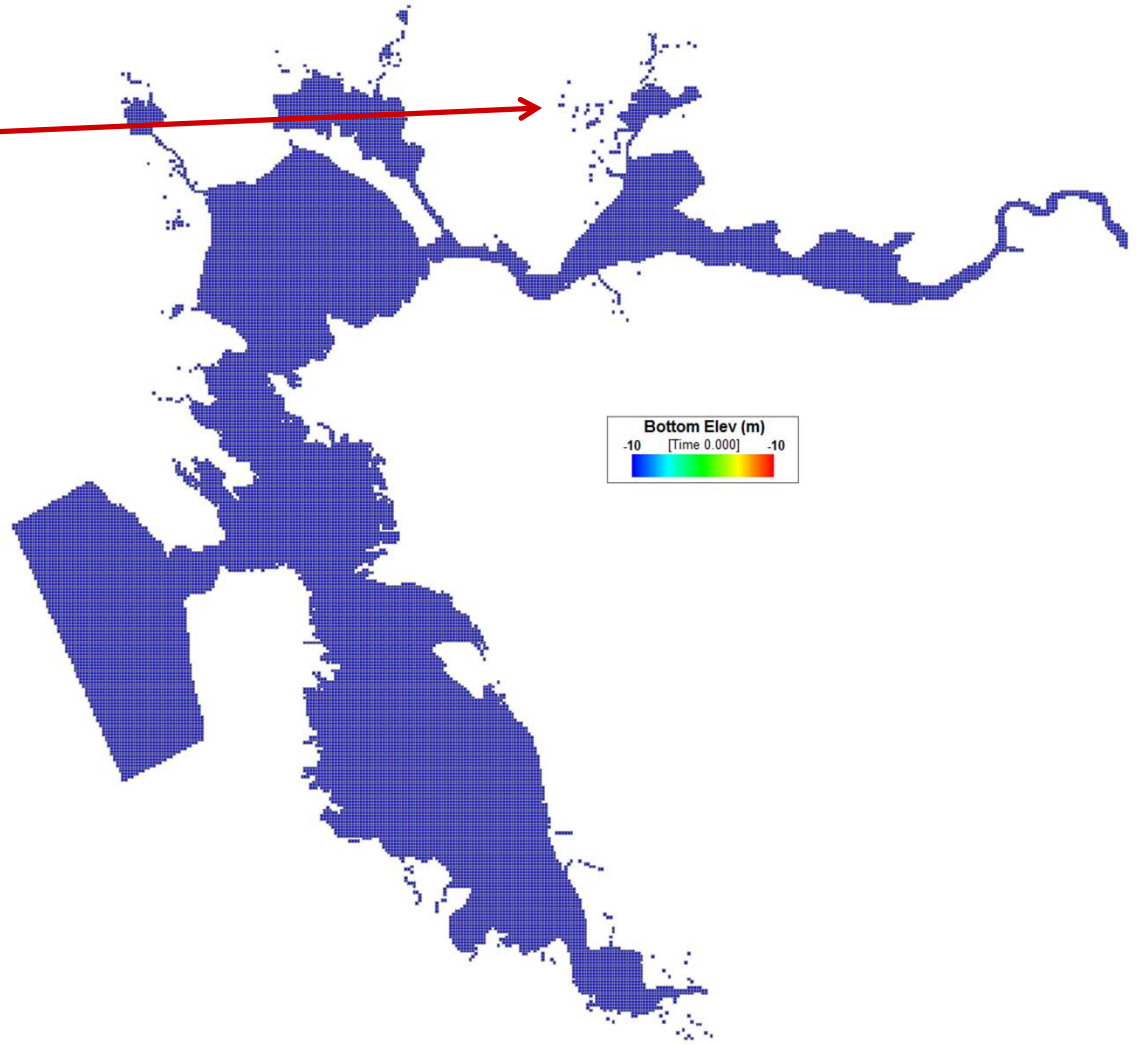
You may want to turn off the grid to improve visibility

Viewing the Grid (Using EE)

You can see some disconnected cells. These can be manually deleted by checking the “Enable Edit” button, right clicking each cell, and then clicking “Deactivate.”

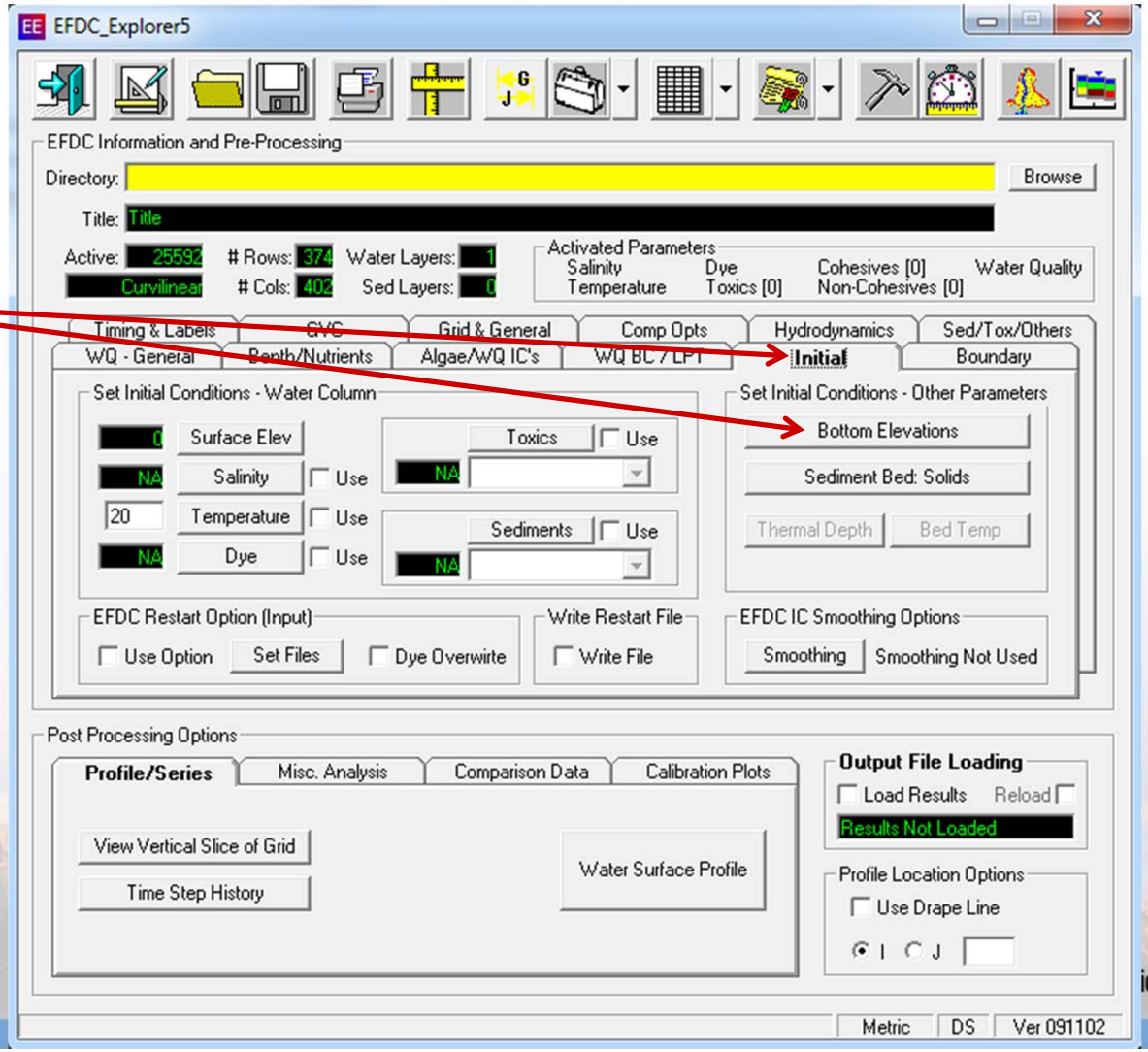
We will not worry about this for now.

Instead, we will assign bathymetry data now.



Adding Bathymetry (Using EE)

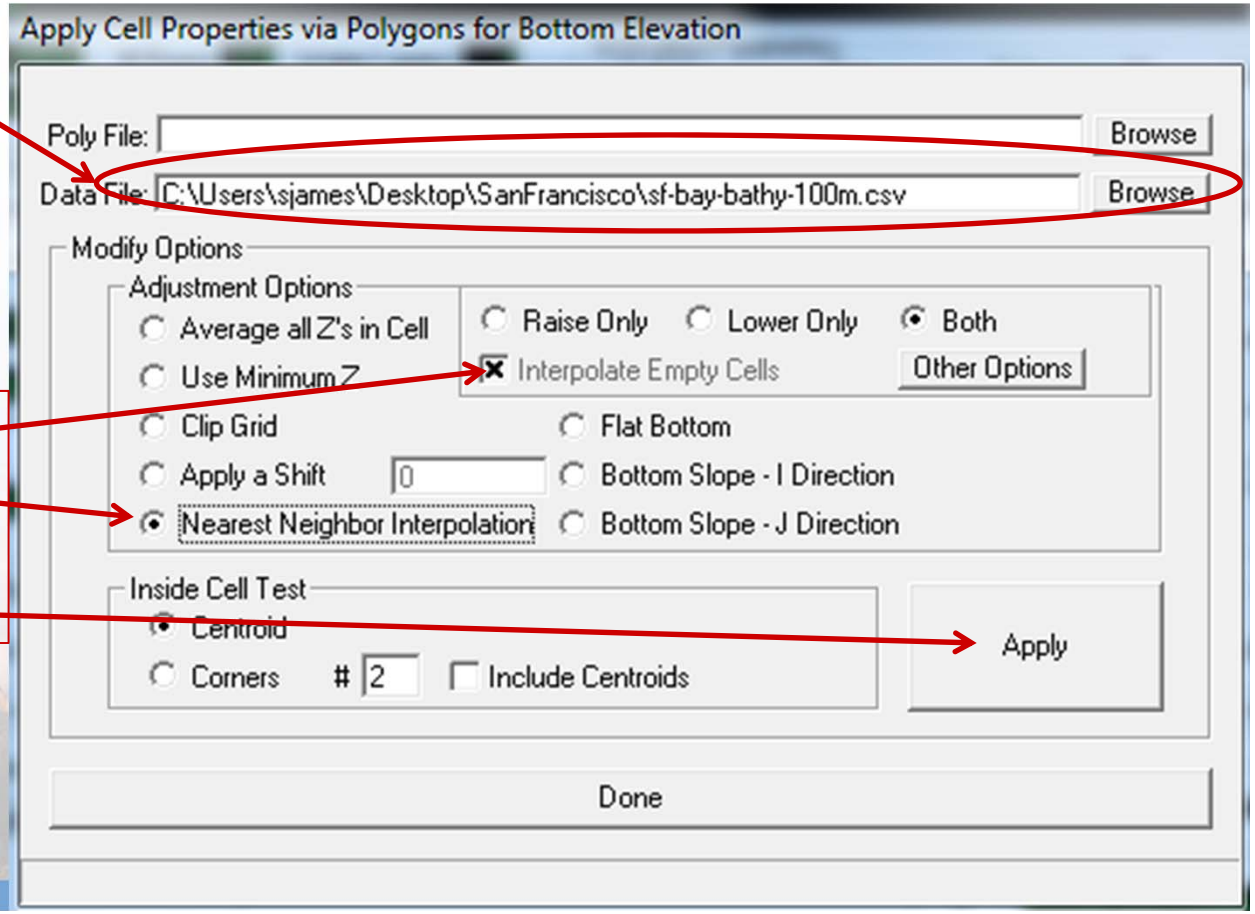
Switch to the "Initial" tab and then click on "Bottom Elevations"



Adding Bathymetry (Using EE)

Select the 100-m bathymetry data file

1. Interpolate Empty Cells
2. Nearest Neighbor Interpolation
3. Apply



Let's take a break…
and let the computer work.

Go back to "View Grid" when the bathymetry has been assigned.

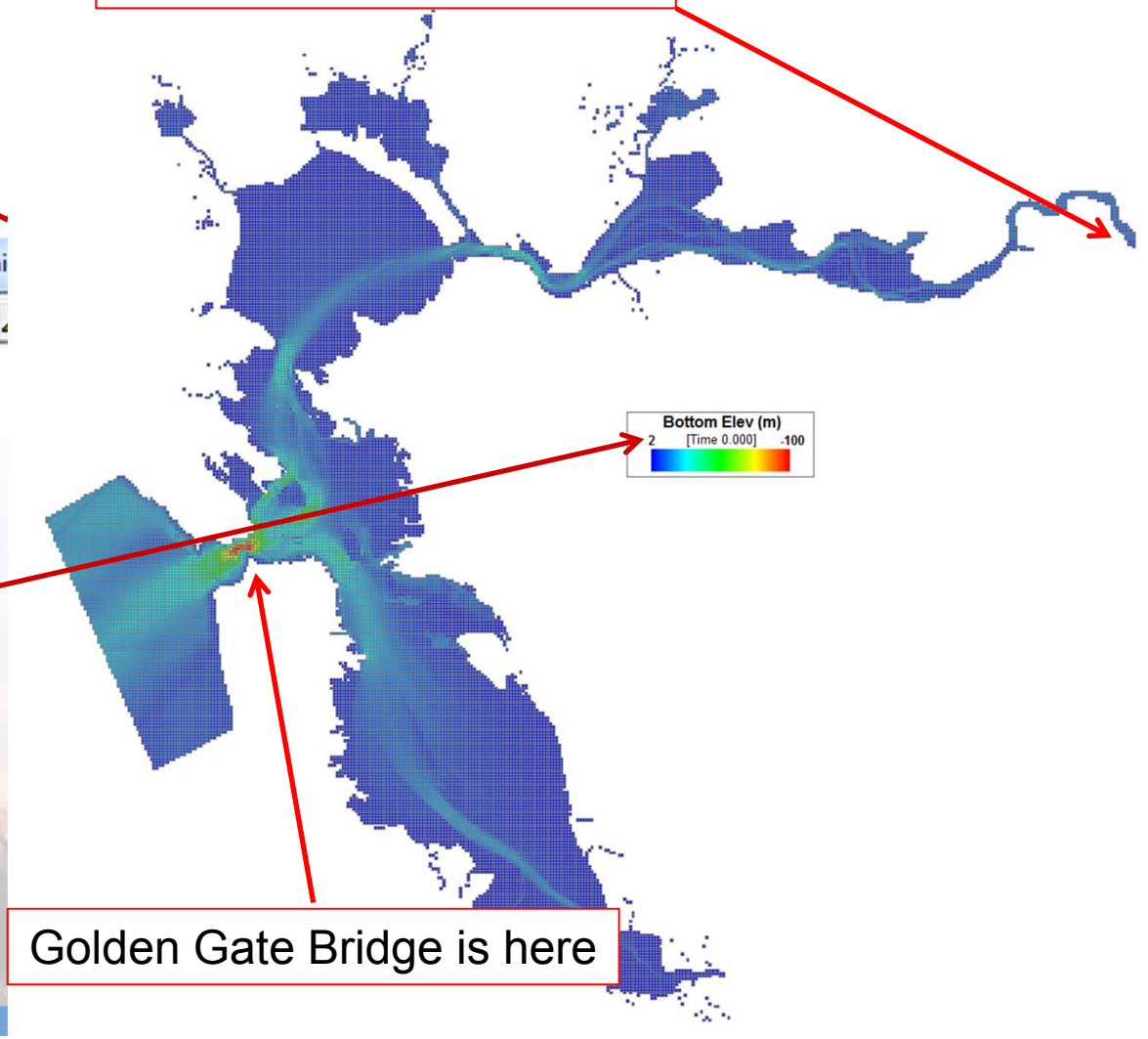
Sandia National Laboratories

What Did We Get?

You will need to click the "Refresh View" button.

Select "Bottom Elev" from the "Viewing Opt's" in the upper right corner.

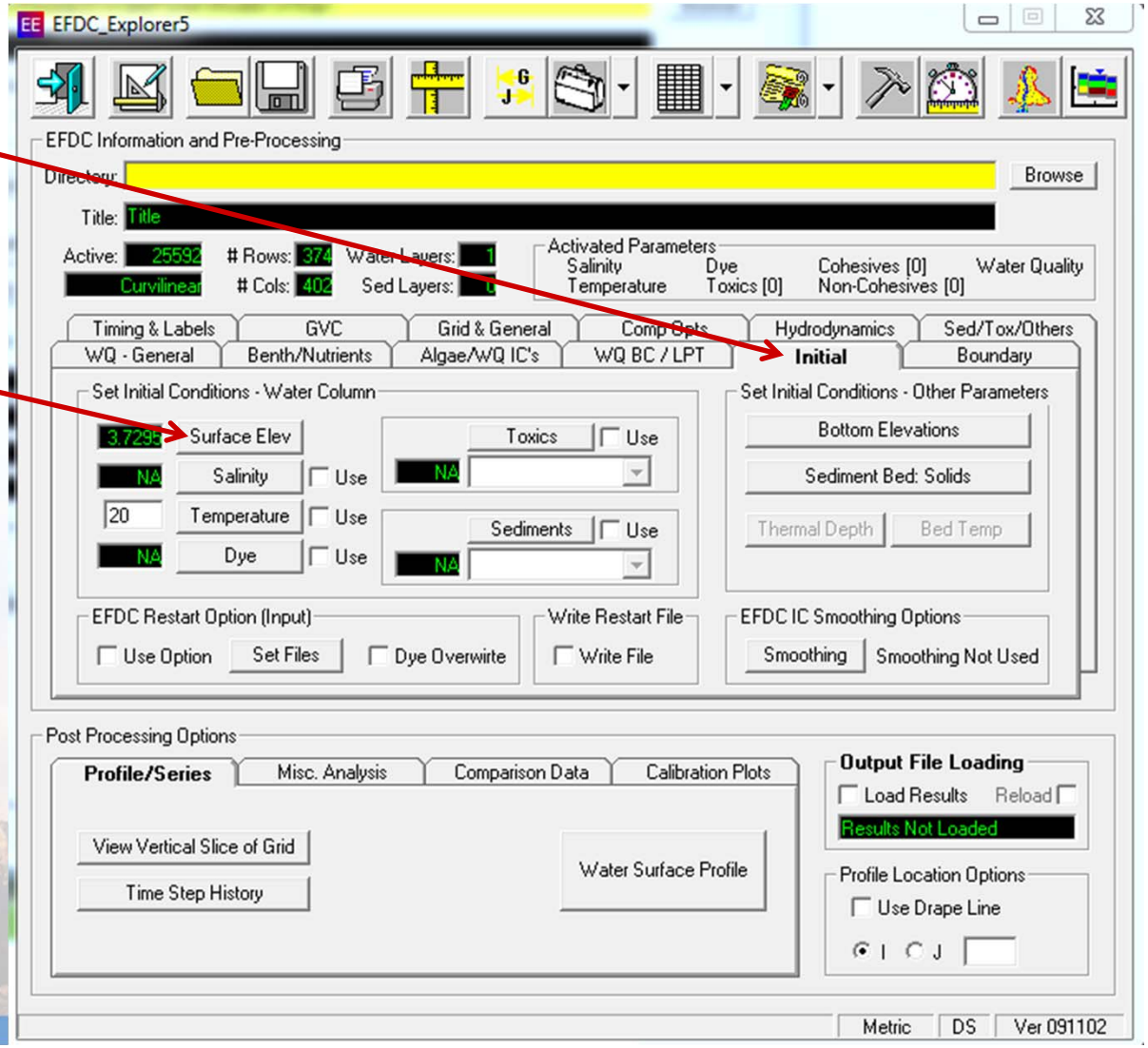
I right-clicked on the legend and adjusted the "Color Ramp" in the lower right corner.



Assigning Initial Conditions

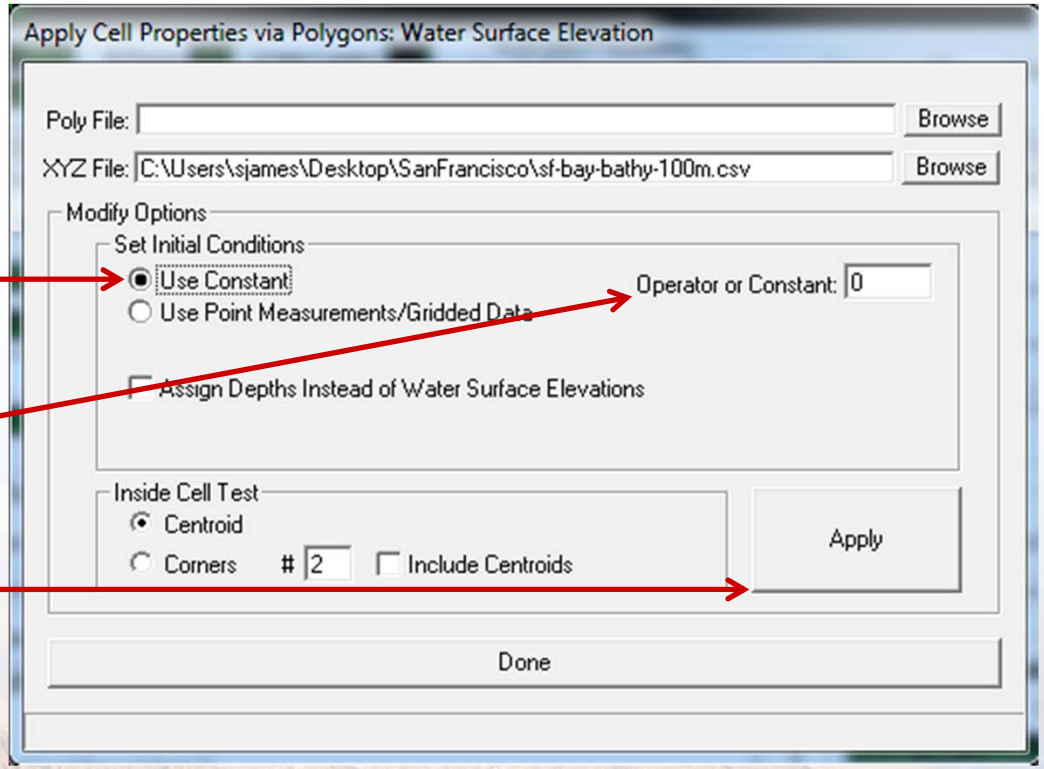
Click the “Initial” tab

Click “Surface Elev”



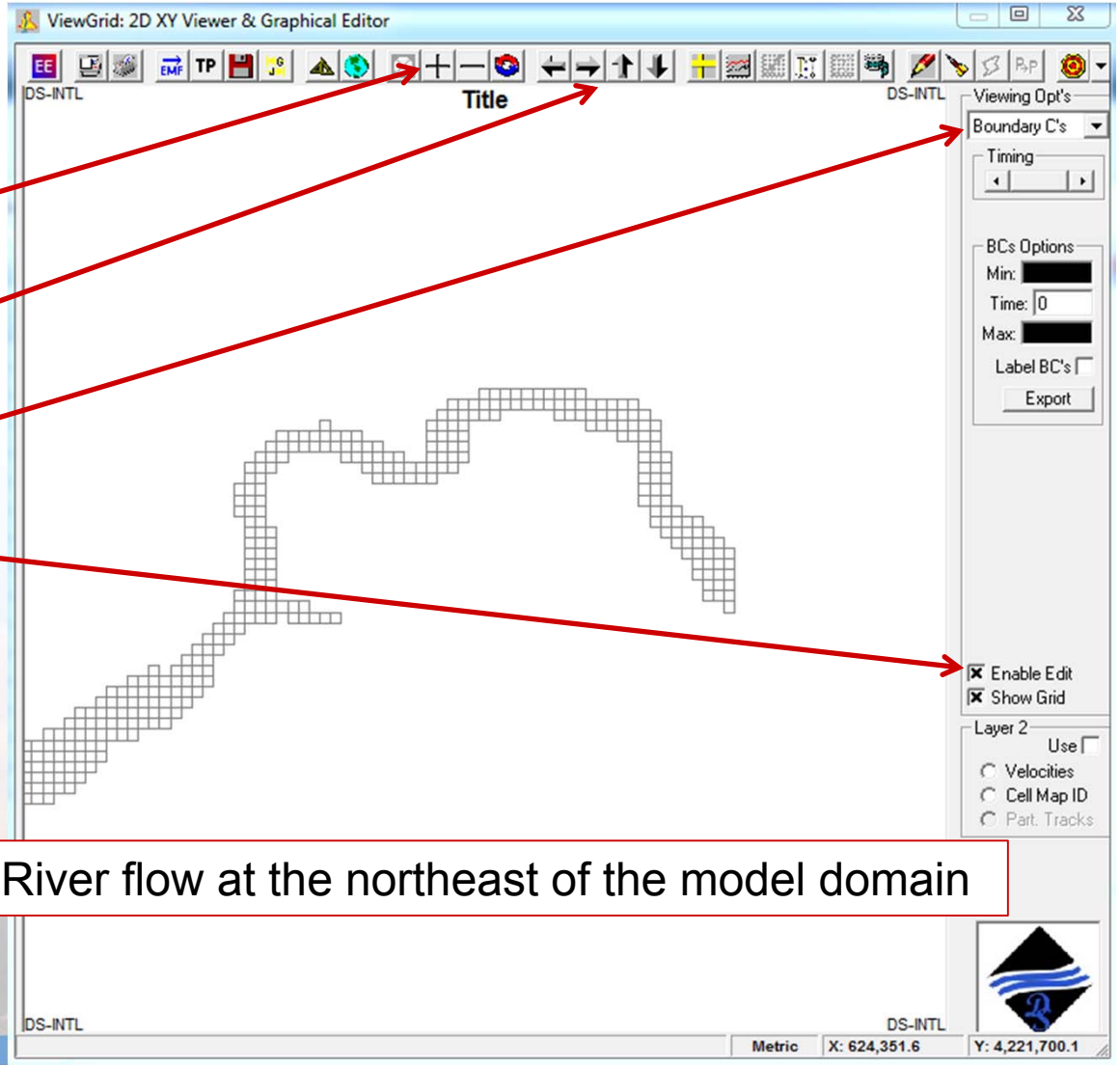
Assigning Initial Conditions

Select “Use Constant.”
Then enter the
“Operator or Constant”
(0 is fine).
“Apply” this value.



Assigning Boundary Conditions

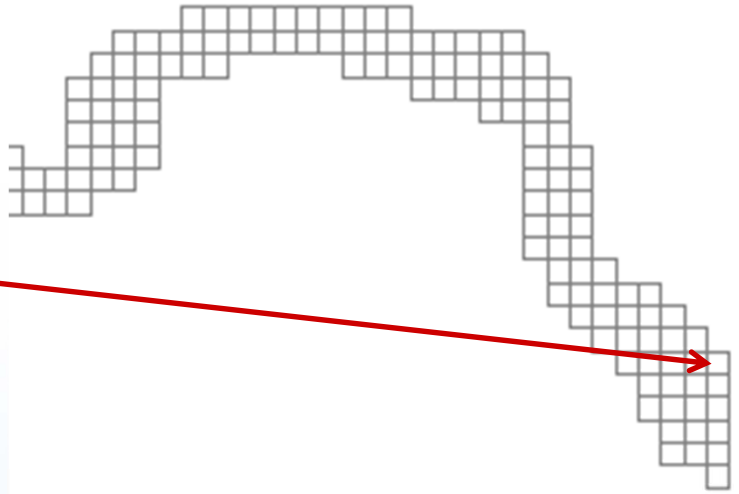
1. “Zoom in”
2. “Pan” appropriately
3. Select “Boundary C’s” from “Viewing Opt’s”
4. “Enable Edit”

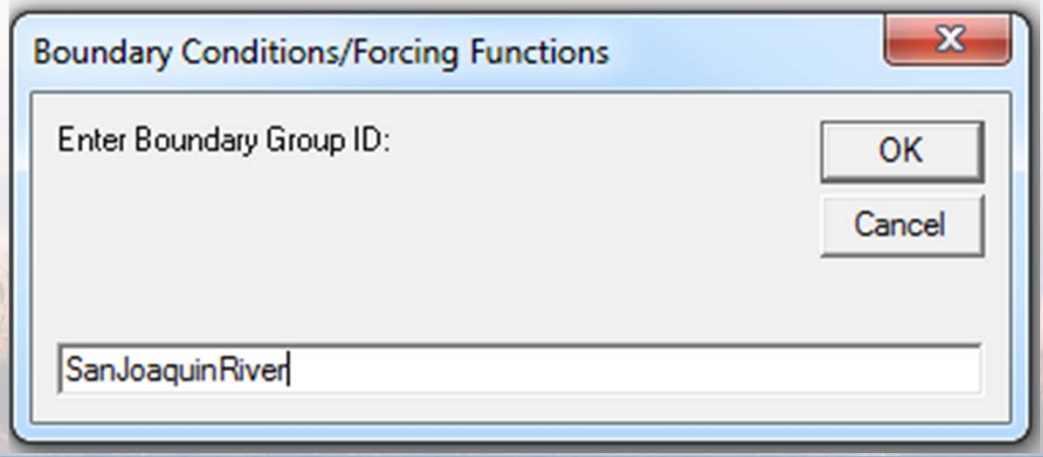


San Joaquin River flow at the northeast of the model domain

Assigning Boundary Conditions

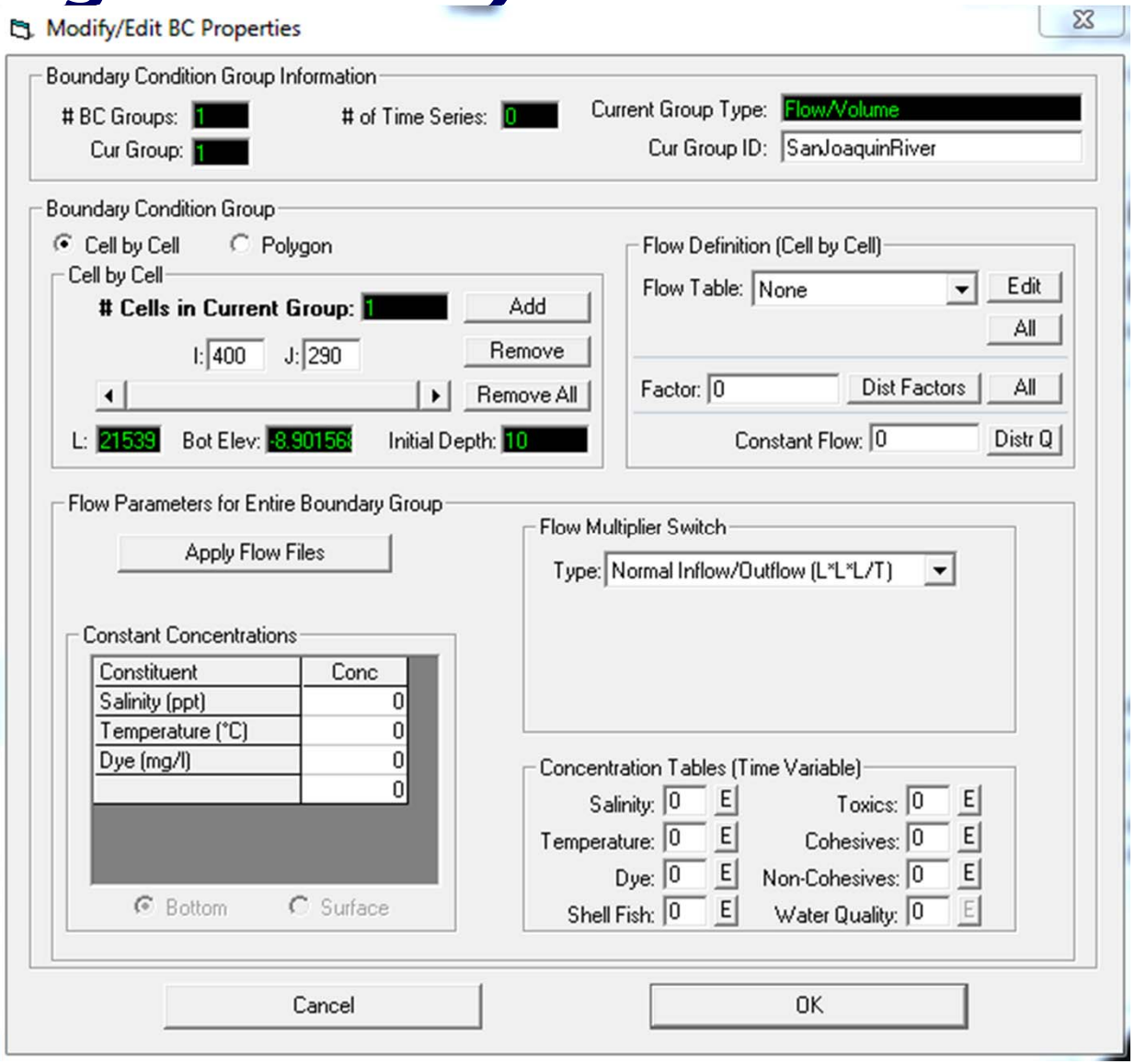
1. Right-click this cell
2. Click “New”
3. Enter an appropriate name





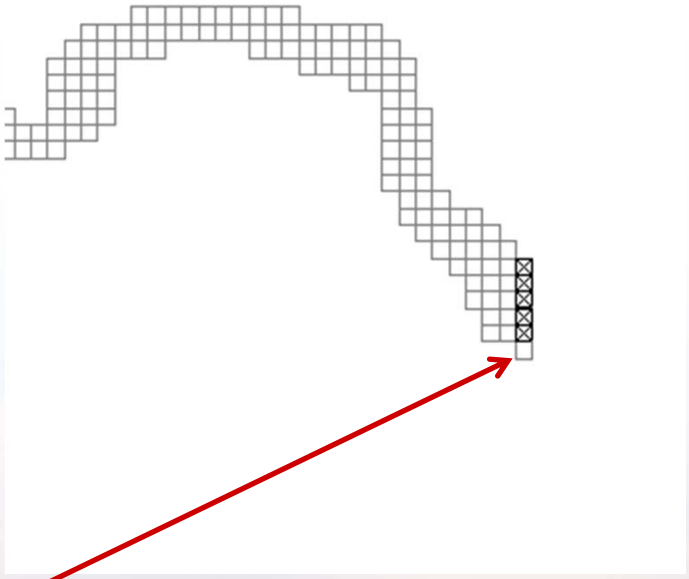
Assigning Boundary Conditions

1. Enter 1 for “Flow”
2. Then, just click “OK”
   (we will assign flows later)



Assigning Boundary Conditions

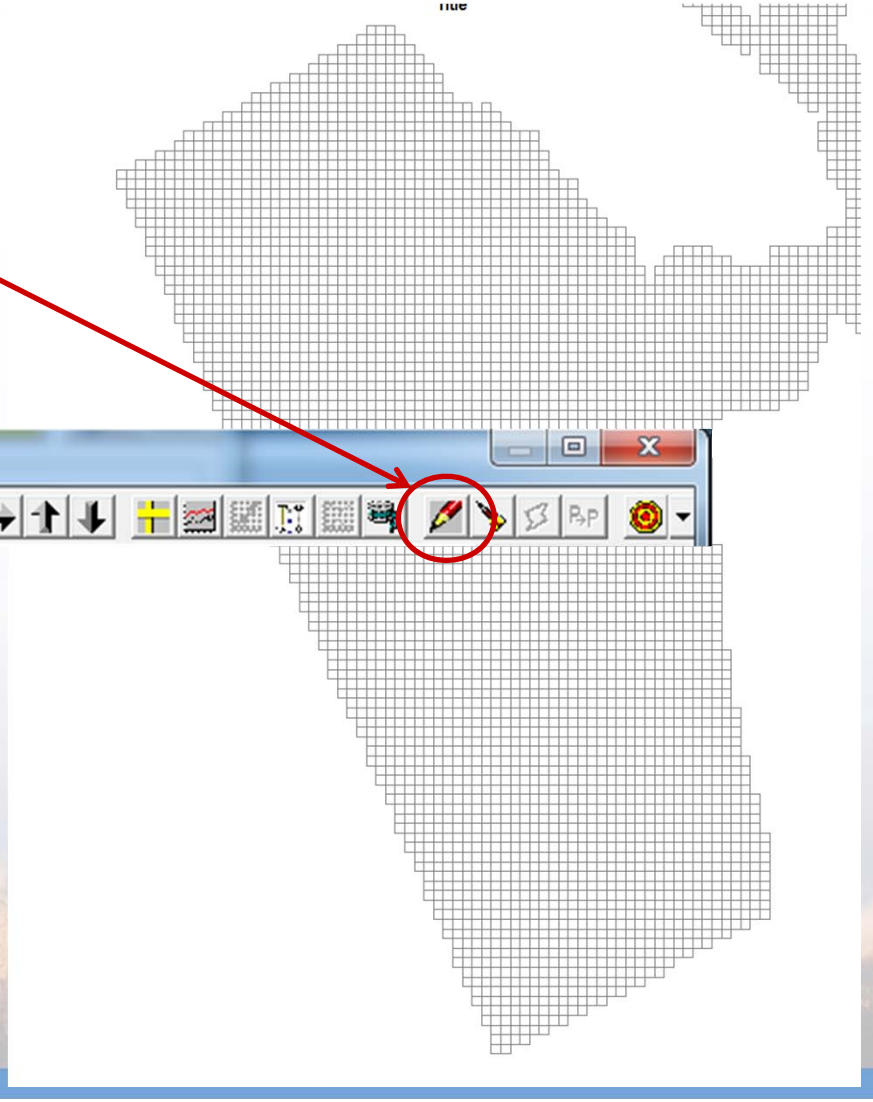
Right-click consecutive adjoining cells and "Add to Adjacent"



Why do we not assign this cell?

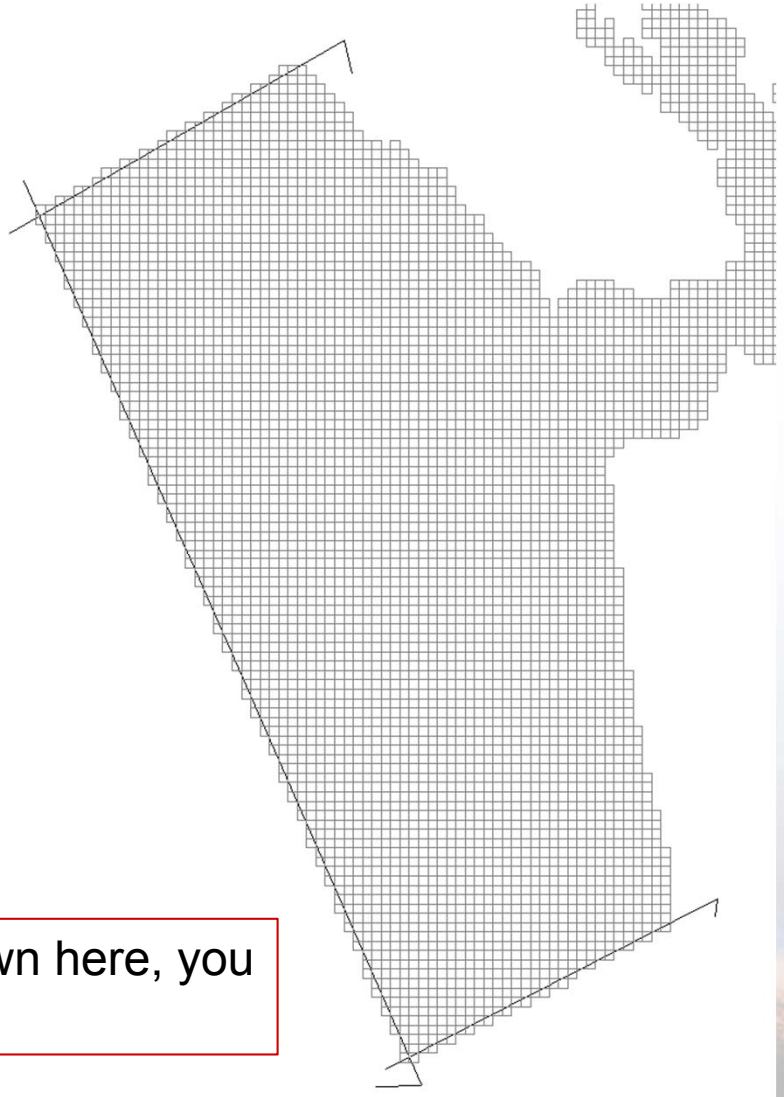
Assigning Boundary Conditions

We will assign the numerous open-ocean boundary cells using the “Active Polyline Tools.” Click it (after zooming in on the open-ocean region).



Assigning Boundary Conditions

1. First click “Cancel” as we will be creating three polylines (and not opening one).
2. Carefully follow the directions in the yellow box to create a polyline that intersect one of the three boundary edges (north, west, or south).
3. Enter “StringID” (name).
4. Unclick “Active Polyline Tools”
5. Enter a file name to save the file.
6. Do this three times (once for each boundary).
7. Exit the Grid Viewer.

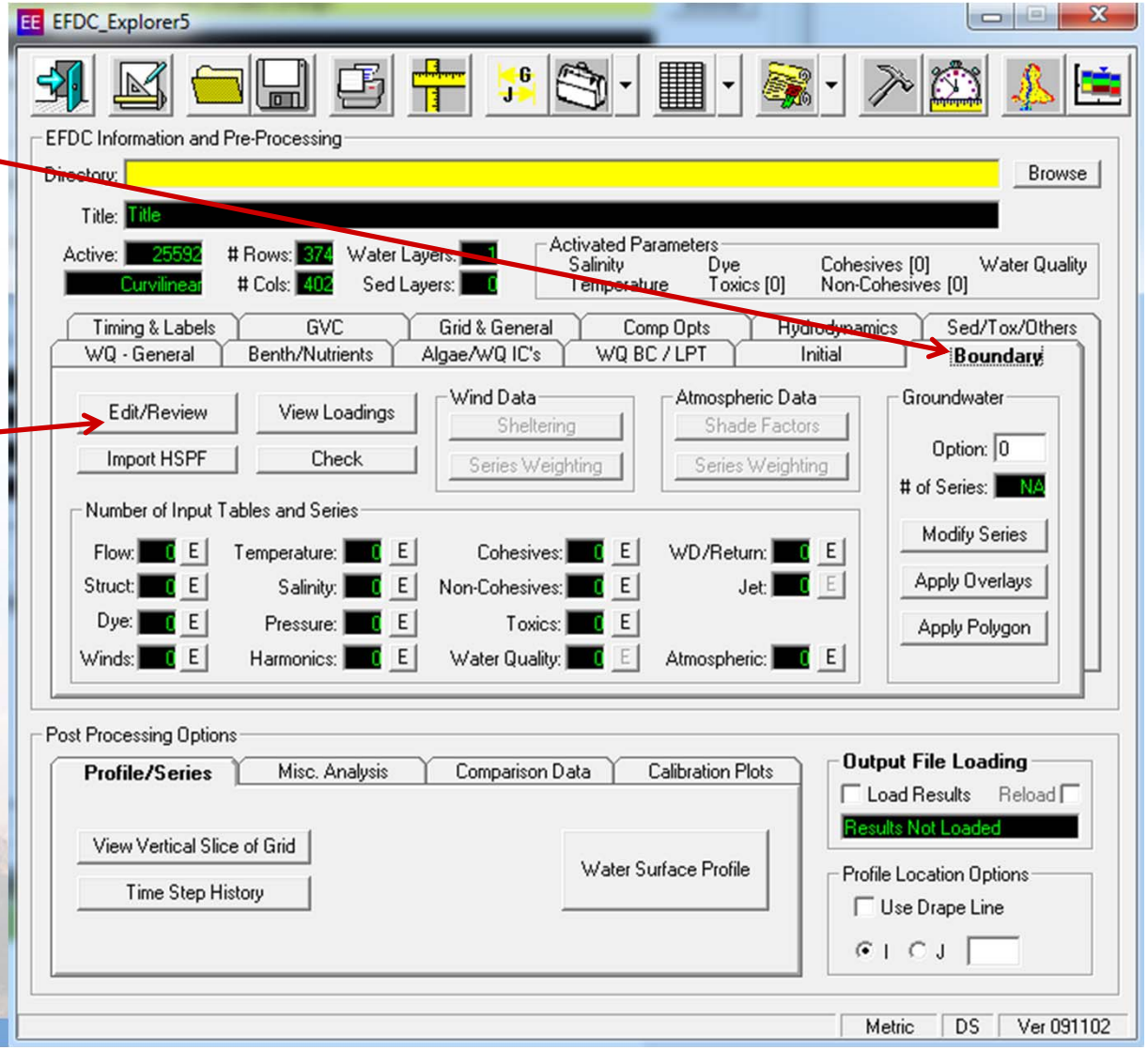


All three of my polylines are shown here, you will do one at a time.

Assigning Boundary Conditions

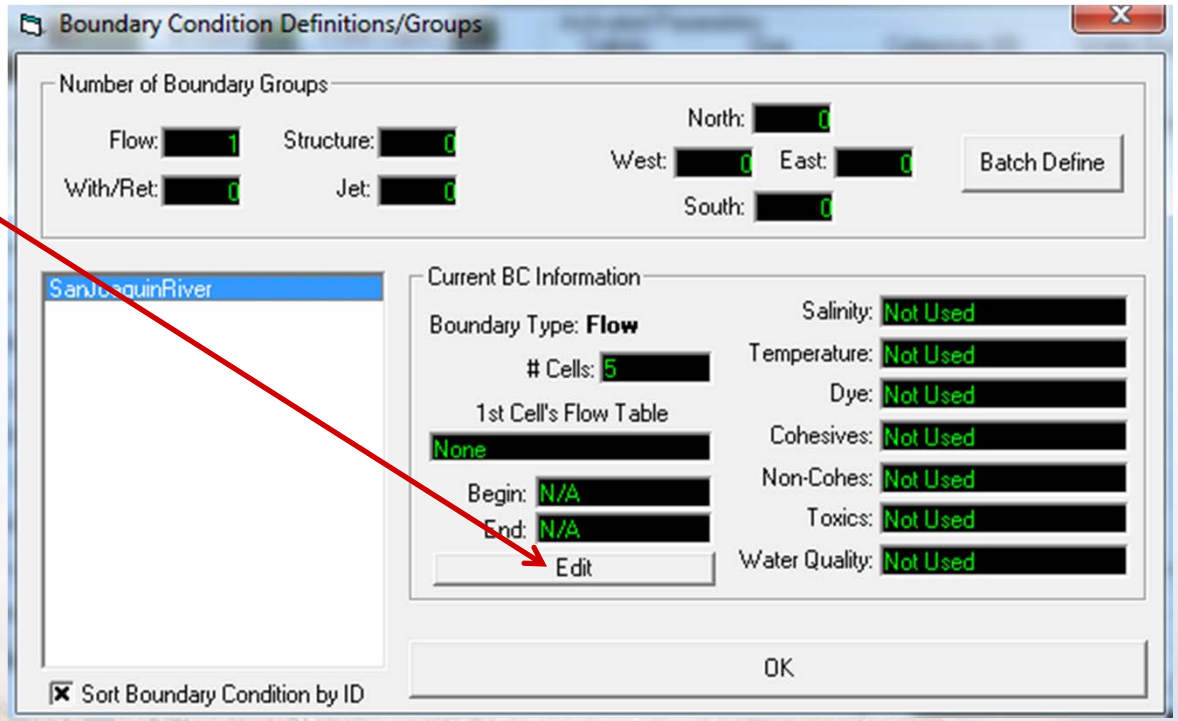
Switch to the "Boundary" tab

Click "Edit/Review"



San Joaquin River Steady Flow

Click “Edit”

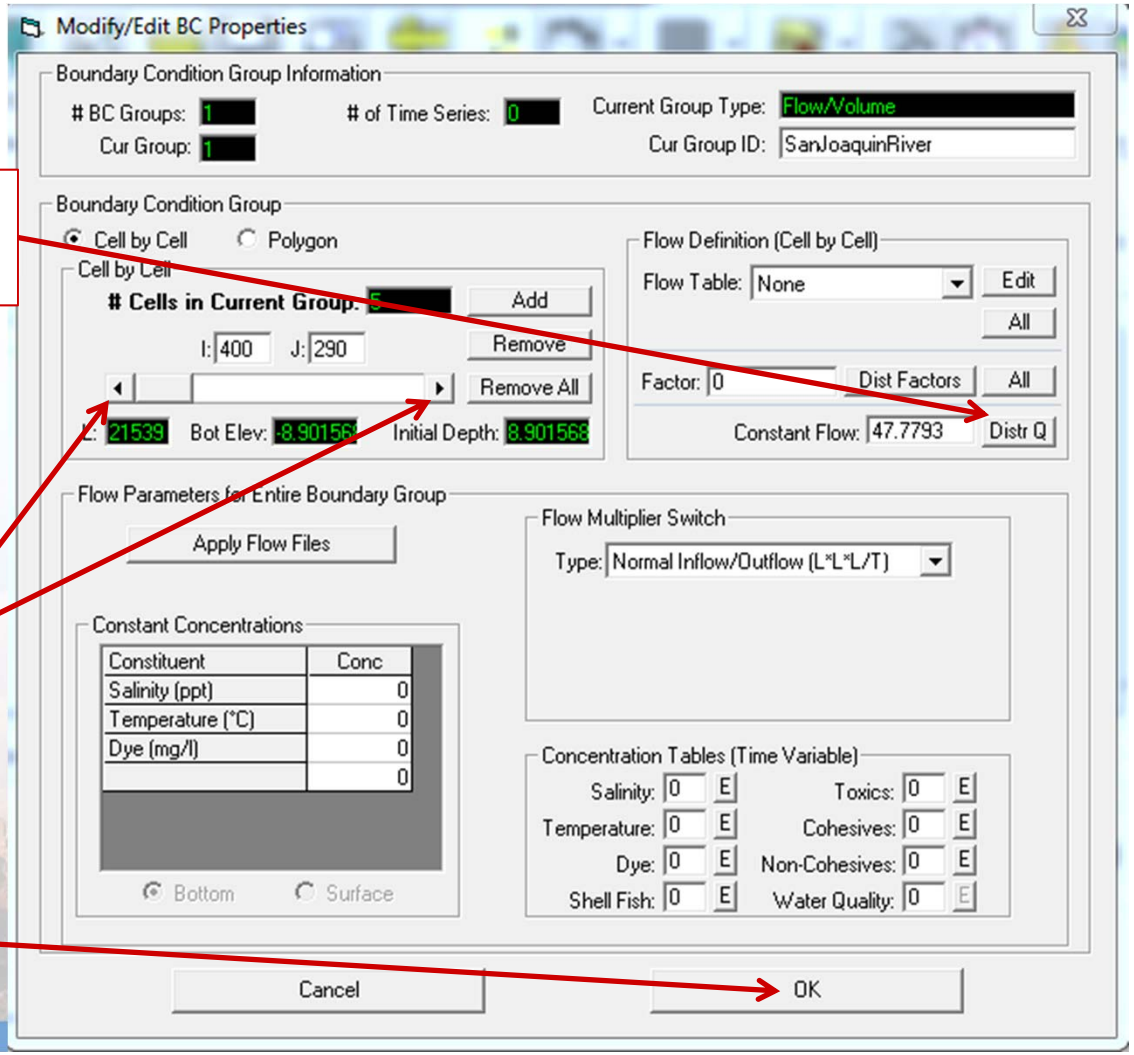


San Joaquin River Steady Flow

Click “Distr Q” and enter a flow rate. I used 180 m^3/s (cms).

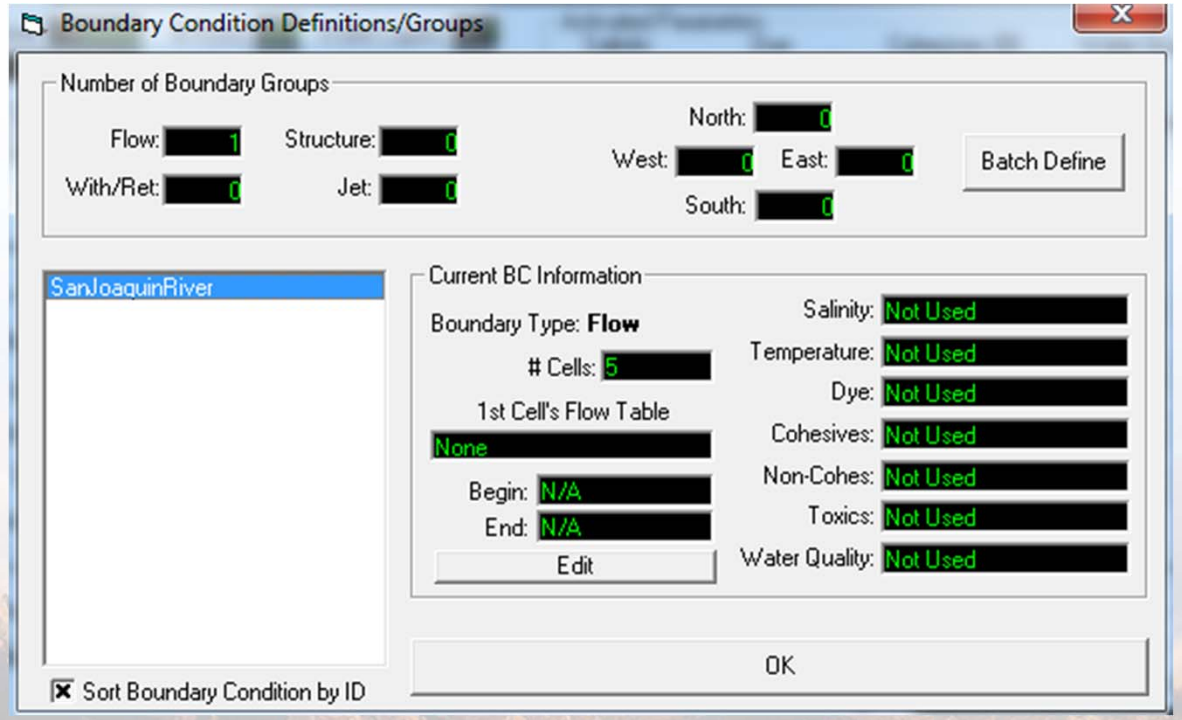
You can click here to see the flow applied to each of the five boundary cells.
Why are they different?

Click OK



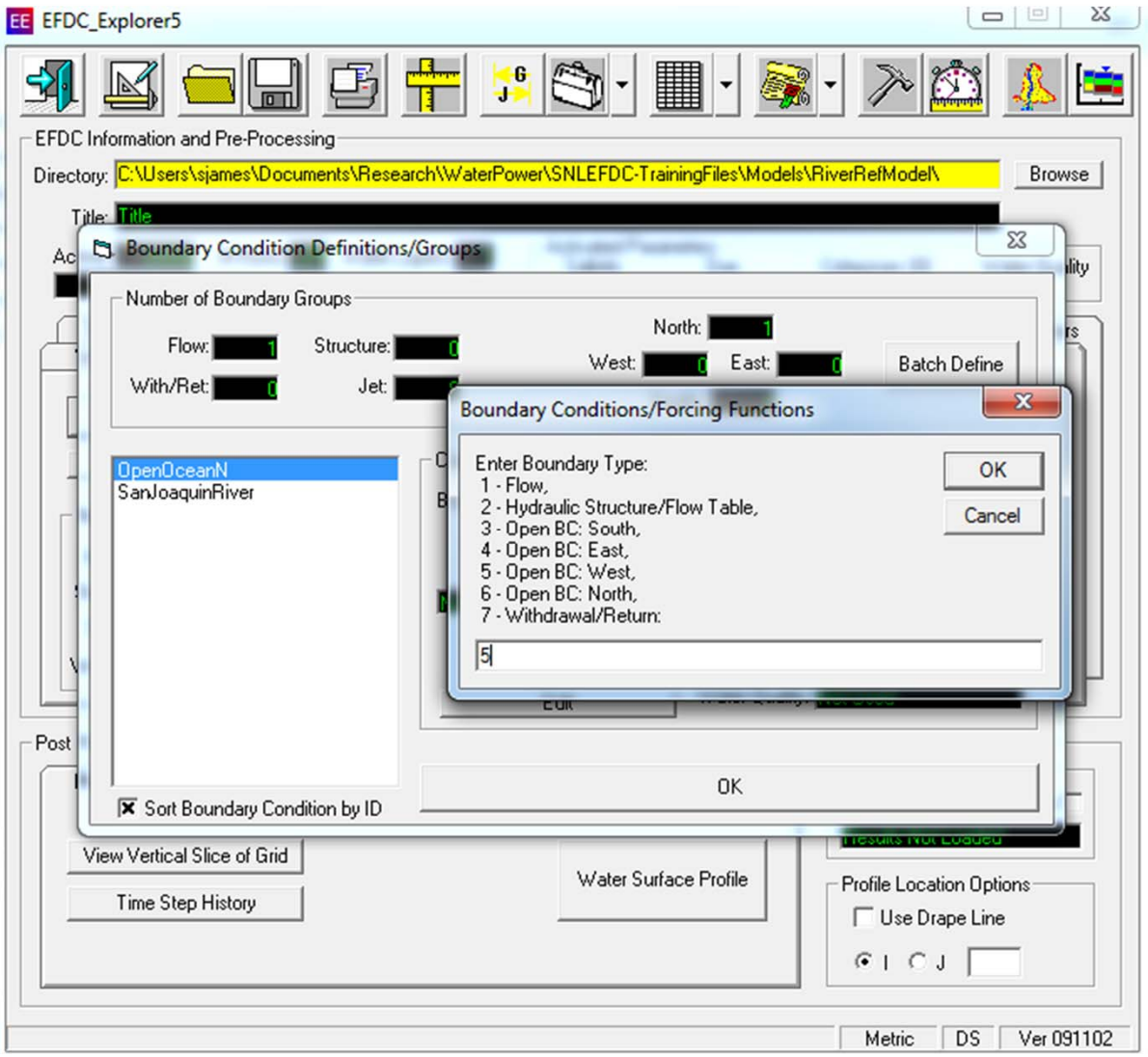
Open Ocean Heads

Click the Insert button on your keyboard (after mouse clicking in the white space). Enter a name for the Open Ocean Boundary. Click “OK”



Open Ocean Heads

Enter (the appropriate) Boundary Type:
"Open BC:" on the
- "South" - 3
- "West" - 5
- "North" - 6



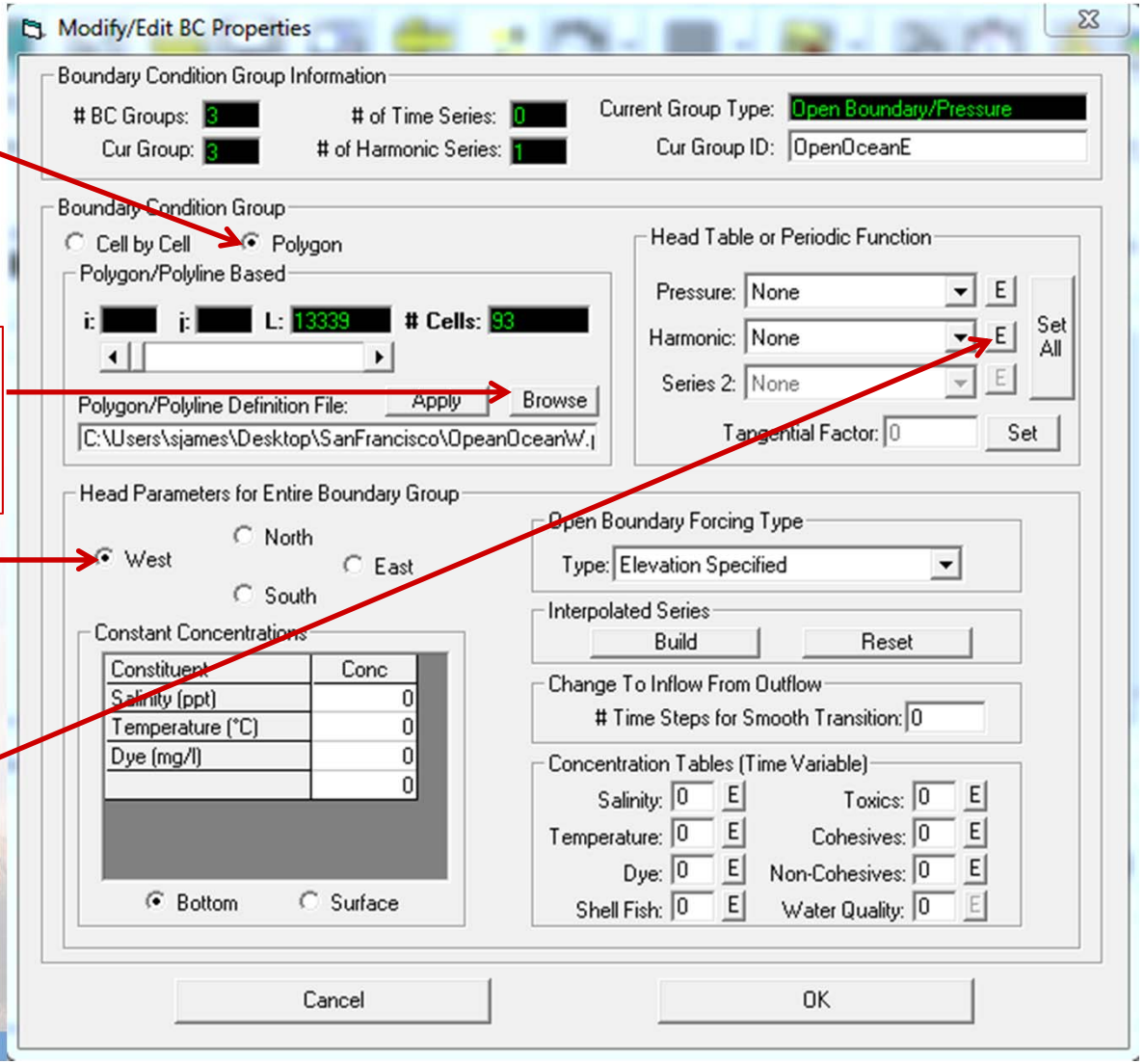
Open Ocean Heads

Assign the "Boundary Condition Group" by "Polygon"

Select the appropriate "Polygon/Polyline Definition File" and "Apply"

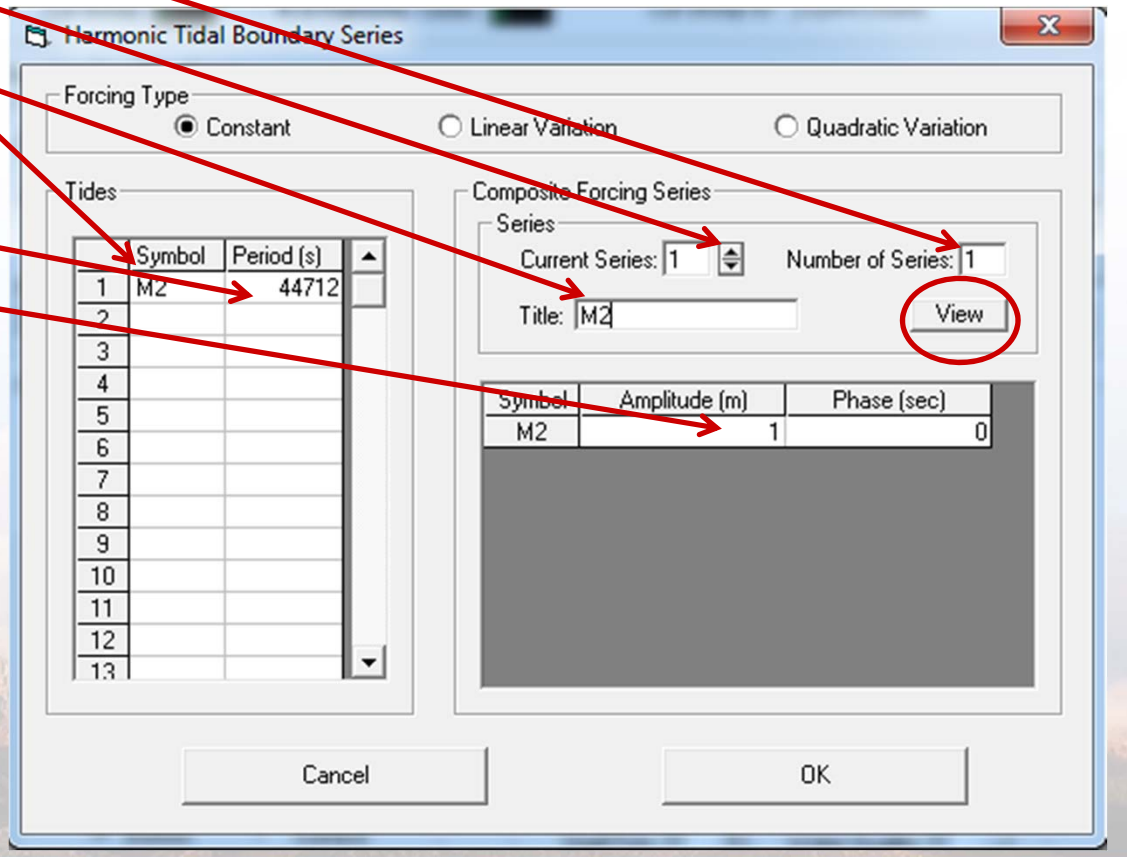
Make sure this agrees…

Assign a "Harmonic" series



Open Ocean Heads

First, enter the “Number of Series” (1 - Enter)
Second, click the “Current Series” to 1
Third, enter “Symbol” and “Title” (M2)
Fourth, enter the “Period” (44712, which is 12.42 hr)
Fifth, enter the “Amplitude” (1)
Sixth, “View” the results
Seventh, click “OK”

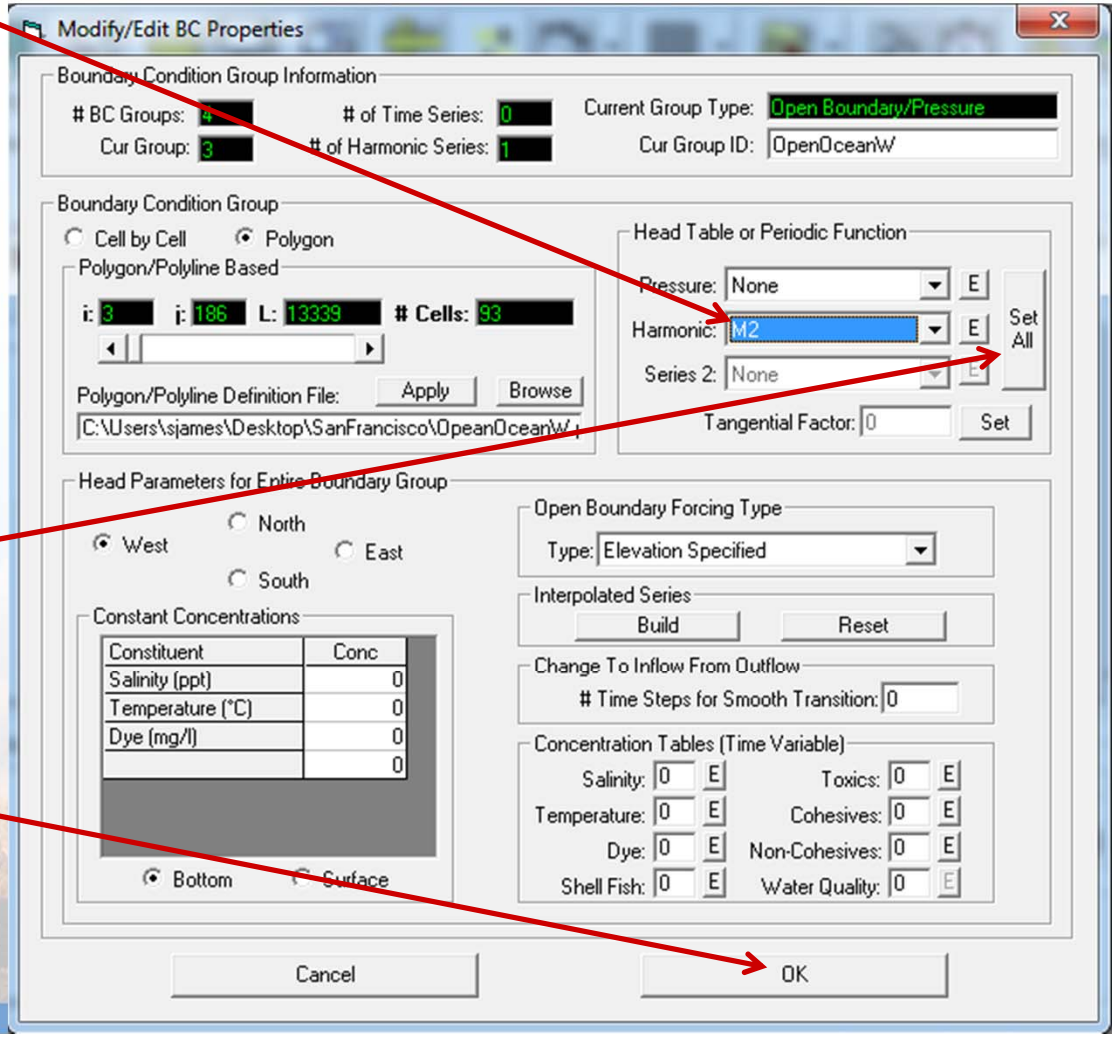


Open Ocean Heads

Select the “M2” Harmonic data

“Set All” of the cells in this group to this boundary condition.

Click “OK”

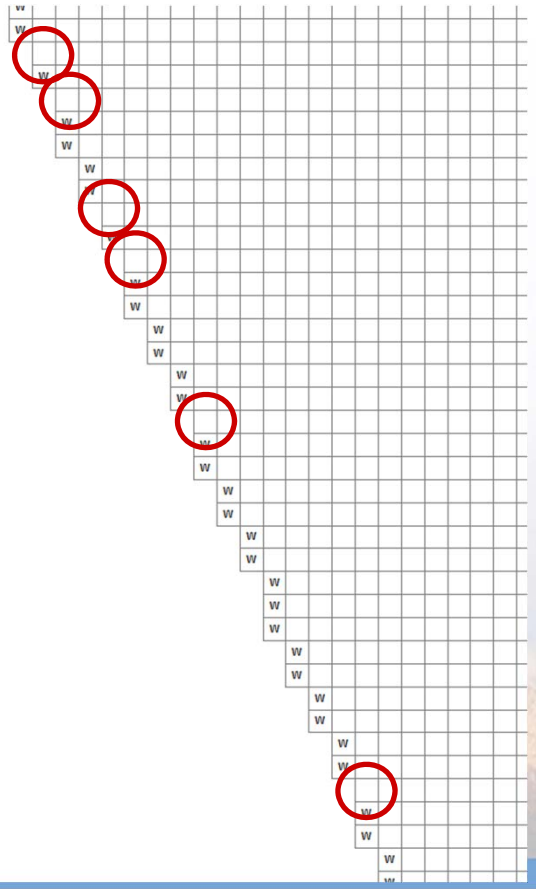


Open Ocean Heads

1. Repeat the same steps twice more to assign the remaining of the north, west, or south boundaries.
2. Then, lets look at the boundary conditions in the “Grid Viewer.”
3. We will need to clean up our boundaries, make sure you “Enable Edit.”
4. Make sure your “Viewing Opt’s” are set to “Boundary Cs.”

Open Ocean Heads

I see several cells that need to be assigned boundary conditions

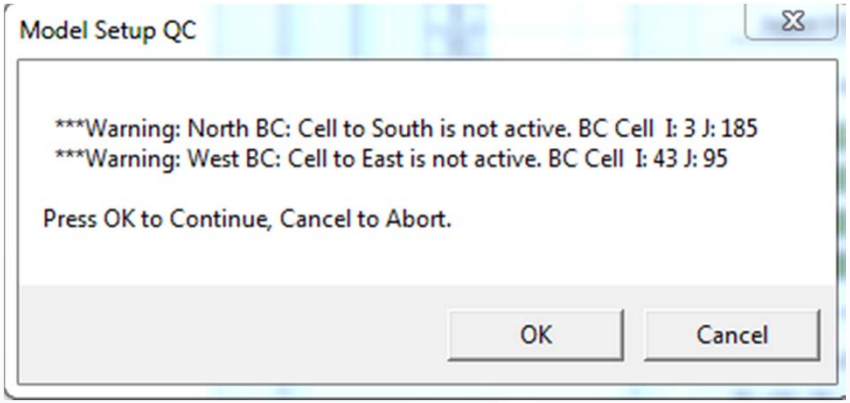


Right click and "Add to Adjacent"

Exit and Save

Open Ocean Heads

Are there any problems?



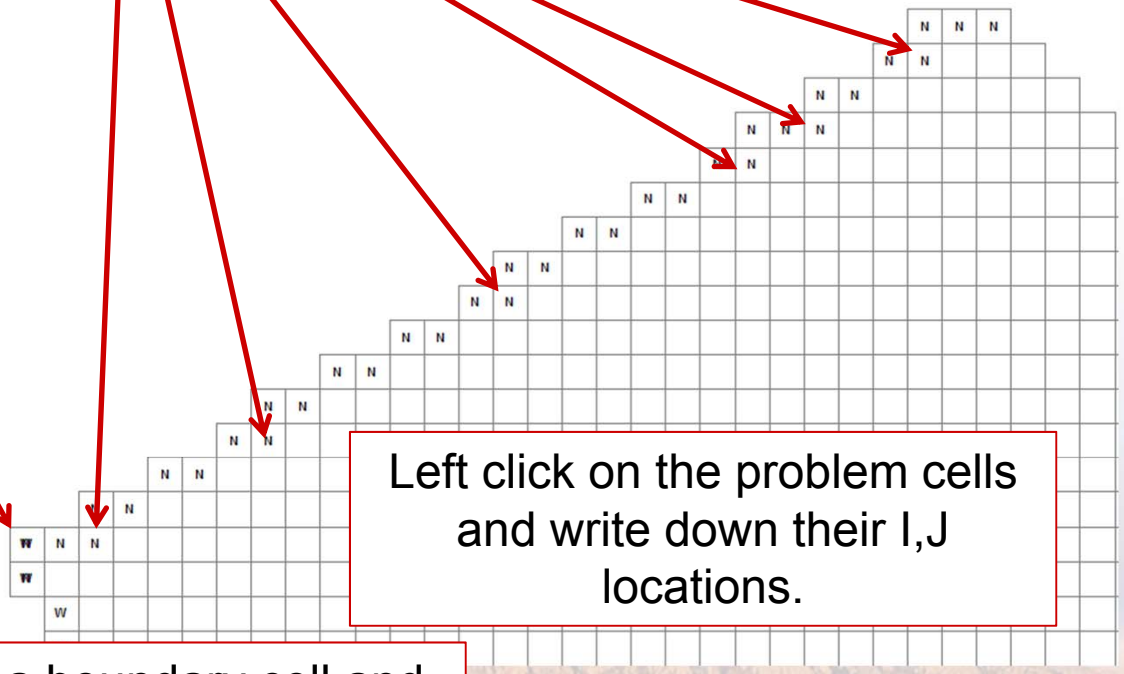
There may be more than just this, let's clean up the boundary cells.

Sandia National Laboratories

Open Ocean Heads

Don’t need N boundary conditions to the south of N BC’s

Extraneous
(and double assigned)



Left click on the problem cells and write down their I,J locations.

Then, right click on a boundary cell and click “Edit”

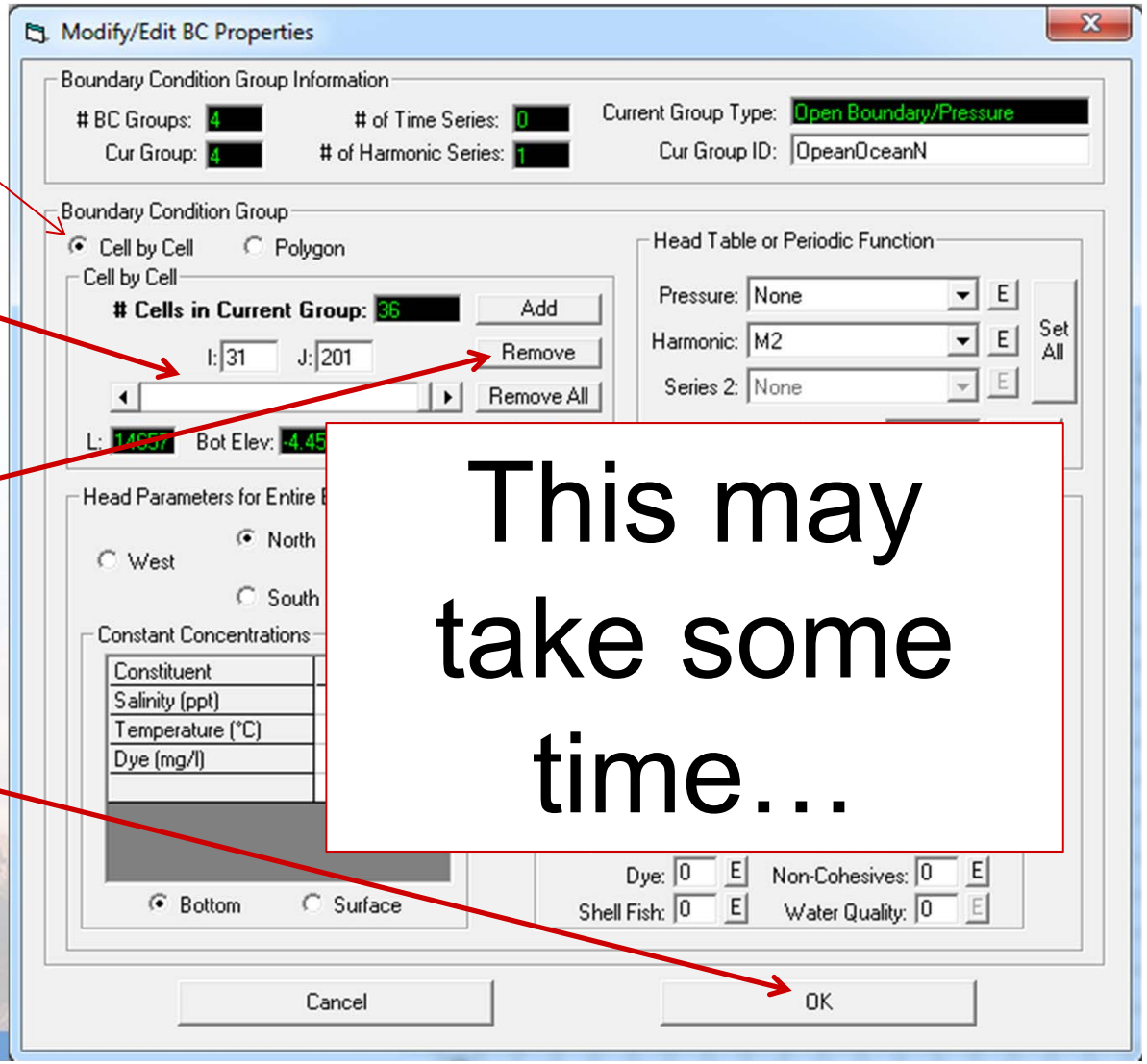
Open Ocean Heads

Click “Cell by Cell”

Cycle through and find the bad cells.

Remove them

Click OK when done

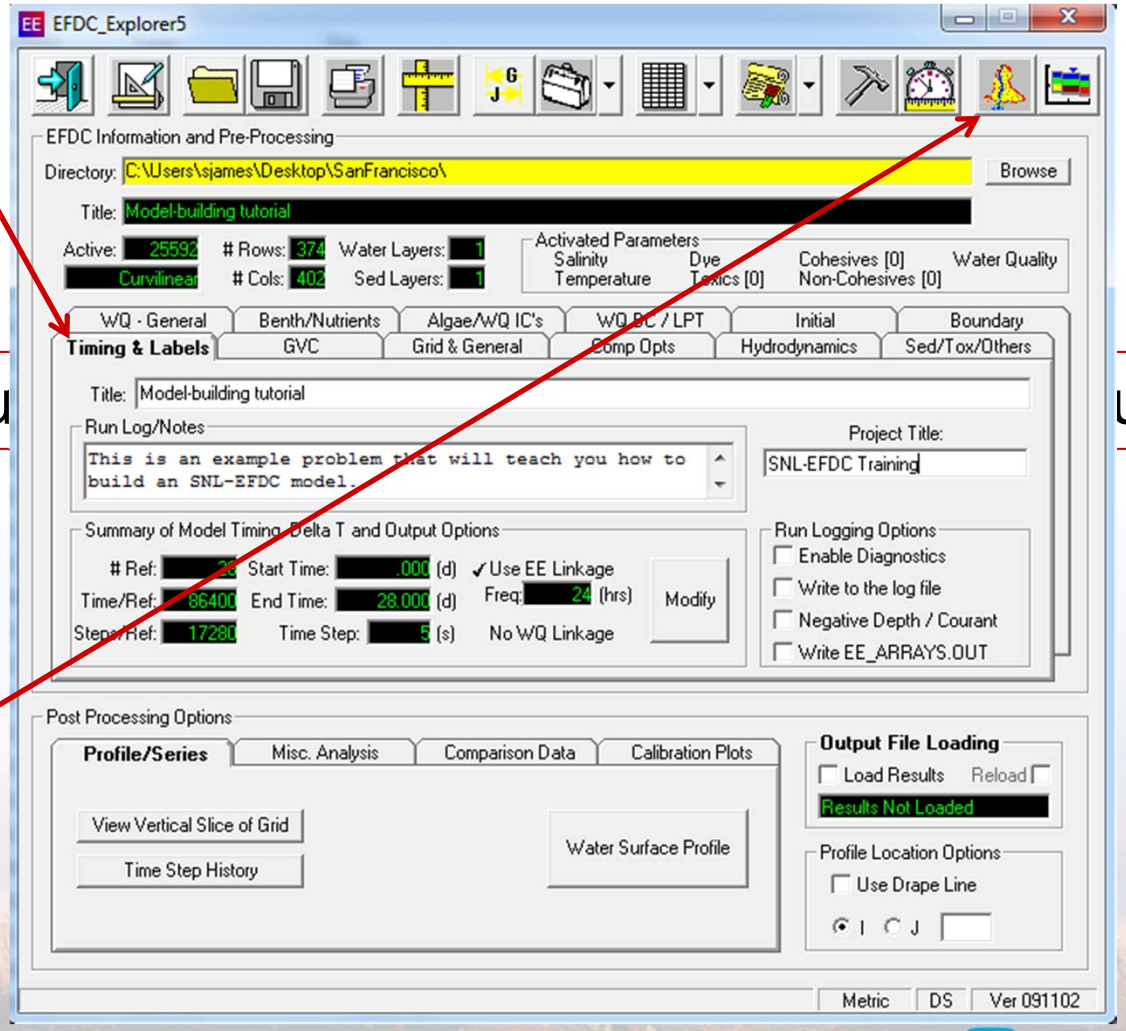


The Time Step

Click the "Timing & Labels" tab

Let's run throu ues…

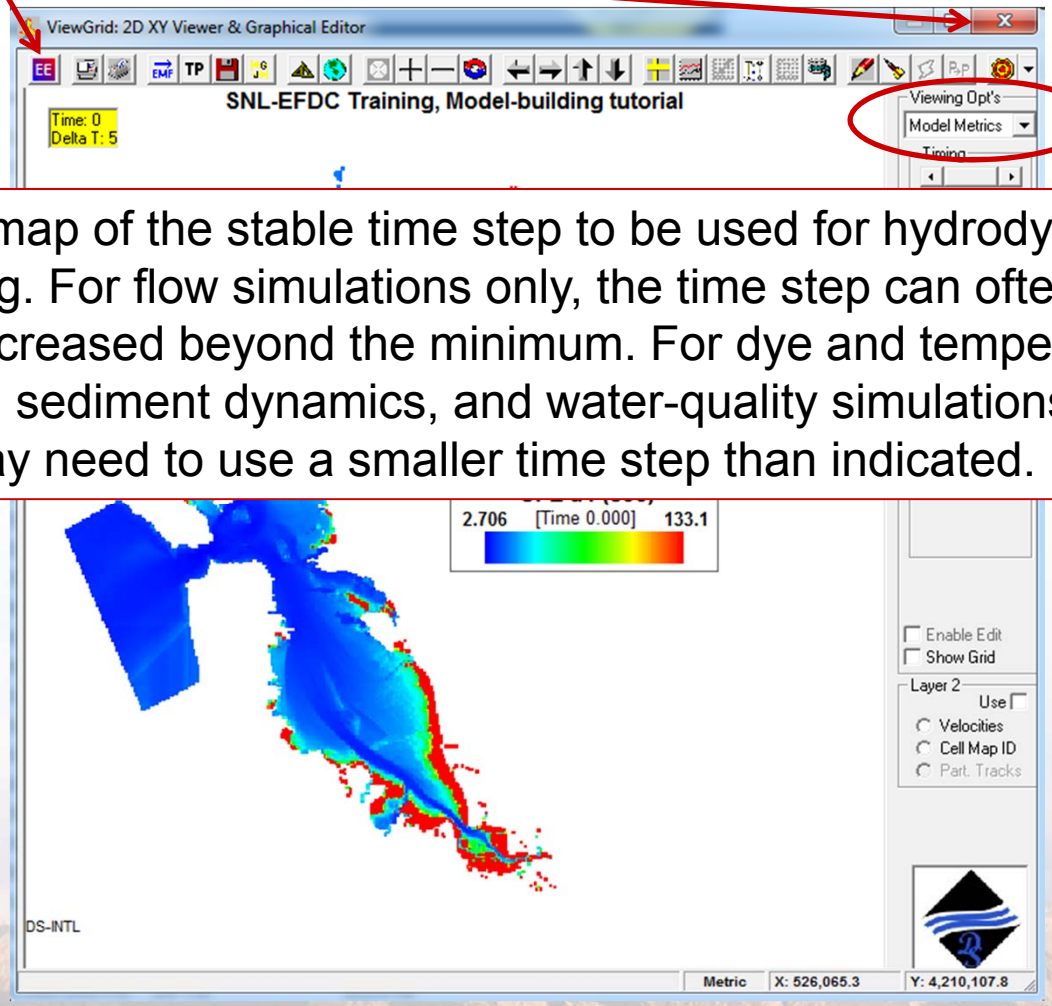
Before we "Modify" the time step, we should know what value to use.



Sandia National Laboratories

The Time Step

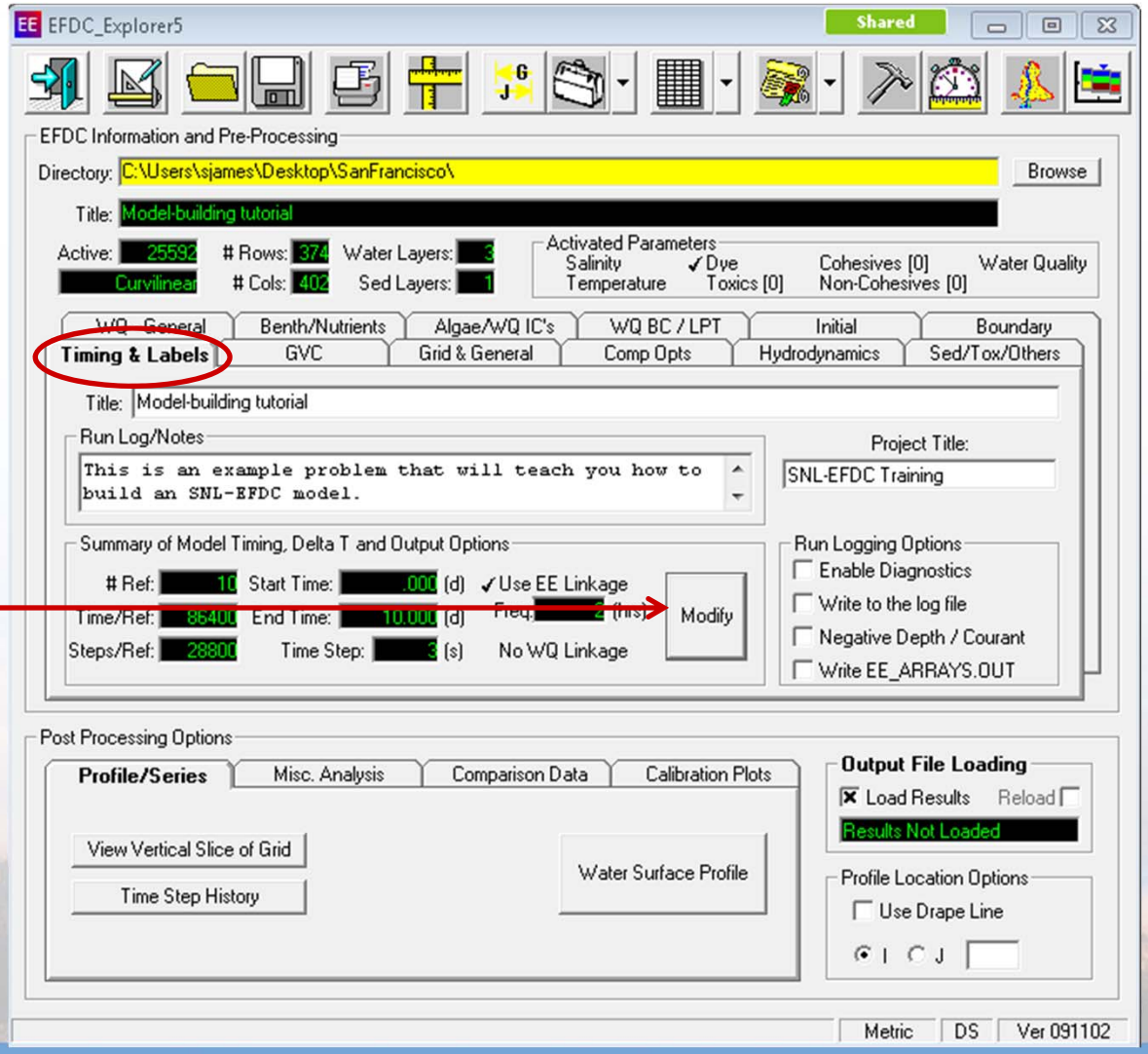
Exit out of the Grid Viewer



This is a map of the stable time step to be used for hydrodynamic modeling. For flow simulations only, the time step can often be slightly increased beyond the minimum. For dye and temperature transport, sediment dynamics, and water-quality simulations, you may need to use a smaller time step than indicated.

Timing & Labels

Click “Modify”

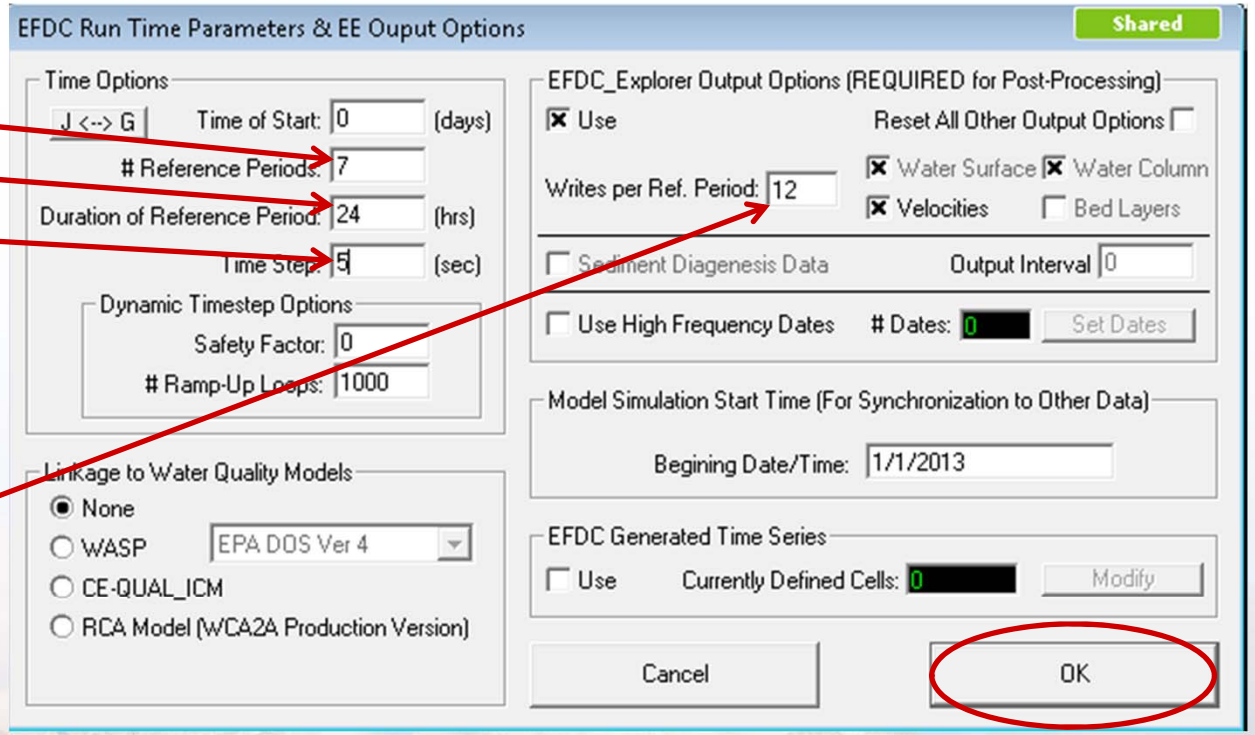


Timing & Labels

Click “Modify” on the “Timing & Labels” tab

Let’s simulate a week of 1-day reference periods using a 5-sec time step

Select the number of writes per reference period. Careful! This could generate large output files.



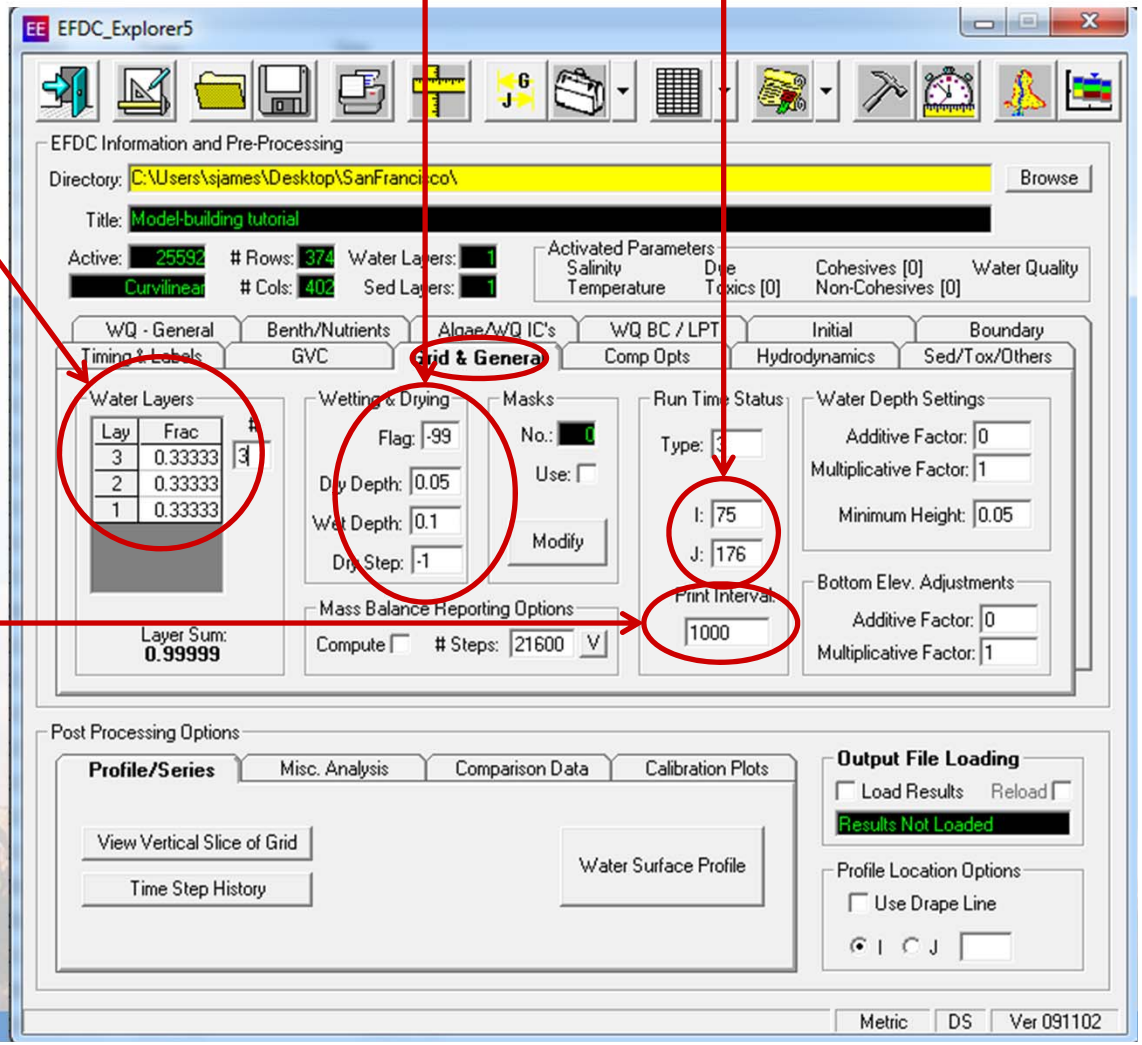
Grid & General

Allow “Wetting & Drying”

Screen display

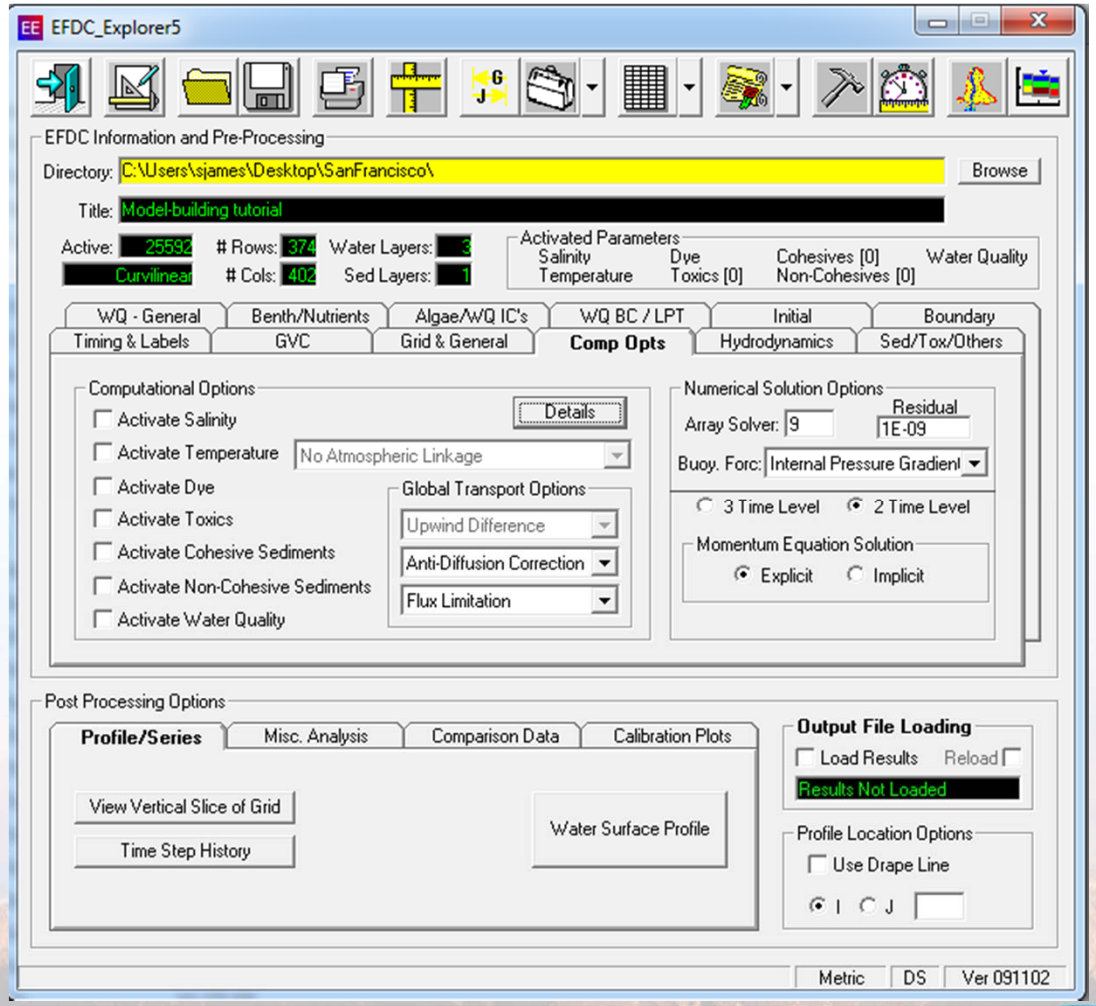
Select the number of layers. Note: Computation time scales linearly with this number.

Careful with this value. While it will cause numbers to stream on the screen faster, it will also make a HUGE output file called “tecplot2d.dat”



Comp Opts

Activate Dye

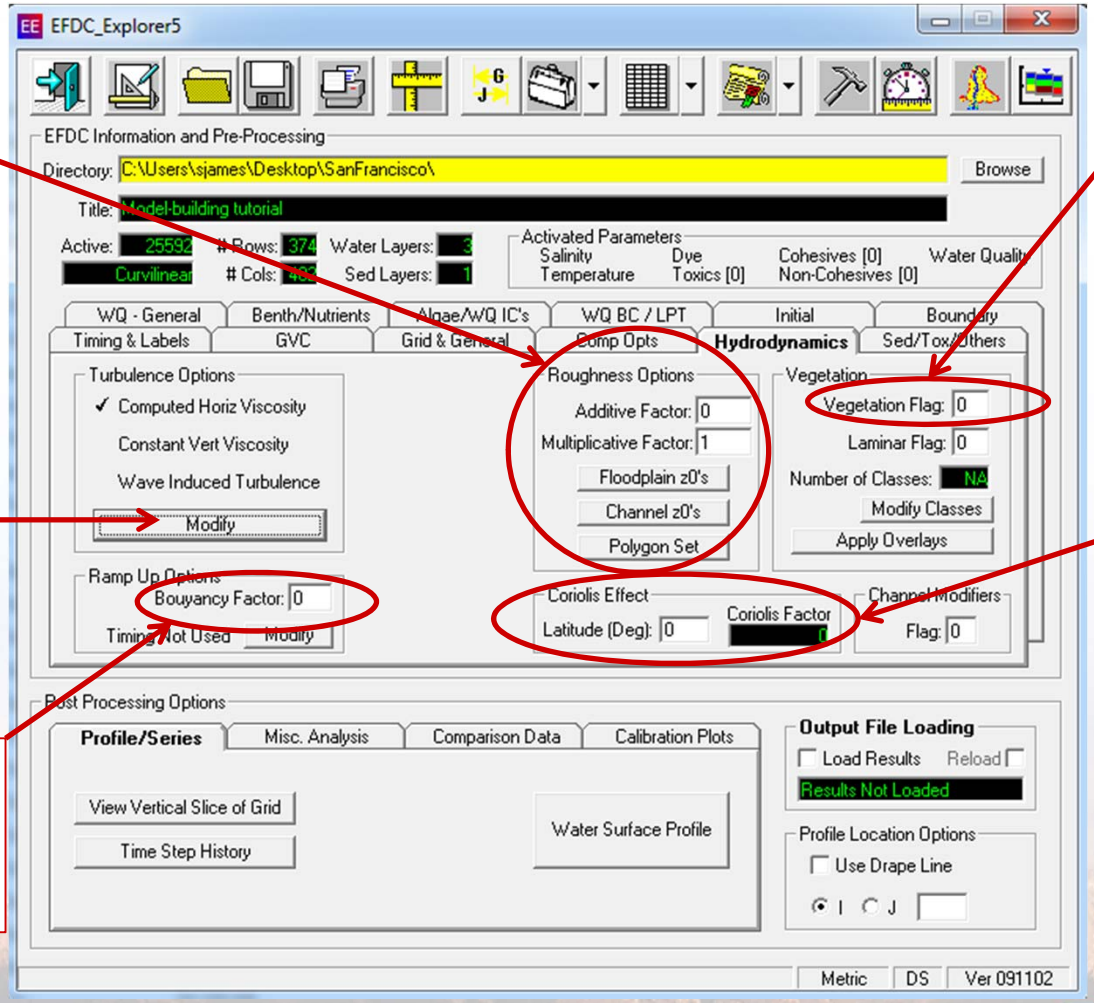


Hydrodynamics

Assign bottom roughnesses here (calibration parameter)

Important parameters!!! Click here!

Buoyant forces are off (*T* and *S*). Set to 1 to turn on.



We will turn on MHKs here (later)

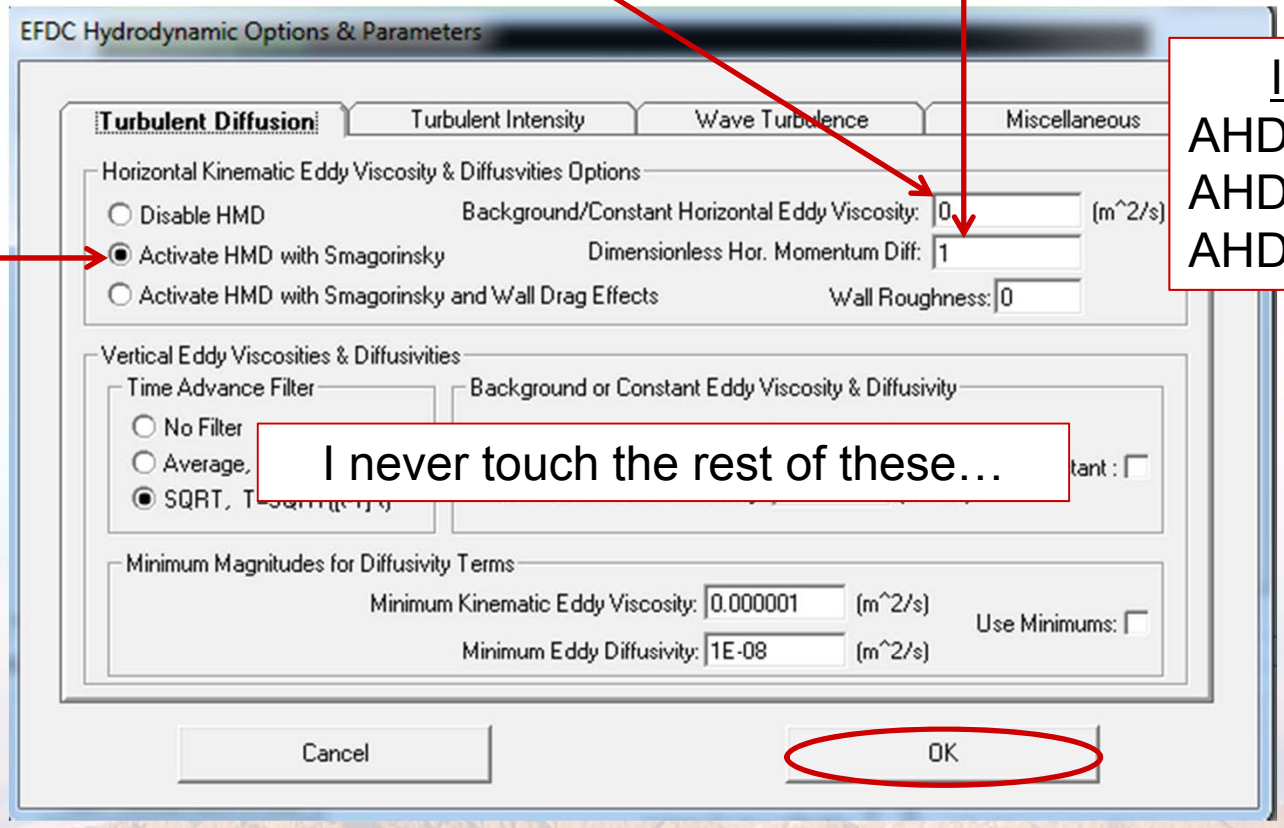
Coriolis forces are off

Modify the Turbulence Options

AHO should be between 0 and 0.25

AHD (calibration parameter)

Activate Horizontal Momentum Diffusion



I never touch the rest of these…

In General

AHD > 0.1

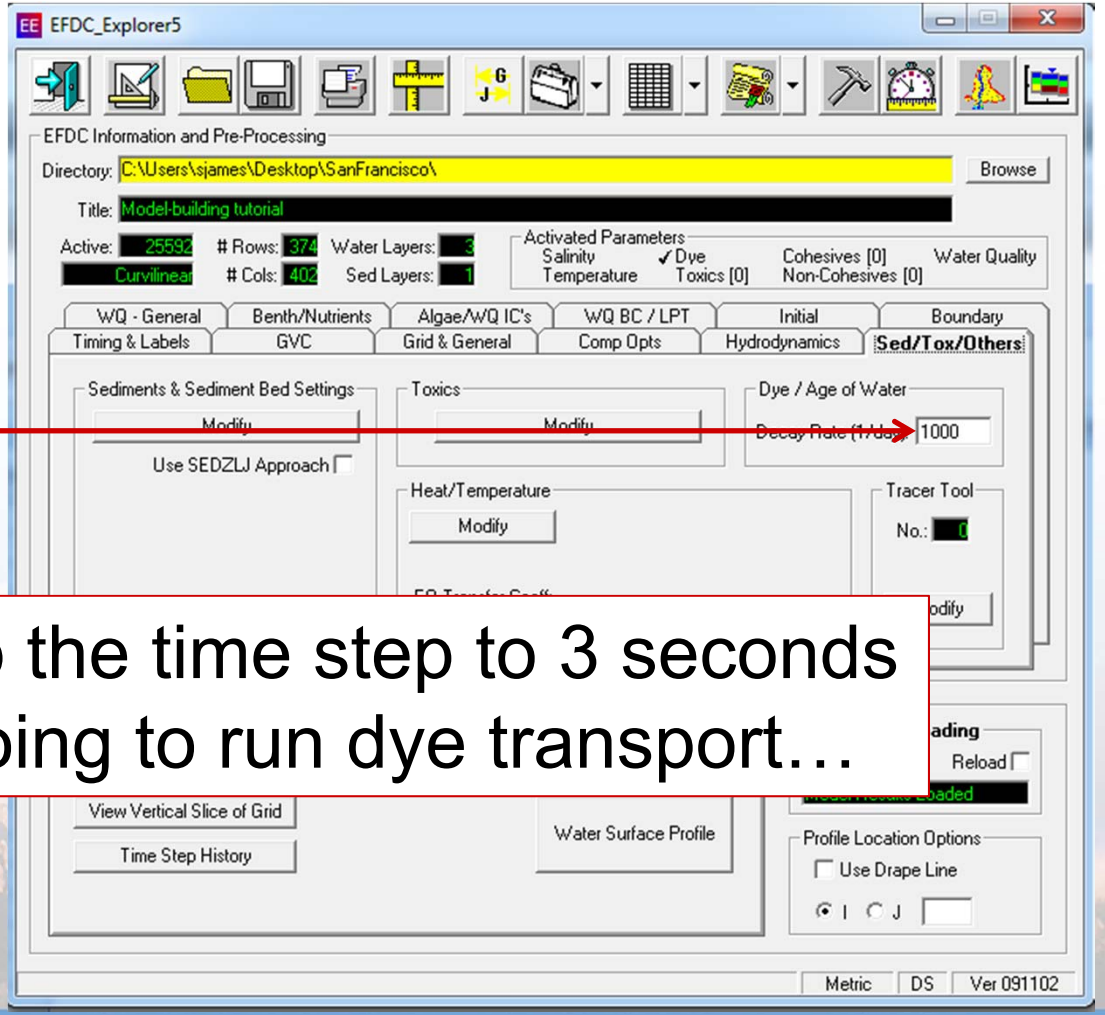
AHD = $\Delta x \Delta y/1000$

AHD < 5

Should We Add Dye?

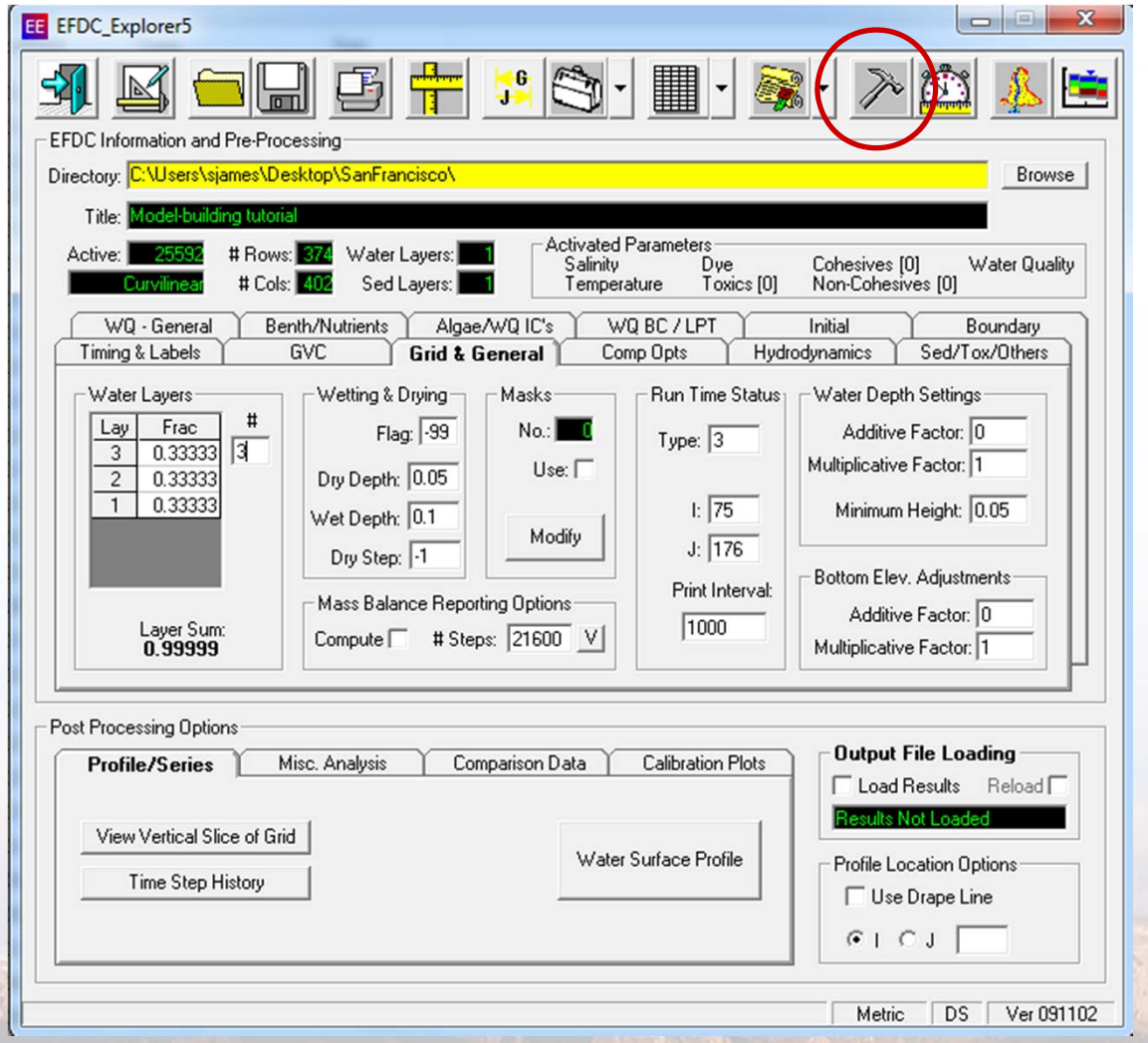
"Activate Dye" in the "Comp Opts" tab.

Activate "Age of Water" by setting the "Decay Rate (1/day)" = 1000



Careful! Drop the time step to 3 seconds if you are going to run dye transport…

Run the Model



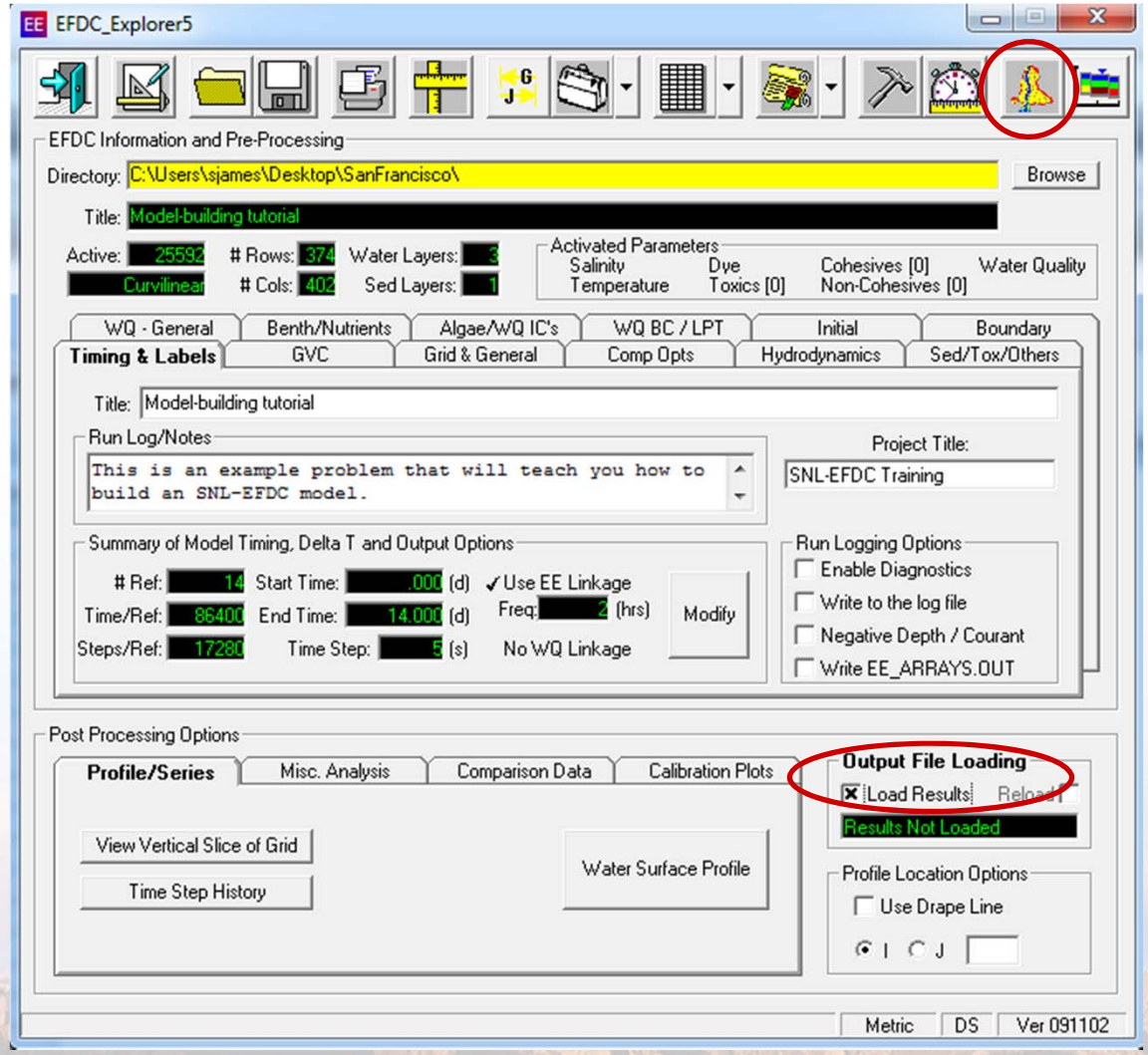
Sandia National Laboratories

Let's take lunch…

View the Results

"Load Results" from "Output File Loading"

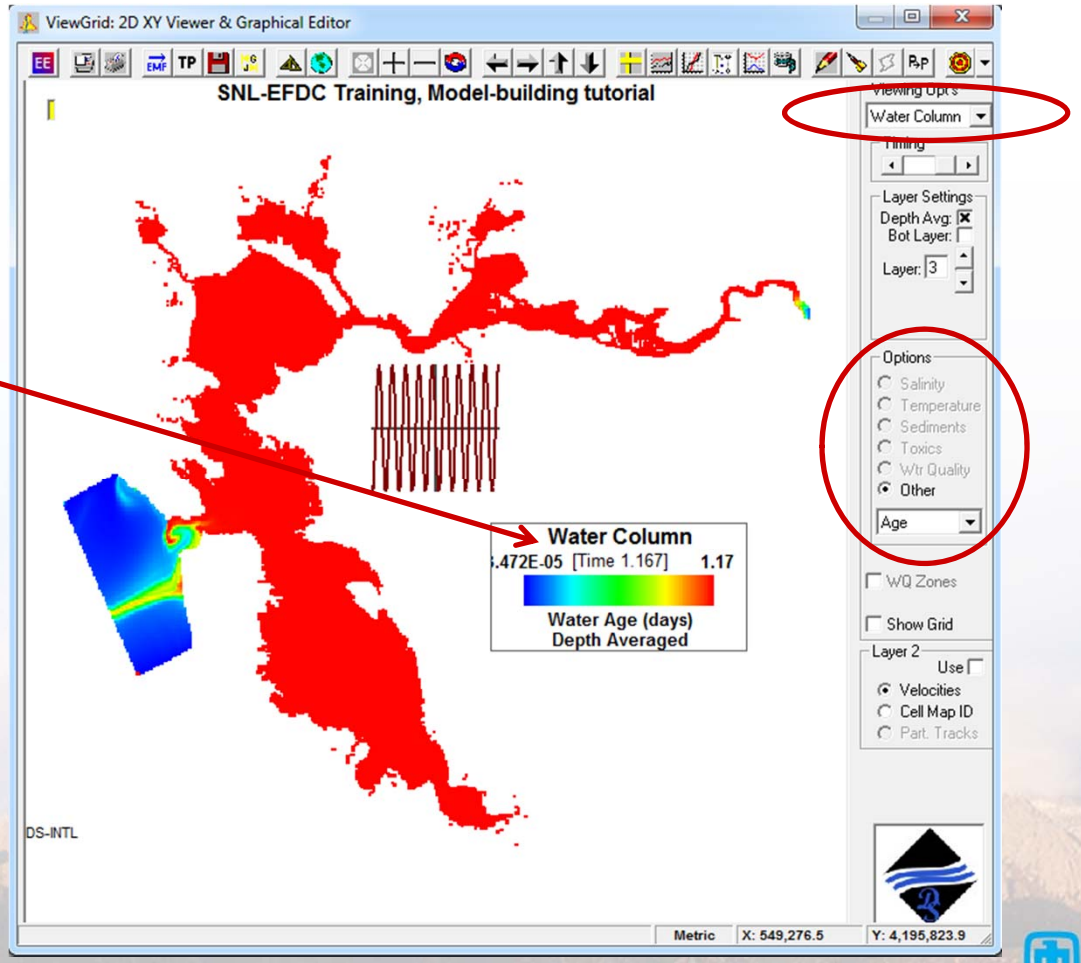
Then "View Grid"



View a Water-Age Animation

Select “Water Column” from “Viewing Opt’s”
Select “Other” and “Age” from “Options”

Right click the legend

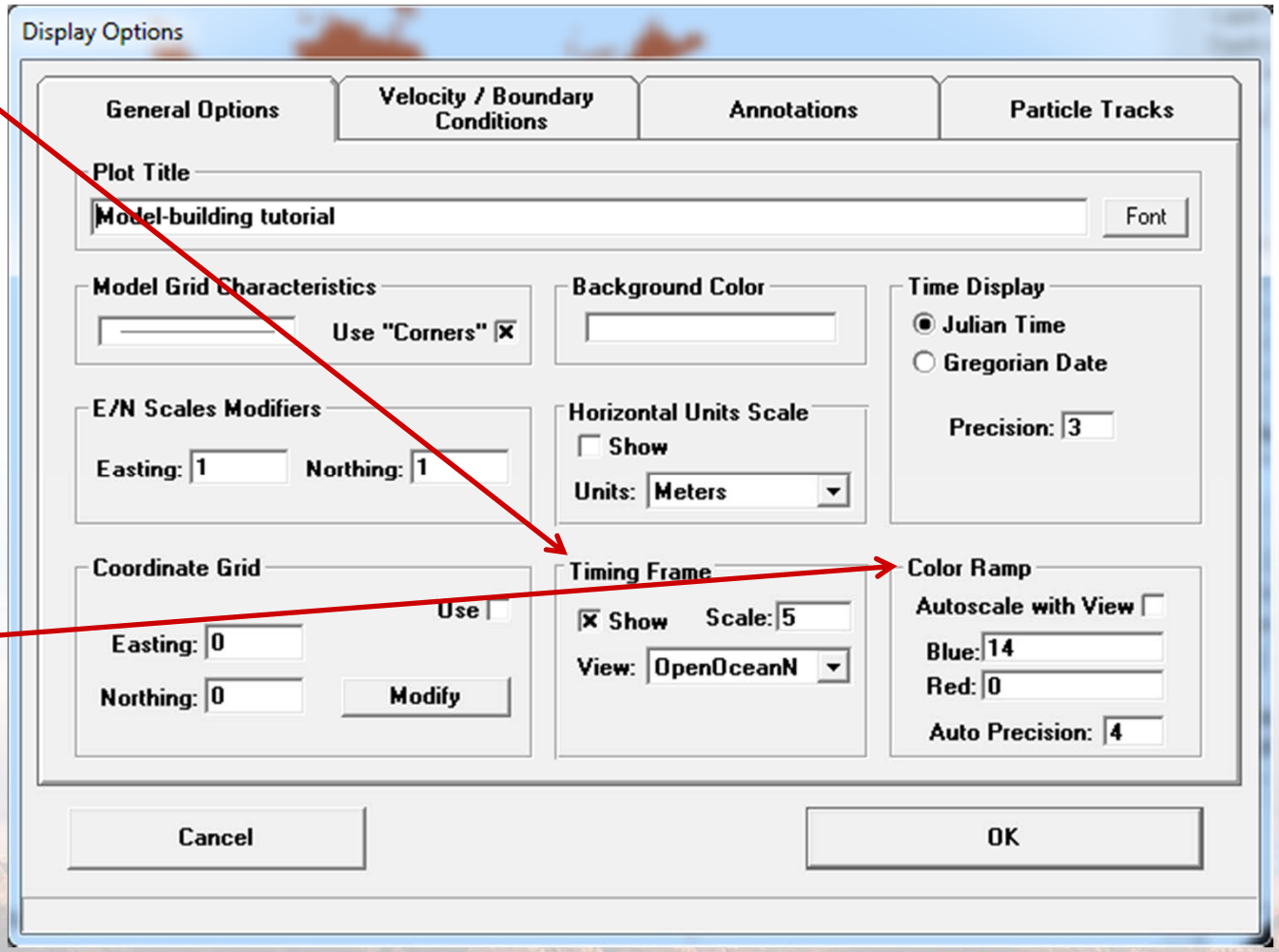


Sandia National Laboratories

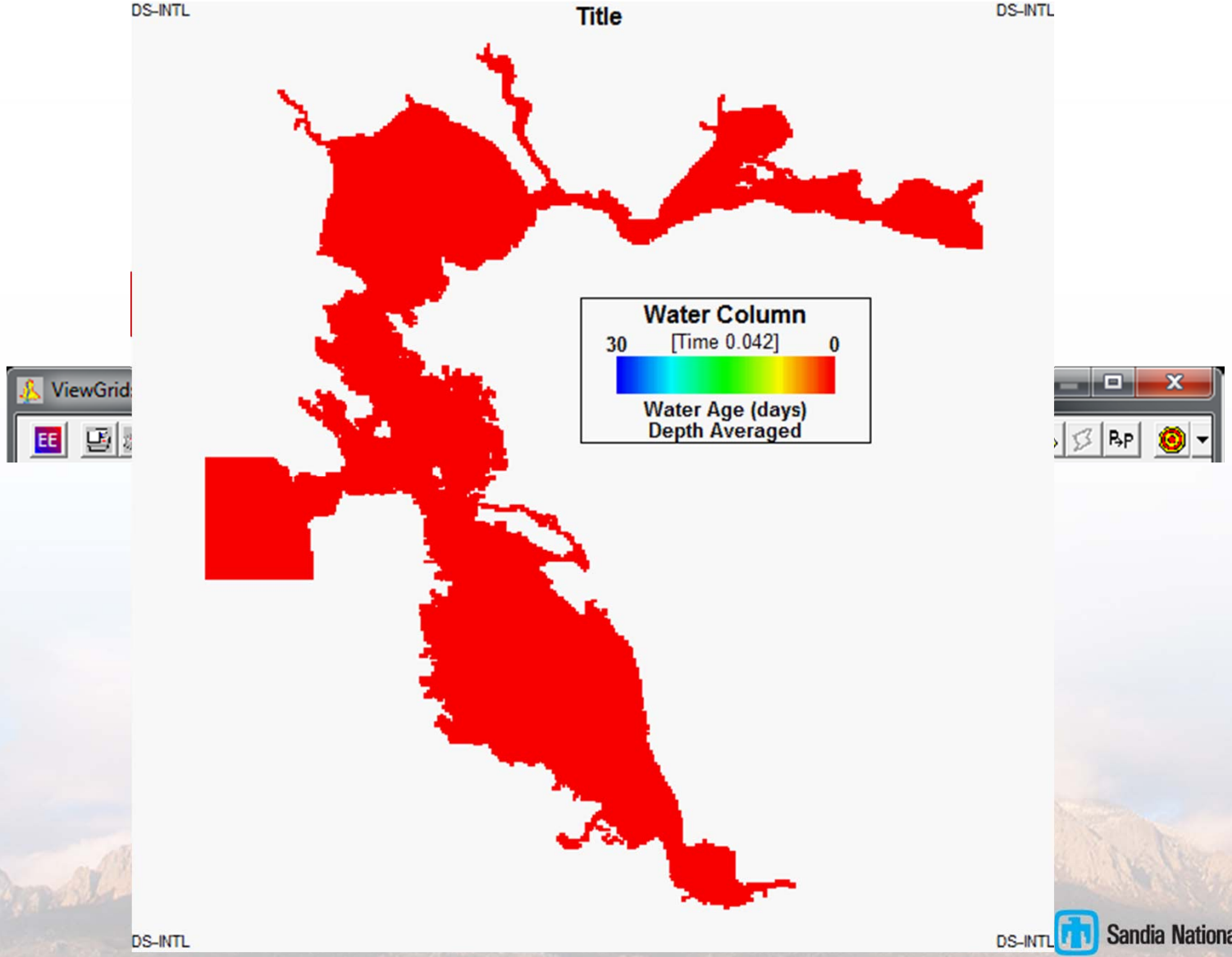
View a Water-Age Animation

Show the “Timing Frame” and “View” an ocean BC

Change the “Color Ramp” (if you like)



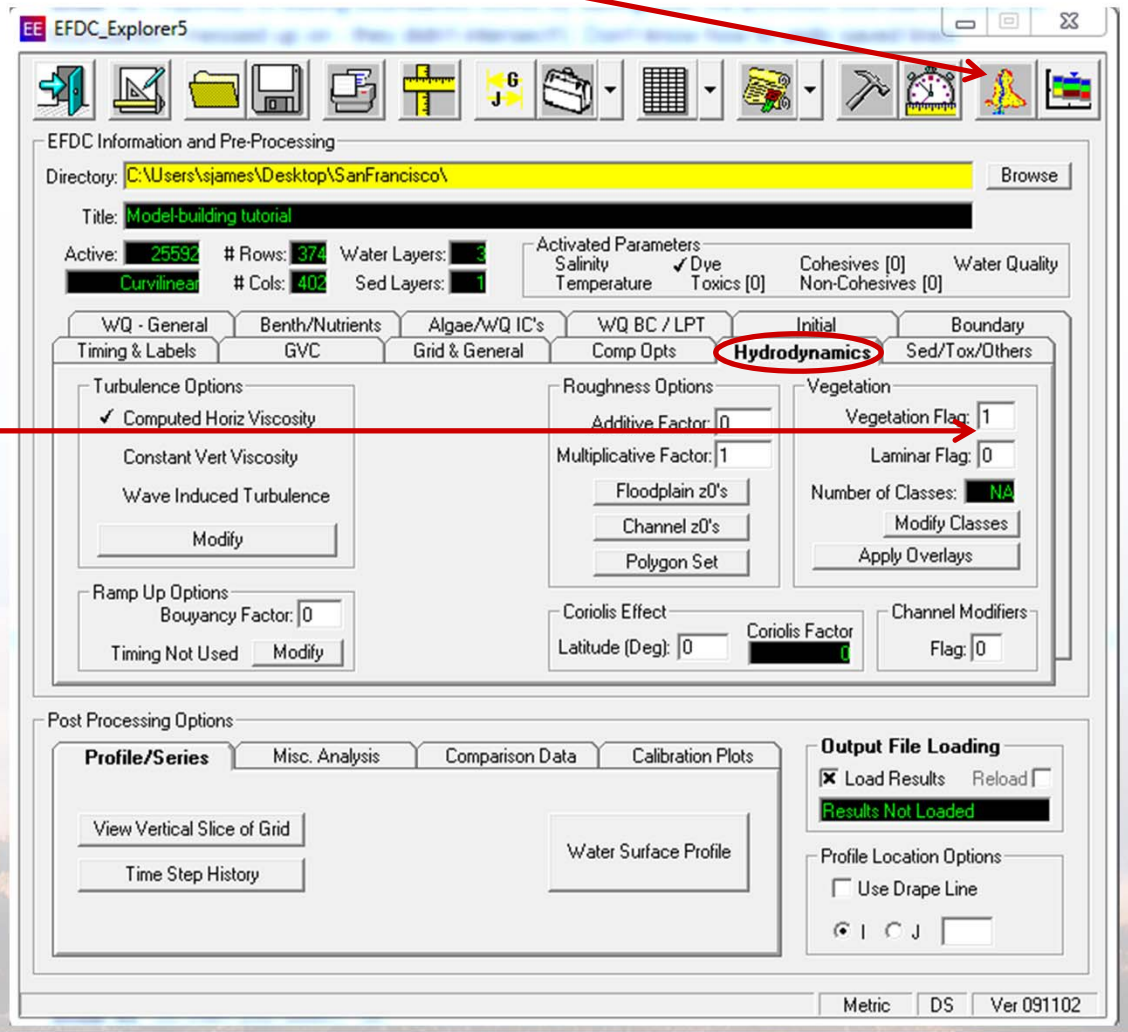
View a Water-Age Animation



The MHK Module

Let's add some devices

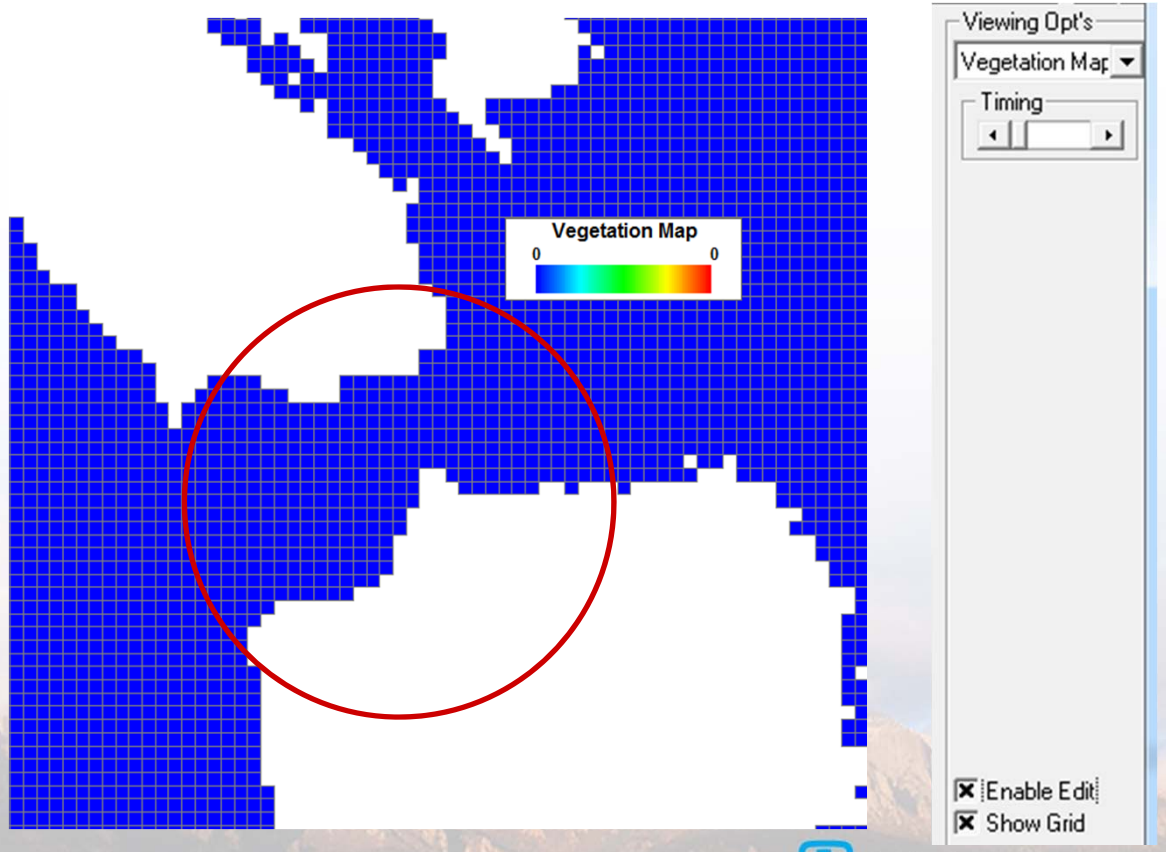
By changing the "Vegetation Flag" to 1 you can introduce vegetation AND MHK devices.



The MHK Module

Turn on the “Vegetation Flag” in the “Hydrodynamics” tab…
…and re-enter the “Grid Viewer.”
Then, select the “Vegetation Map” under “Viewing Opt’s”…
…and “Enable Edit.”

Right click some (non-boundary) cells and add some turbines under the Golden Gate Bridge.

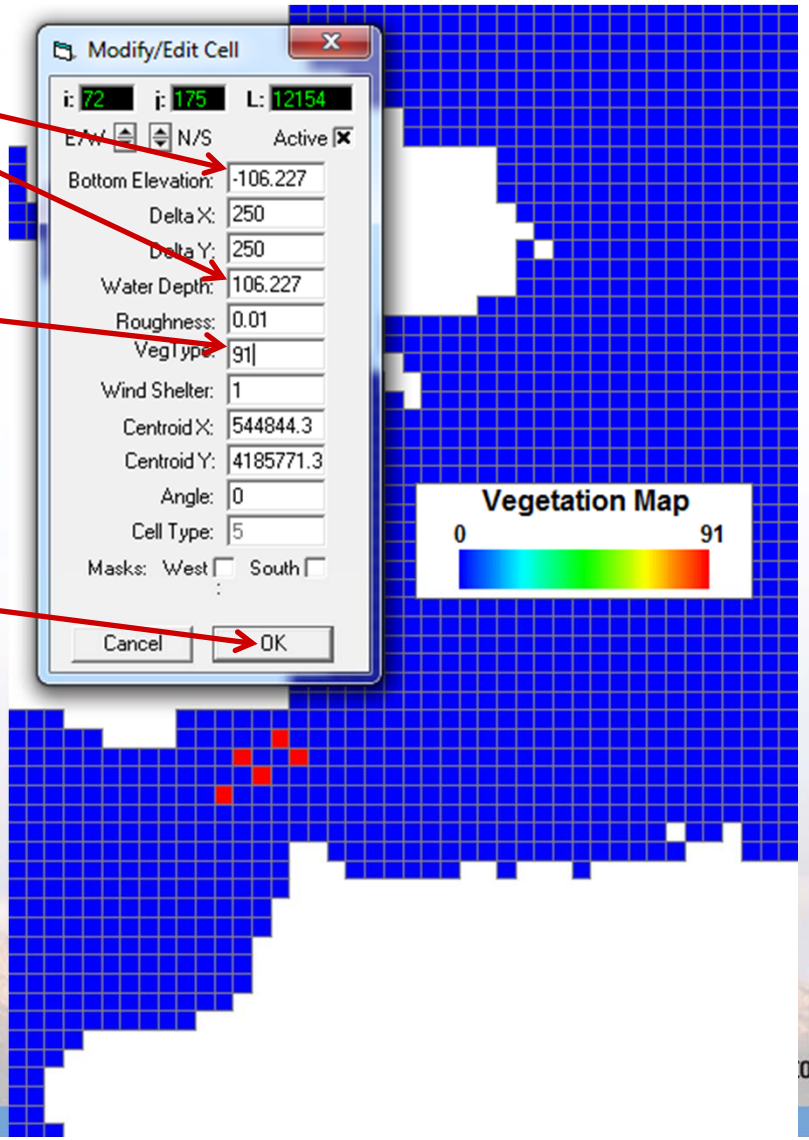


Add MHK Devices

Ensure that the water is sufficiently deep.

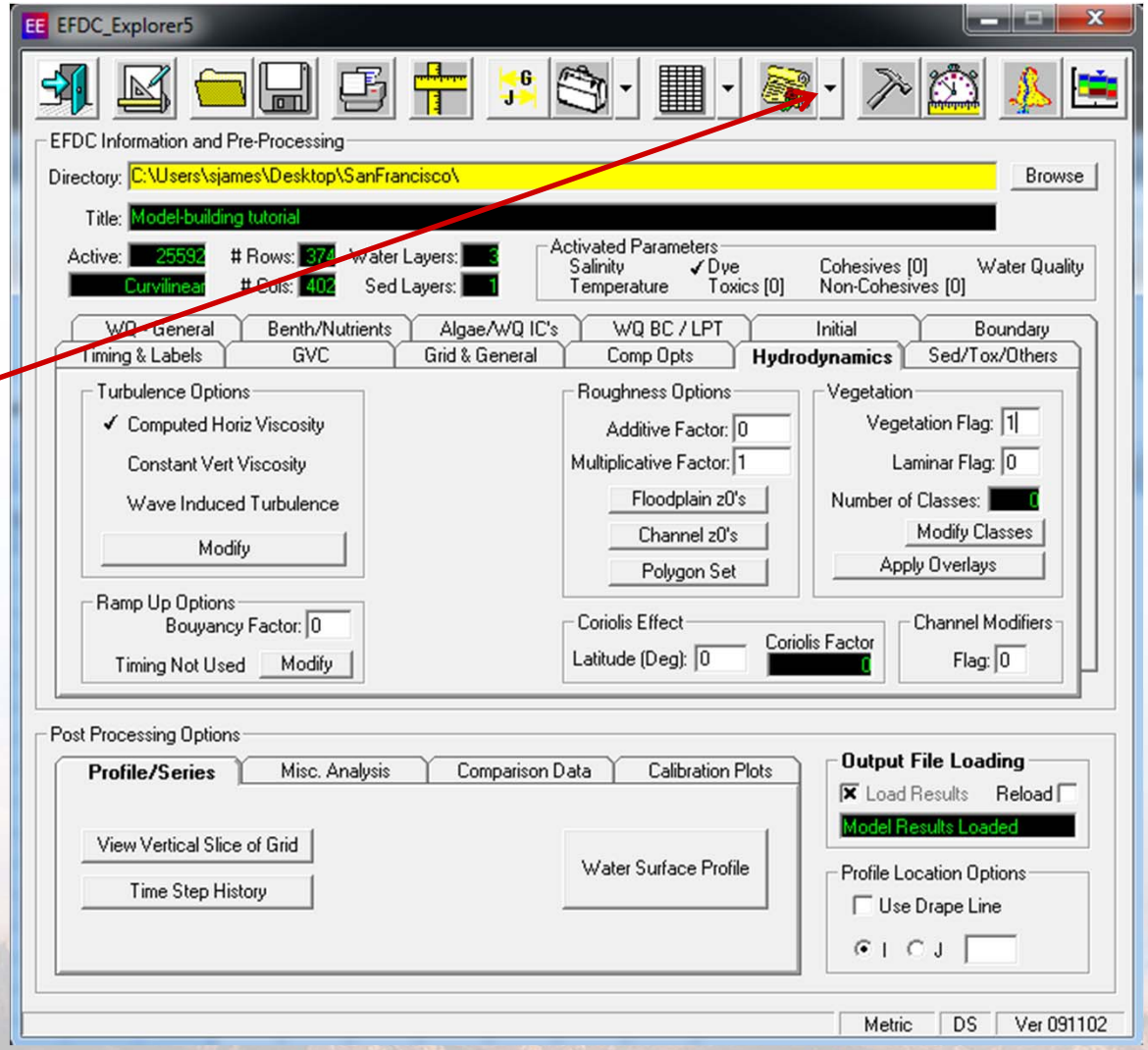
Enter a “Veg Type” of 91 (or greater)

Click “OK” and add as many as you like…



We Must Hand Edit MHK.INP

In the “Text Editor,” select “Other.INP” and open the MHK.INP file. No need to change anything unless you want to change the size/characteristics of the MHK devices.

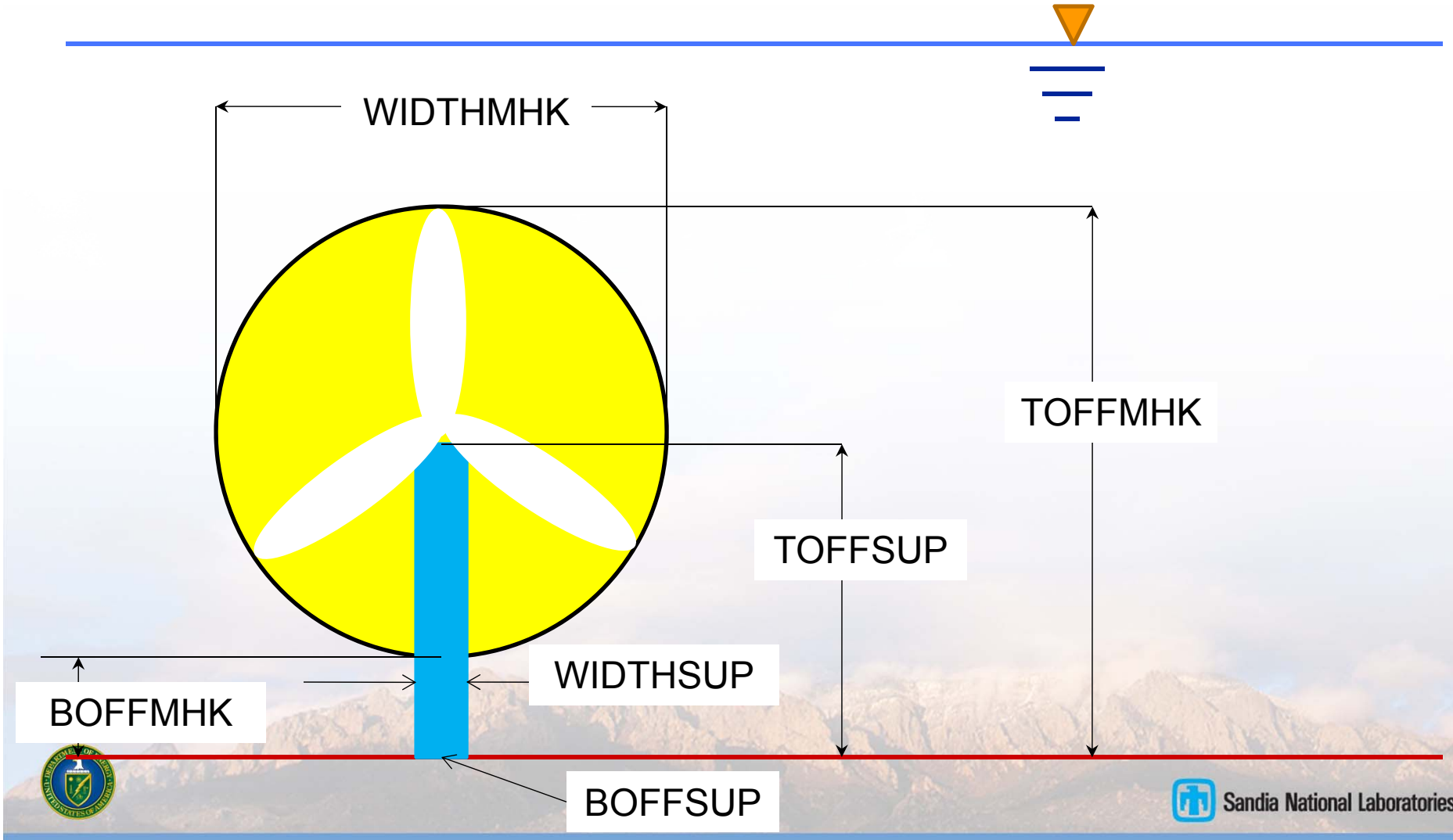


MHK.INP

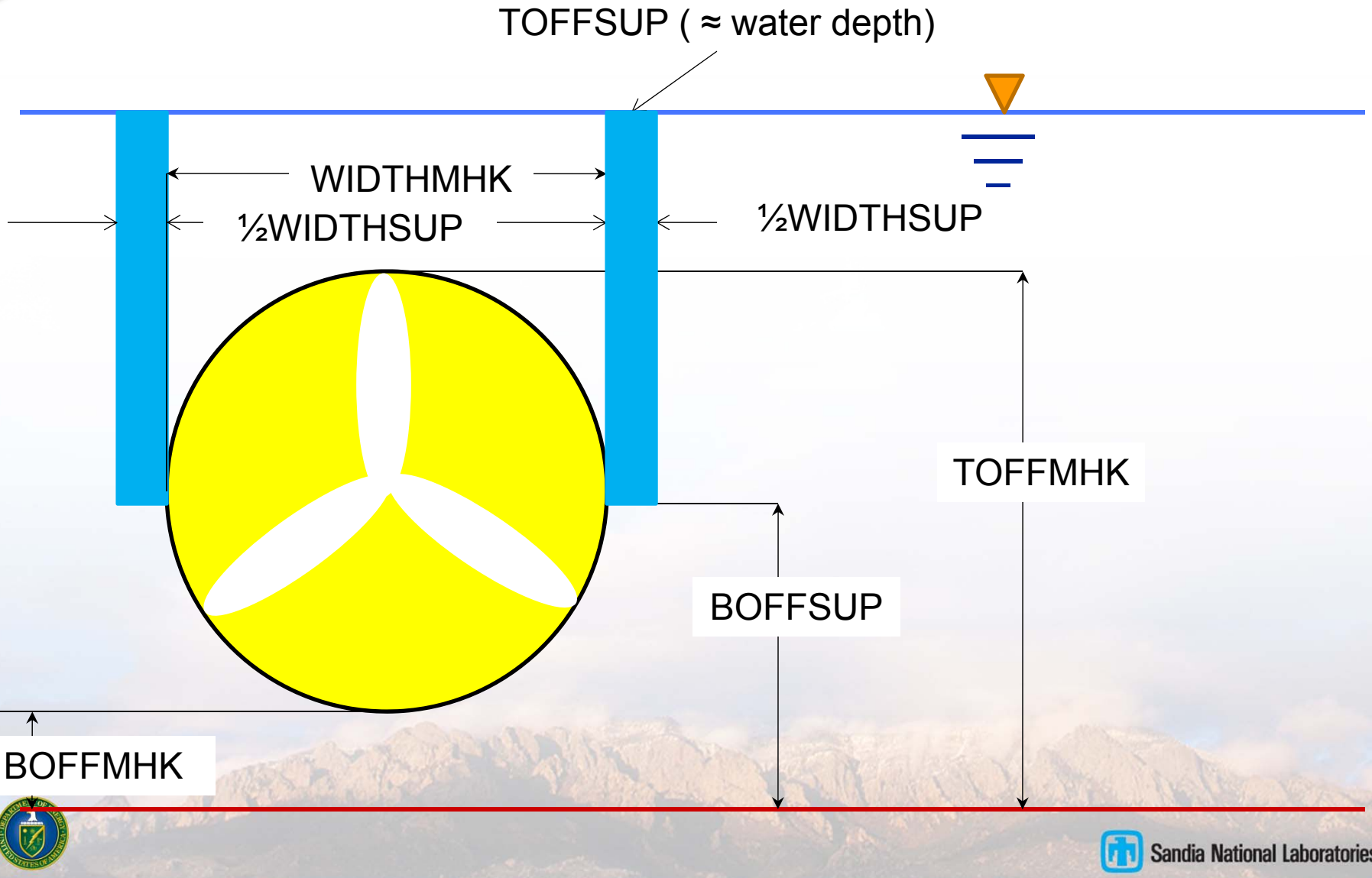
C WIDTHMHK is the width of MHK device type
C WIDTHSUP is the width of MHK support structure type
C BOFFMHK is the bottom offset of the MHK device type (how far from the bottom)
C BOFFSUP is the bottom offset of the MHK support structure type
C TOFFMHK is the top offset of the MHK device type
C TOFFSUP is the top offset of the MHK support structure type
C CTMHK is the thrust coefficient of MHK device type
C CDSUP is the coefficient of power dissipation of MHK support structure type
C VMINCUT is the minimum velocity cut-in for MHK device type power curve
C VMAXCUT is the maximum velocity cut-out for MHK device type power curve
C DENMHK is the number of MHK devices in a cell

Look at the values I entered for you.
Change them if you like. $C_T < 1$; $C_D = 1.2$ (for cylinders)
Don't put too many devices in a cell DENMHK ≤ 5.

Bottom-Mounted MHK



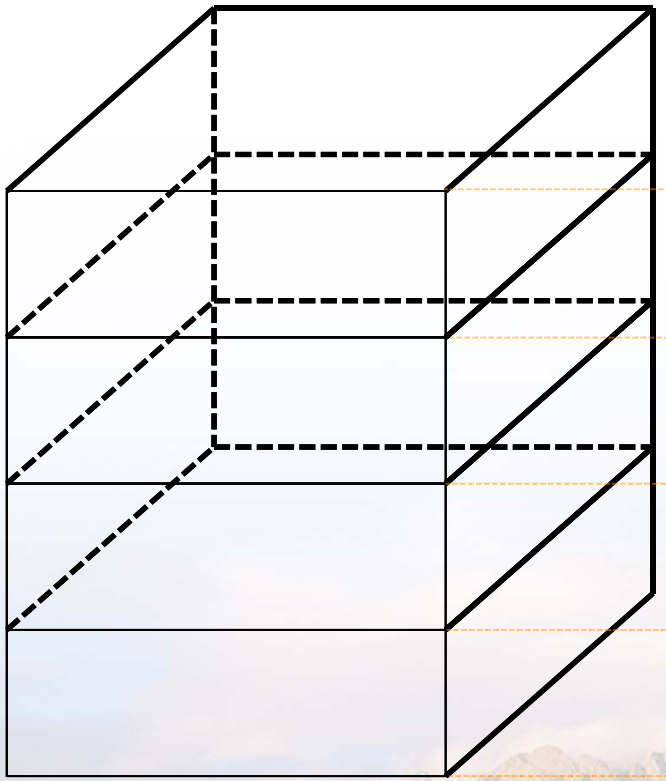
Top-Mounted MHK



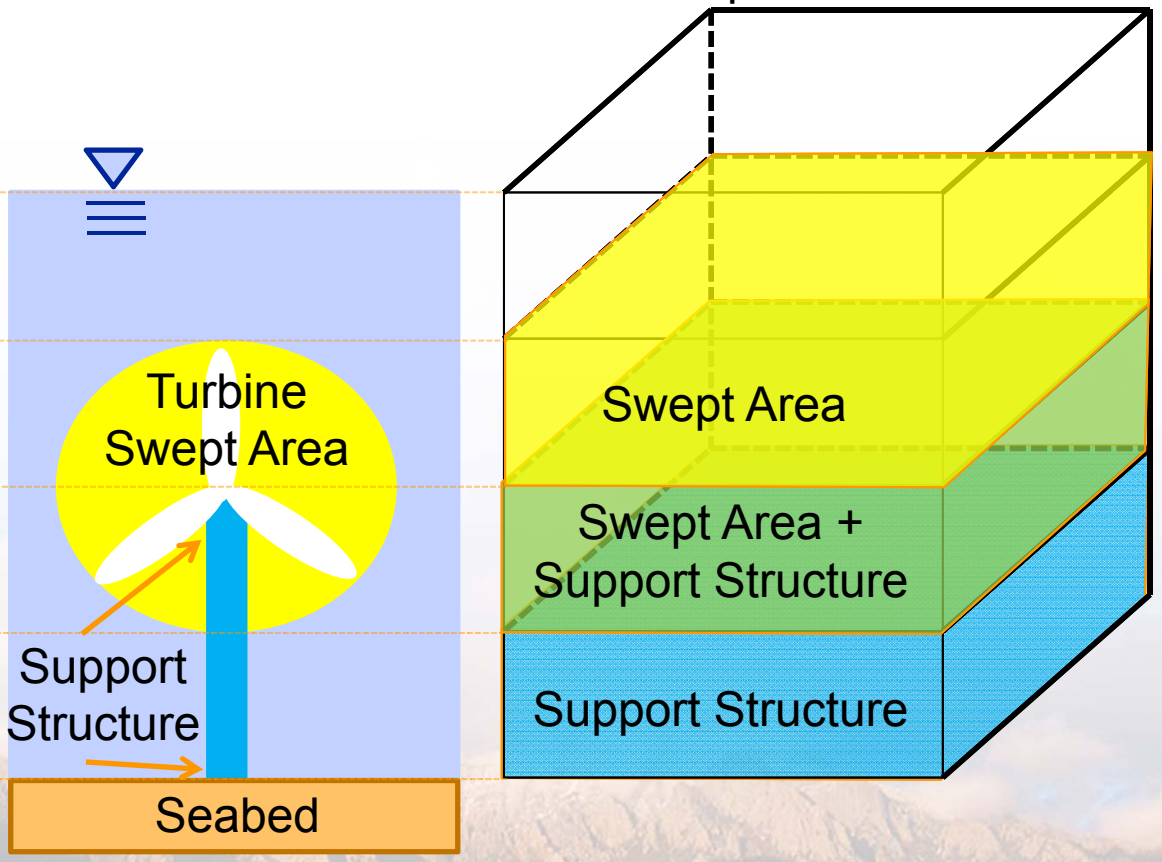
Porous Approximation

- MHK can be represented with multiple cells or contained within a single cell
- MHK can occupy an entire cell or a fraction of a cell

Hydrodynamic Model Grid



Hydrodynamic Model Grid
+
SNL-EFDC Turbine Representation



Save the Model Run it Later…

Sandia National Laboratories