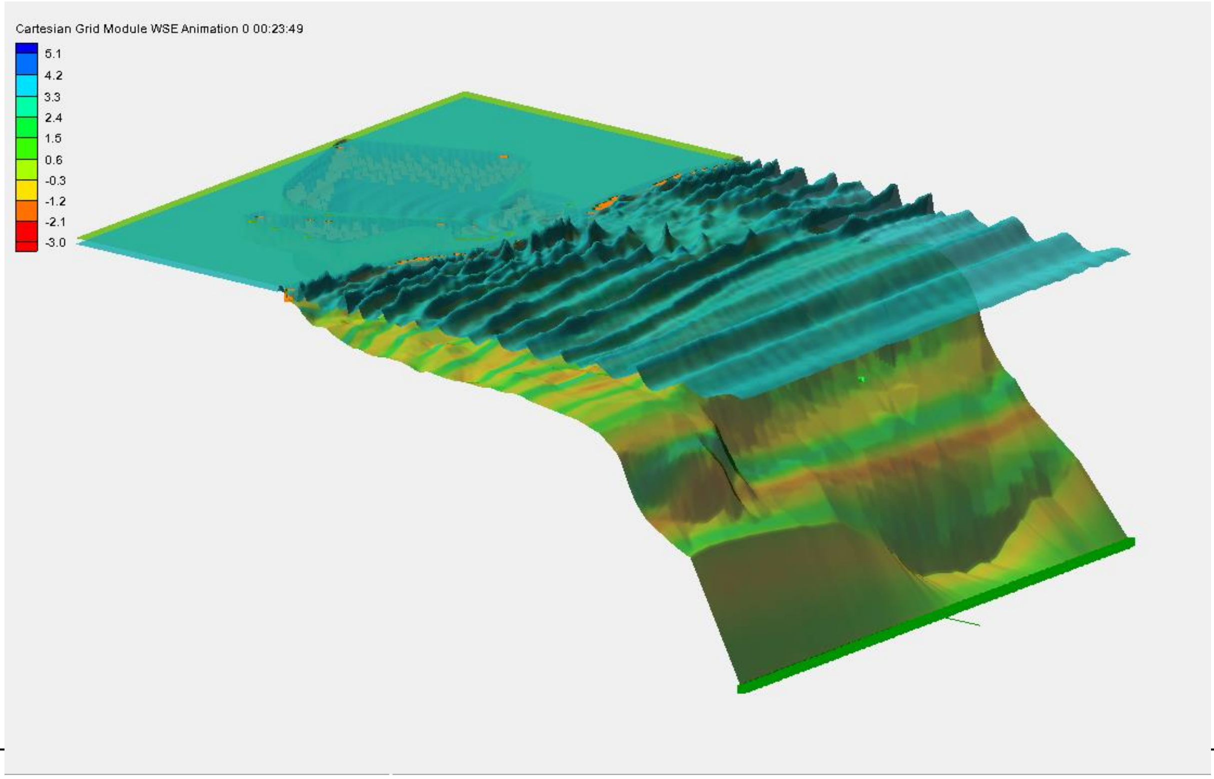
SMS

v. 12.1

SMS 12.1 Tutorial

BOUSS-2D



Objectives

This lesson teaches how to use the interface for BOUSS-2D and run the model for a sample application. This example steps through the process of setting up and running a simulation. Data from Barbers Point, Hawaii will be used. Data files for the tutorial are in the *tutorials\Models\BOUSS2D\data files* directory.

Prerequisites

- Overview Tutorial

Requirements

- Map Module
- Cartesian Grid Module
- Scatter Module
- BOUSS-2D

Time

- 60-90 minutes

AQUAVEO™

1 Introduction

As a phase-resolving nonlinear wave model, BOUSS-2D can be used in the modeling of various wave phenomena including shoaling, refraction, diffraction, full/partial reflection and transmission, bottom friction, nonlinear wave-wave interactions, wave breaking and dissipation, wave run up and overtopping of structures, wave-current interaction, and wave-induced currents. The data used for this tutorial includes images, bathymetry data, and coastline data for the south west corner of the island of Oahu.

Open the background data for this project by doing the following:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Select the file “topo1.jpg” from the *data files* folder for this tutorial and click **Open**. Since the directory includes a WLD file, SMS will display the image.
3. Select *File* | **Open** to bring up the *Open* dialog again.
4. Select the file “topo2.jpg” and click **Open**.
5. Select *File* | **Open** to bring up the *Open* dialog again.
6. Select the file “bp_bathy_filtered.pts” and click **Open**. The file will start the *File Import Wizard*. This file includes depth values obtained from a local survey in the Barbers Point harbor and the immediate coastal region outside the harbor.
7. In the File Import Wizard – Step 1 of 2, select Delimited and check Space.
8. Change Start import at row to “2” and uncheck Heading row.
9. Click **Next** to go to step 2 of the *File Import Wizard*.
10. Accept the defaults in step 2 and click **Finish** to close the *File Import Wizard*.

   - If the scatter set does not show up, go into *Display* | **Display Options** to bring up the *Display Option*s dialog.
   - Select *Scatter* from the list on that left, turn on *Points* and click **OK** to close the *Display Options* dialog.

11. Select *File* | **Open** to bring up the *Open* dialog one more time.
12. Select "bp_coast.cst" and click **Open**. This file contains the coastline definition for the entire island of Oahu.

The data after reading in the bathymetric data should appear as shown in Figure 1. **Zoom** out and the data should appear as shown in Figure 2.

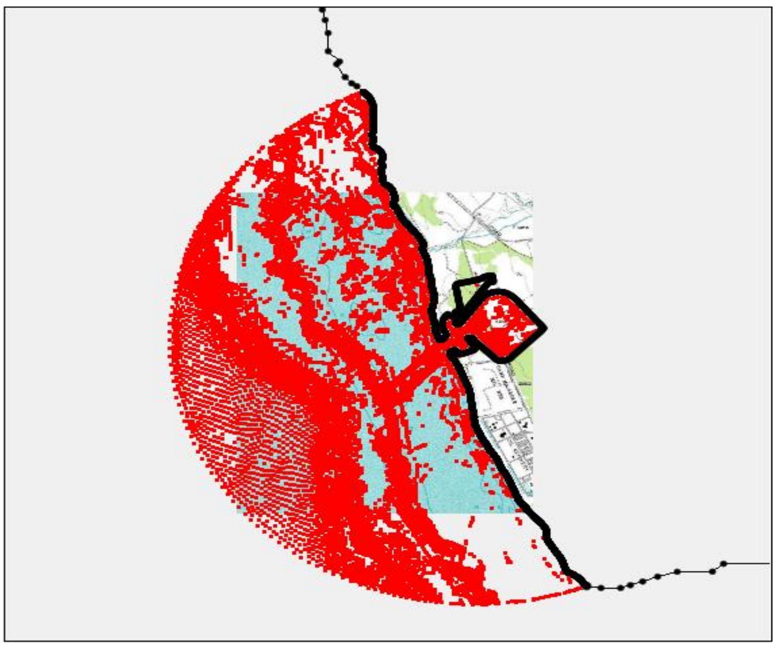


Figure 1 Bathymetry and images for project area

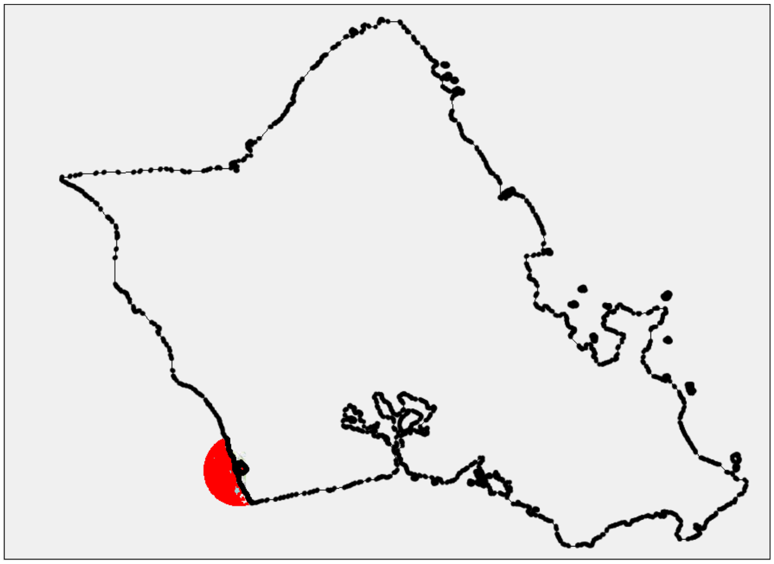


Figure 2 Coastline of Oahu with bathymetric data

2 Specifying Model Units

The images came from http://terraserver.homeadvisor.msn.com and are therefore registered to the Transverse Mercator NAD 83 coordinate frame. The bathymetry has been transformed to be relative to this coordinate frame. Reading in the images will set the project to this coordinate frame. However, the vertical units must first be changed to meters.

To do this:

1. Select *Display* | **Projection…** to bring up the *Display Projection* dialog.
2. Check that the *Global projection* option is turned on and set to the following:
   - "UTM, Zone: 4 (162 degrees West – 156 Degrees W – Northern Hemisphere), NAD83, meters."
3. Change the *Units* in the *Vertical* section to "Meters" and click **OK** to exit the *Display Projection* dialog.
4. Next, right-click on "Area Property" and select **Projection…** to open the *Object Projection* dialog.
5. Make sure the settings are the same as in step 2, and click **OK** to set the object projection.
6. Repeat steps 4–5 with the scatter data "bp_bathy_filtered".
7. Click the **Frame** macro when done.

3 Trimming the Coastline

There is currently more coastline here than is needed, including many other harbors, land features, and islands unrelated to Barbers Point harbor. Use the following steps to trim the coastline to the area involved:

1. **Zoom** into the area being modeled (the scattered data points).
2. Make sure the Map Module is selected, then right-click on the "Area Property" and select *Type* | *Models* | *BOUSS-2D* | **BOUSS-2D**.
3. Select the **Create Feature Arc** tool, then click on the coastline just north of the data (P1 in Figure 3), then click inland (P2) and then on the coastline east of the simulation area (P3).

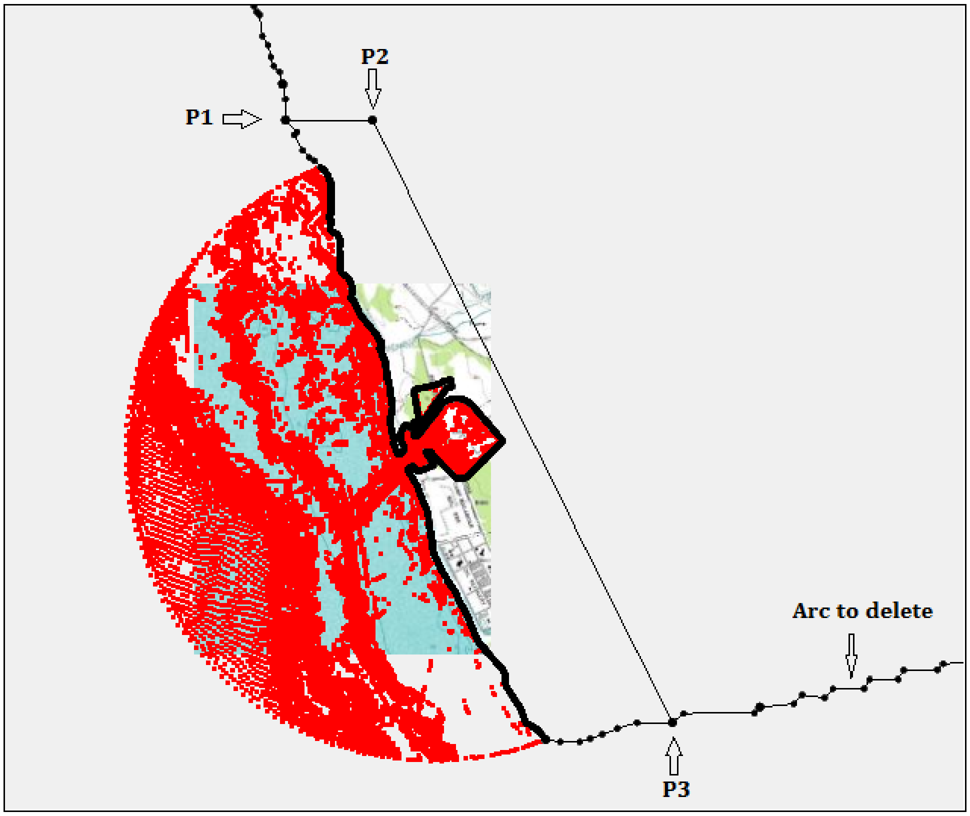


Figure 3 Arc bisecting area around simulation from island

4. Switch to the **Select Feature Arc** tool, select the coastline away from area of interest and hit the *Delete* key to eliminate this arc (marked "Arc to delete" in Figure 3).
5. Click **Yes** in the confirmation dialog.

6. **Frame** the display and drag a box around the island arcs and *Delete* them. (This is easily done by dragging a box around the islands on the north side of Oahu and deleting them, and then dragging a box around the islands on the east side.)
7. **Frame** the display again when done.
8. Build a polygon to represent the land around Barbers Point by selecting *Feature Objects* | **Build Polygons**.
9. Select the **Select Polygon** tool, and select the land polygon.
10. Choose *Feature Objects* | **Select/Delete Data…** This brings up the *Select/Delete Data* dialog shown in Figure 4.

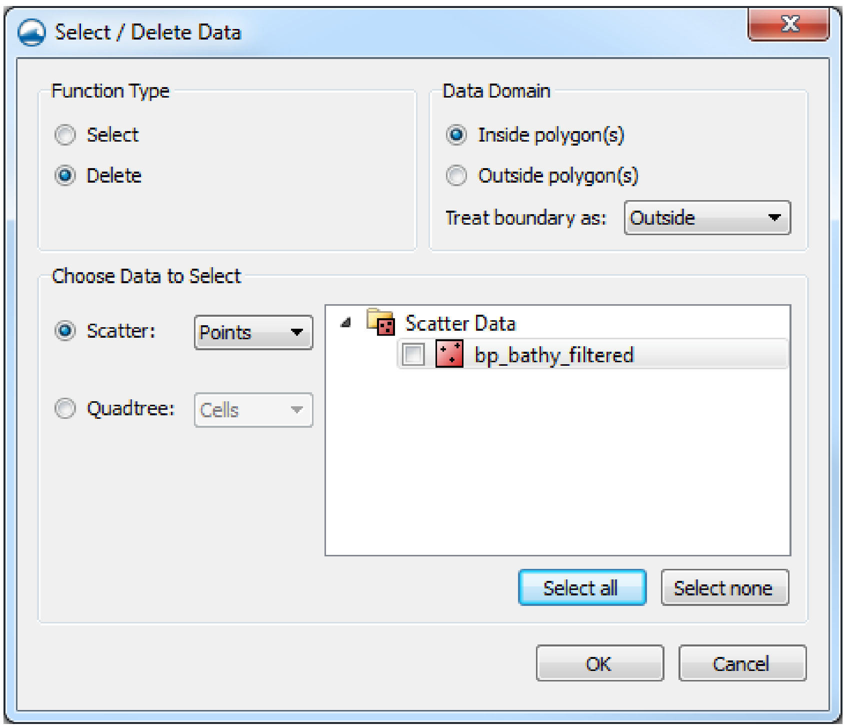


Figure 4 Trim/Select Data Options

11. Set the options to Delete and Inside the polygon(s).
12. Make sure "Triangles" is selected under the *Scatter* option. This will delete triangles inside the polygon.
13. Make sure to check on the "bp_bathy_filtered" under *Scatter Data*.
14. Click **OK** to close the *Select/Delete Data* dialog.

The surface now represents the seabed around the region of Barbers Point Harbor in Hawaii. The next step is to create a computational grid for BOUSS-2D.

4 Creating the Grid

The computational domain of BOUSS-2D is a Cartesian grid that can be defined with three mouse clicks. To ensure consistency, create the grid by following these steps:

1. **Zoom** into the harbor area as shown in Figure 5.

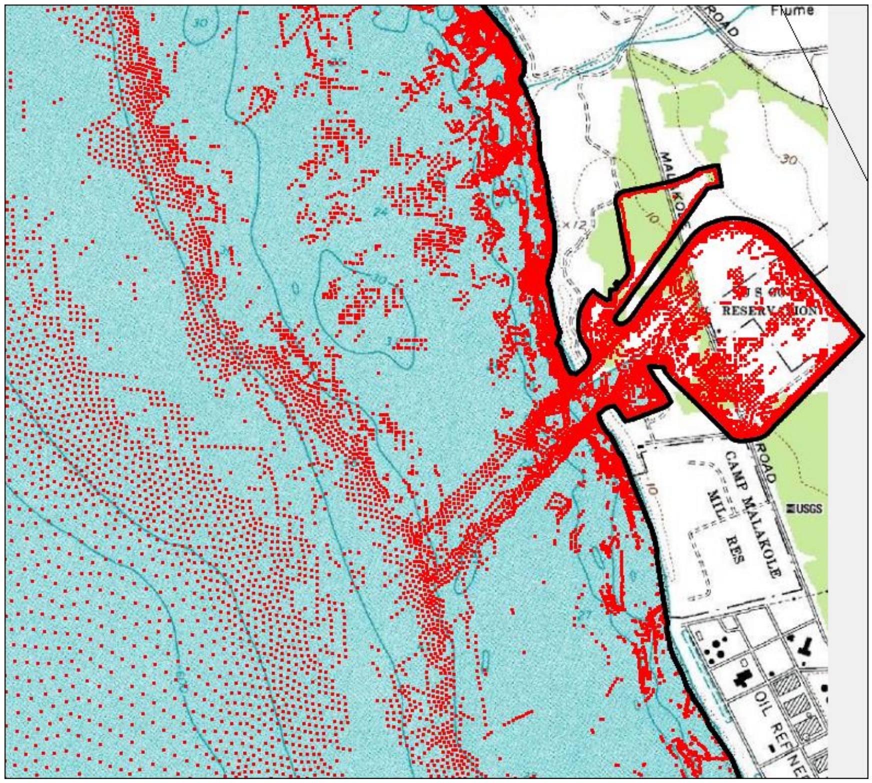


Figure 5 Zoomed view of harbor area

2. Click on “Area Property” to make it active.
3. Using the **Create 2-D Grid Frame** tool, click a grid approximately as shown in Figure 6. The grid does not have to be exactly the same since it will be modified in the *Map → 2D Grid* dialog.

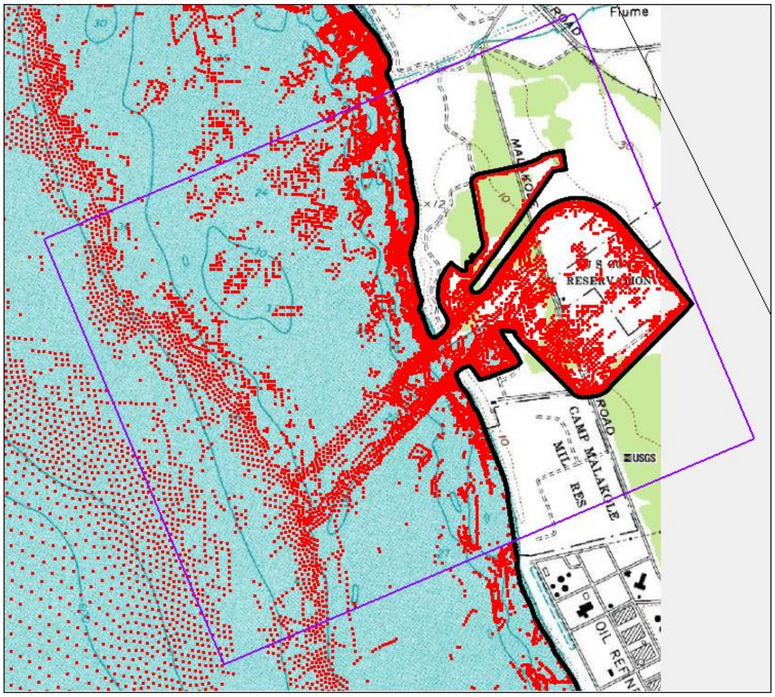


Figure 6 Grid frame

4. Right-click on "Area Property" and select *Convert* | **Map → 2D Grid**. The *Map → 2D Grid* dialog will pop up as seen in Figure 7.
5. Make the following changes:
   - Origin X = "590100"
   - Origin Y = "2356750"
   - Angle = "23.0"
   - I size = "2500"
   - J size = "2000"
6. An appropriate cell size depends on the wavelength of the waves being modeled. Click on the **Grid Helps** button to open the *BOUSS-2D Map → 2D Grid Helps* dialog to get help with determining a cell size.
7. Turn on the *T (wave period)* option. Enter "15.0" and click **OK** to close the *BOUSS-2D Map → 2D Grid Helps* dialog.
8. The recommended cell size is 11.7 meters. Smaller cells increase the definition of the model, but also increase computation time. For tutorial purposes, enter "10" as the *Cell size* in both *I Cell Options* and *J Cell Options*.

9. In the *Elevation Options*, set *Source* to “Scatter Set”, then click the **Select…** button. This will bring up the *Interpolation* dialog.
10. Set the *Extrapolation* to “Single Value” and set *Single Value* to “1.0”. This assures that the land will be treated as land. This step would not be required if survey data included points on the shore with positive elevations.
11. Click **OK** to close the *Interpolation* dialog.
12. Click **OK** to close the *Map → 2D Grid* dialog and create the grid.

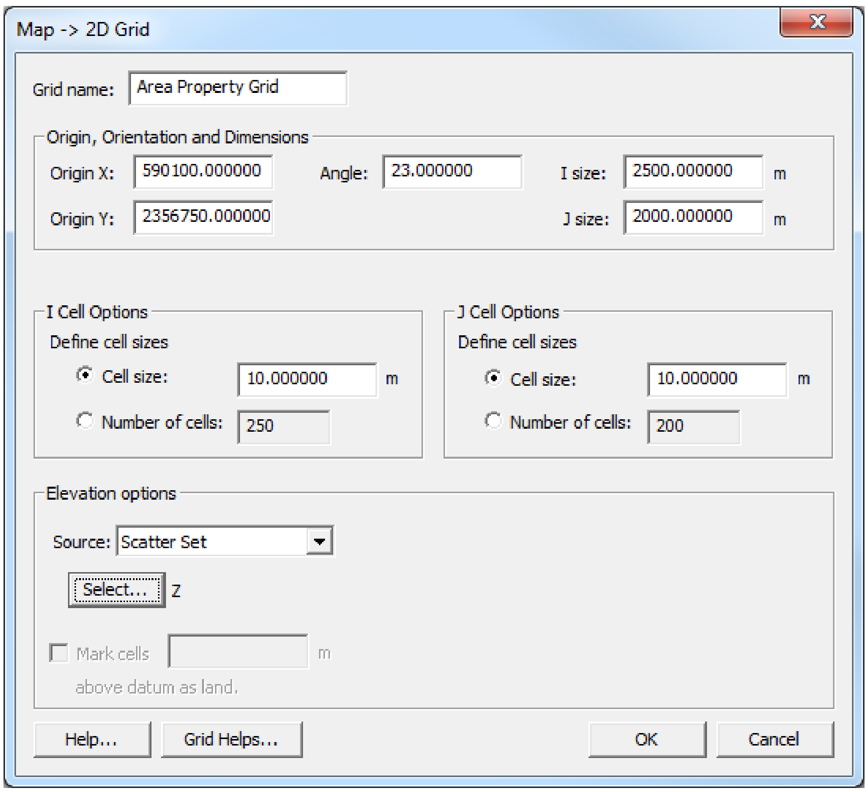


Figure 7 Parameters to create the grid

13. Right-click on the newly created grid “Area Property Grid” and select **Rename**.
14. Rename the grid “10m Grid”.

It is a good habit to change the name of grids once they are created so that basic information about them can be recognized through the name. It is now easy to see that the grid has a cell size of 10m in I and J directions.

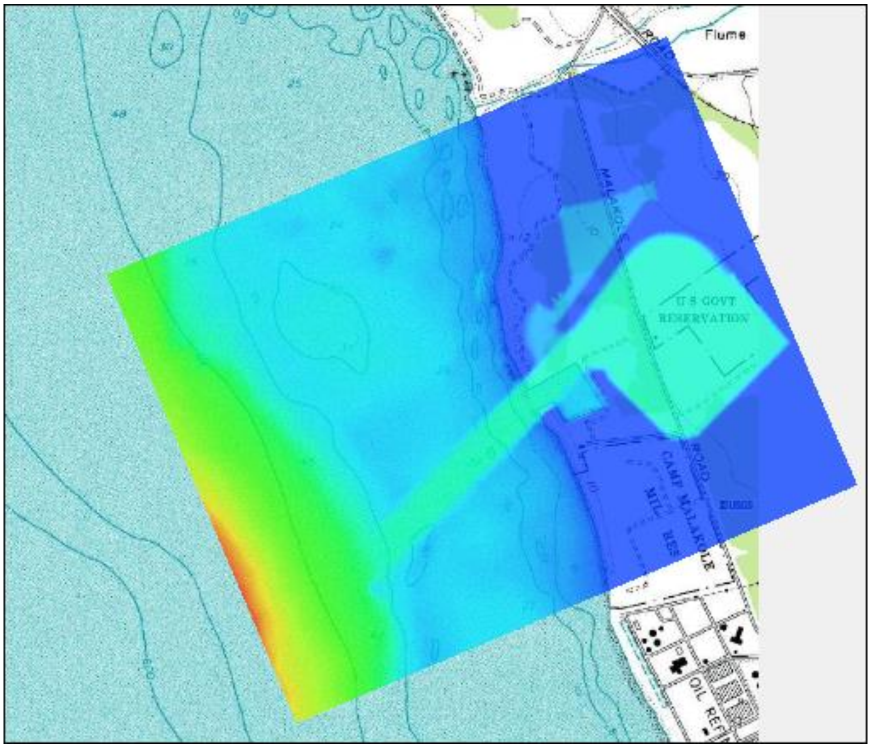


Figure 8 Resulting grid

15. Turn off Scatter Data and Map Data in the Project Explorer.
16. Select *Display* | **Display Options** to open the *Display Options* dialog.
17. Select *Cartesian Grid* from the list on the left then turn off *Cells* and turn on *Contours*.
18. On the *Contours* tab, set the *Contour method* to “Color Fill” and *Transparency* to “25”.
19. Click **OK** to close the *Display Options* dialog.

The display should appear similar to the one in Figure 8.

5 Generate a Wave Maker

Wave Makers are created through map coverages. To create a wave maker, follow these steps:

1. Right-click on the Map Data item in the Project Explorer and select **New Coverage** to open the *New Coverage* dialog.
2. Set the *Coverage type* to “Wave Maker” under the BOUSS-2D folder.

3. Set *Coverage Name* to “Wave Maker” and click **OK** to close the *New Coverage* dialog.
4. Click on the new “Wave Maker” coverage to make it active.
5. With the **Create Feature Point** tool, create a point in the new coverage as indicated in Figure 10.

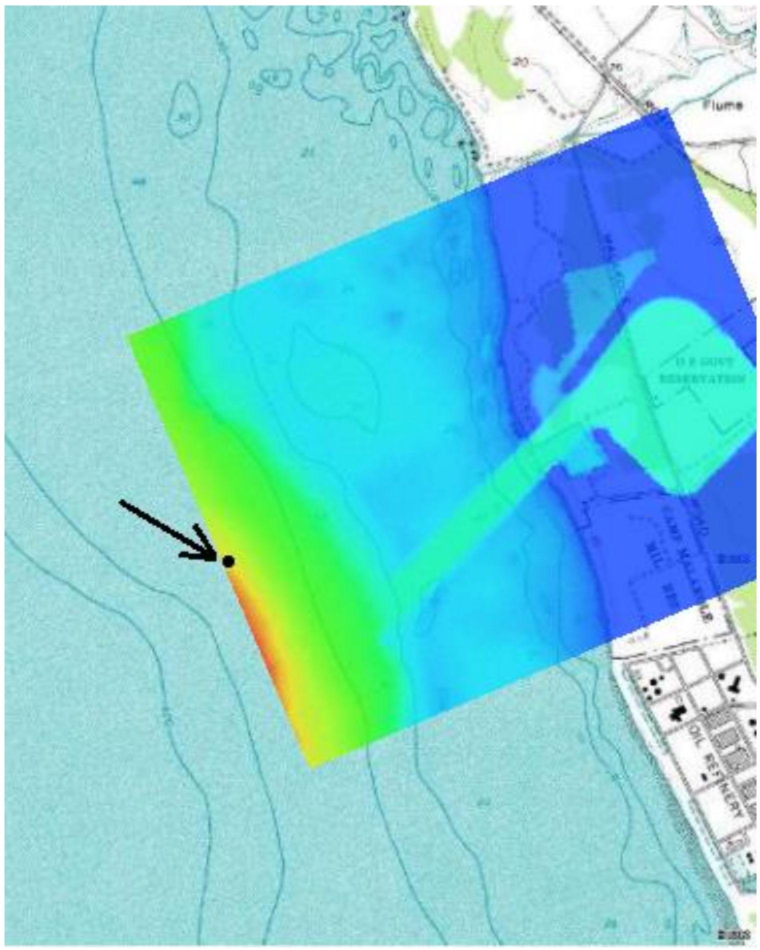


Figure 9   Wave Maker map coverage with feature point

6 Defining the Wave Maker

The BOUSS-2D wave maker must be positioned along a straight line in the grid. In SMS, create the wave maker along a straight arc at a desired location. Ideally, depth should be constant along this arc.

To define a wave maker, follow these steps:

1. Using the **Select Feature Point** tool, select the newly created point.
2. Right-click and select **Node Attributes...** to bring up the *BOUSS-2D Wave Generator Properties* dialog.
3. Under *Wave Simulation Parameters*, set *Type* to “Irregular Unidirectional”. Leave the defaults to synthesize the time series and generate the spectra from parameters.
4. Set *Series duration* to “750.0” seconds.
5. Click **OK** to acknowledge the message about changing the period for all wave makers.
6. Under *Spectral Parameters,* set *Type* to “JONSWAP Spectrum” and *Option* to “Specify hs and Tp”. This allows the significant wave height and peak period values (H_s & T_p) to be set as follows:
    - Sig. Wave Height (m) as “3.0”
    - Peak Wave Period (s) as “15.0”
    - Leave the other parameters set to their defaults
7. Under *Directional Parameters,* verify *Projection* is set to “Meteorologic” and change the *Wave angle* to “245”.
8. Click **OK** to close the BOUSS-2D Wave Generator Properties dialog.
9. If asked, click the **Yes** button to force a constant elevation. As SMS generates the wave maker, it is informing us that the offshore edge of the grid is not of constant depth.

7 Creating Roughness Coefficients

To create the roughness coefficients, do the following:

1. Right-click on Map Data and select **New Coverage** to open the *New Coverage* dialog again.
2. Select “Roughness” under the BOUSS-2D folder as the *Coverage type*, enter “Roughness” in *Coverage Name,* and click **OK** to close the *New Coverage* dialog.
3. Click on the new “Roughness” coverage to make it active.
4. With the **Create Feature Arc** tool, click out five enclosed arcs as shown in Figure 10.
5. Select *Feature Object* | **Build Polygons**.

6. Select the **Select Feature Polygon** tool and double-click on polygon 1 to bring up the *Roughness* dialog.
7. Set the *Chézy coefficient* to “60” and click **OK** to close the *Roughness* dialog.
8. Repeat step 6–7 for each of the other four polygons, setting the *Chézy coefficient* for each as follows:
   - Polygon 2: “30”
   - Polygon 3: “25”
   - Polygon 4: “40”
   - Polygon 5: “30”
9. Right-click on the “Roughness” coverage and select *Convert* | **Map → Active Grid** to bring up the *Map → Active Grid* dialog.
10. Set the *Default Value* to “30.0”, and set *Dataset Name* to “Roughness”. Click **OK** to close the *Map → Active Grid* dialog.

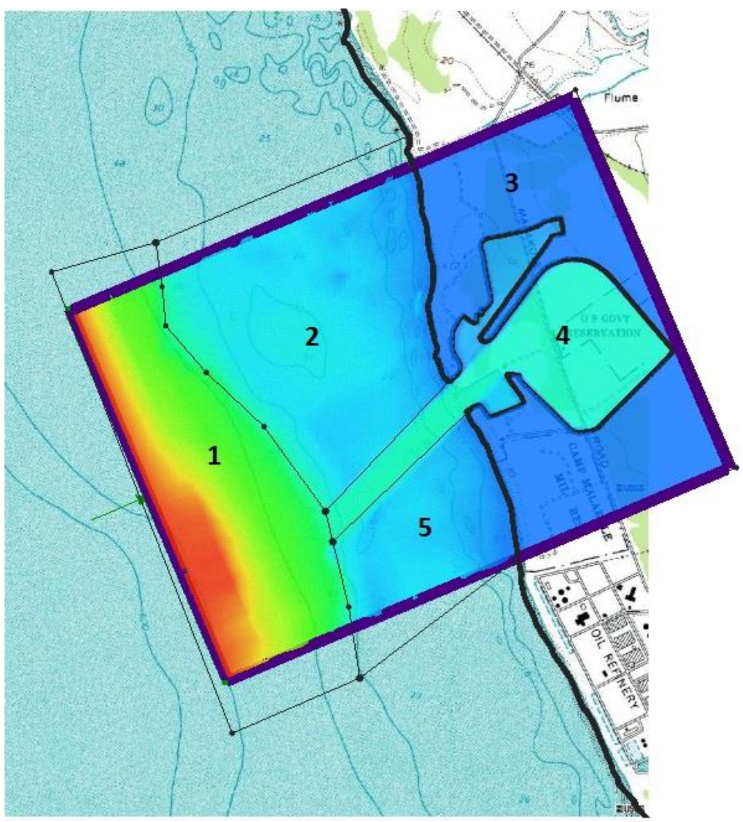


Figure 10 Polygons

8 Creating a BOUSS-2D Simulation

The next step in preparing a simulation is to specify model input parameters by first creating the BOUSS-2D simulation:

1. Right-click on the empty space at the bottom of Project Explorer and choose *New Simulation* | **BOUSS-2D**.
2. Right-click on the newly created “Simulation” item and select Rename.
3. Give the simulation the name “BarbersPoint”.
4. Drag both the “10m Grid” and the “Wave Maker” coverage under the “BarbersPoint” simulation.
5. Right-click on the “BarbersPoint” simulation and select **Model Control** to bring up the *BOUSS-2D Model Control* dialog.
6. Specify *Project title* as “Barbers Point Sample Run”.
7. Under *Time Control,* set *Duration* to”1500” seconds. This value should be greater than the computed default.
8. Set *Time step* to “0.25” seconds. The default time step is set to correspond with a Courant number of “0.6”. Reducing the time step increases stability. The time step should not be increased.
9. Turn on all options under *Time independent* under *Output Options*.
10. In the *Animation Output* section, turn on *Output WSE*, *Output Velocity* and *Override Defaults* options. For this case, first generate a series of solutions corresponding to the last five wave cycles (75 seconds) by changing the settings as follows:
    - Begin Output: “1425.0”
    - End Output: “1500.0”
    - *Step:* “1.0” (fifteen frames per wave)

The defaults for the above three parameters save the water level and velocity at even increments for the entire simulation. This generally results in either a huge output file or a discontinuous set of solution snap shots. An approximate size for the solution file is displayed to the left under *Required memory*.

11. Select the **OK** button to close the *BOUSS-2D Model Control* dialog.

9 Defining Damping Layers

If no damping is applied to the model, wave energy that is not dissipated along gradually sloping beaches will be bounced back into the domain. To calibrate to real world cases, or to prevent wave reflections from spreading back to the wave maker, it may be necessary to employ damping layers.

BOUSS-2D uses a dataset with a damping value at each cell to compute the damping of waves. A damping value of 0.0 indicates no damping and is the default. As the damping value increases (up to 1.0), and the number of damping cells adjacent to each other increases, the energy that is reflected back from the edge of the computation domain decreases.

Damping layers are created in coverages by doing the following:

1. Make sure the “Area Property” coverage is checked and “Elevation” under “10m Grid” is selected in the Project Explorer.
2. Right-click on the “BarbersPoint” simulation and select **Generate arcs along Land Boundary…** to bring up the *Generate Arcs* dialog.
3. For *Generate from* select “Grid”
4. Click the (none selected) button next to Item to bring up the *Select Tree Item* dialog.
5. Select “10m Grid” and click **OK** to close the *Select Tree Item* dialog.
6. Under *Destination coverage*, select *Create New Coverage* and set the *Type* to “Damping”. Leave *Name* at the default of “Damping”.
7. Set *Elevation* to “0” and click **OK** to close the *Generate Arcs* dialog. This builds a new coverage called “Damping” that contains an arc along the boundary.
8. Select the “Damping” coverage.
9. Using the **Select Feature Arc** tool, double-click on the arc. This will bring up the *Damping Properties* dialog.
10. Enter a *Width* of “20.0” (the combined width of two cells) and a Coefficient of “0.2”. Click **OK** to close the *Damping Properties* dialog.

The width value for the damping depends on several factors including wave length and grid resolution. This small damping coefficient will reduce wave reflection inside the harbor.

11. Lastly, drag this damping coverage under the “BarbersPoint” simulation if it is not already linked to the simulation.

10 Saving and Running the Simulation

The final step before running a simulation is to save the files for BOUSS-2D by doing the following:

1. Select *File* | **Save As…** to bring up the *Save As* dialog.

2. Enter "BarbersPoint_damping.sms" in *File name*, and click **Save**. This saves all the data files for execution.
3. Right-click on the BarbersPoint simulation and select **Export BOUSS-2D**.
4. Right-click again and select **Run BOUSS-2D**. The *BOUSS-2D* model wrapper will appear.
   - A *Model Checker* dialog may appear, warning that the selected run time is longer than the recommended time. Click **Run Model** to exit the dialog.
   - If using a normal installation of SMS, the model should launch immediately. If SMS cannot find the BOUSS-2D executable, a message will be displayed asking to locate the desired executable.

This simulation takes several minutes to run with a 0.25 sec time step. The model run time increases or decreases linearly based on the number of computational cells.

After the model run is complete:

5. Turn on the *Load solution* option.
6. Click the **Exit** button to close the *BOUSS-2D* model wrapper.
   - If the *Dataset Time Information* dialog appears, choose to use seconds for all datasets and click **OK** to exit the dialog.

11 Visualize Simulation Results

The model will create nine solution datasets that include spatially varying results at the grid nodes. Seven of these are scalar datasets:

- Mean wave level (Mean Water Level)
- Mean wave directions (Mean Wave Direction)
- Maximum Runup Height
- Significant wave height (Sig. Wave Height)
- Wave breaking animation (Breaking Animation)
- Velocity magnitude animation (Velocity Animation Mag),
- Water surface elevation animation (WSE Animation).

The other two are vector datasets:

- Mean Velocity
- Velocity Animation.

BOUSS-2D can save these results in two ways: in multiple separate files, or in a single binary file (HDF5 format). In this case, BOUSS-2D created the file

"BarbersPoint_sol.h5" at the end of the run. SMS creates a folder in the Project Explorer containing all of the datasets (in this case, "barberspoint").

To display a functional surface of the water surface:

1. Select the *Display* | **Display Options** to open the *Display Options* dialog.
2. Select *Cartesian Grid* from the list on the left, the turn off *Cells* and turn on *Contours* and *Functional surface*.
3. Click on the **Options** button right under the *Functional surface* option to bring up the *Functional Surface Options* dialog.
4. In the *Dataset* section, select the "User defined dataset" option to bring up the *Select Dataset* dialog
5. Select "WSE Animation" from the list of datasets, then click the **Select** button to close the *Select Dataset* dialog.
6. Under *Z Offset*, select "Display surface above geometry".
7. In the *Display Attributes* section, under *Use solid color,* click on the **Color** button to bring up the *Color* dialog.
8. Select the **light blue** (cyan) on the top row under *Basic colors* (fifth box from the left) then click **OK** to close the *Color* dialog.
9. Click **OK** to close the *Functional Surface Options* dialog.
10. On the *Contours* tab, set the *Contour method* to "Color Fill".
11. Select *General* from the list on the left, then turn off "Auto z-mag" in the *Drawing Options* section.
12. Set the *Z magnification* to "20.0". This amplifies the variation in the z-direction due to the very small wave heights compared to the size of the domain.
13. On the *Lighting* tab, select "Enable lights".
14. Turn on *Smooth edges* in the *Surface attributes for all lights* section
15. Click and drag the sphere until the dot is in the upper right quarter of the sphere, about a third of the way from the edge to the center of the sphere (see Figure 11). This will give good lighting contrast for the 3D view.

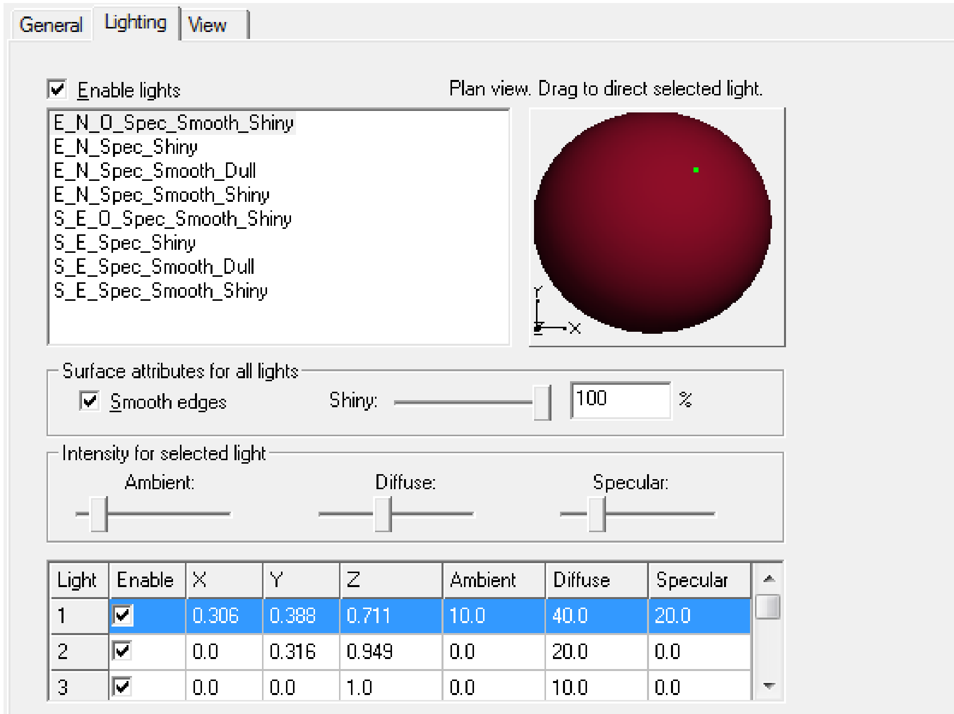


Figure 11 Lighting tab showing lighting direction

16. In the *View* tab, select *View angle* and enter a *Bearing* of “40.0” and a Dip of “25.0”.
17. Click **OK** to close the *Display Options* dialog.
18. Select the “Elevation” dataset in the Project Explorer to show the contour on the bottom of the ocean. The system may take a several minutes to update the display, depending on the capabilities of the computer.

Figure 12 shows this functional surface of the water surface over the bathymetric surface. The contour colors may vary. In this case, the contours are set to display a hue ramp with blue at the maximum end and the depth function active.

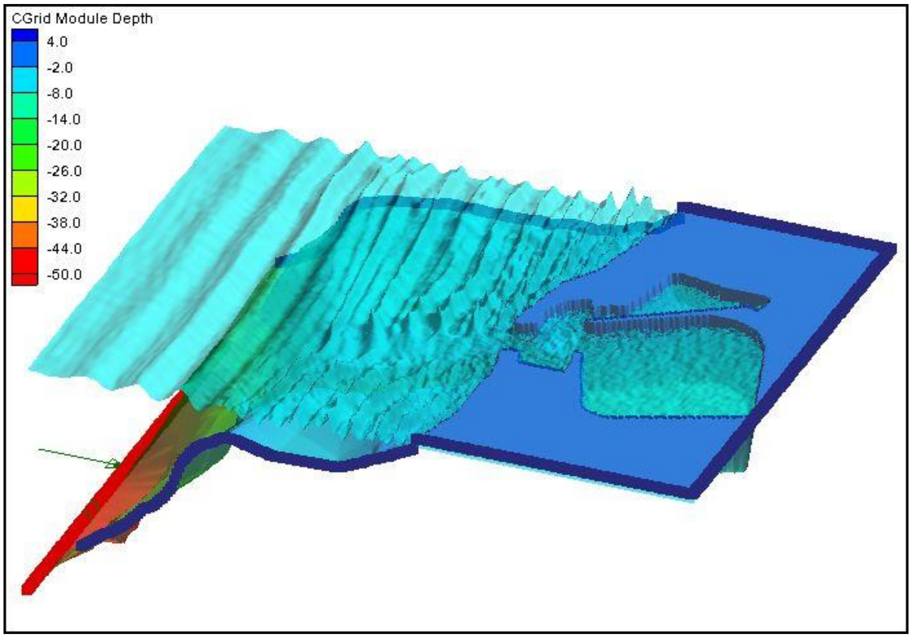


Figure 12 Water level functional surface over the bathymetry (magnified 20x)

All of the standard visualization methods described in the *Data Visualization* tutorial also work for the solutions generated by BOUSS-2D. Experiment with other options to view the solution.

12 Adding Probes

Probes can be added to the already existing Cartesian Grid in order to visualize the water surface elevation, pressure and velocities time series at that specific point on the grid. This can be useful to see the differences in water surface elevation for different parts of the grid.

To do this:

1. Make sure that the Grid is selected in the Project Explorer.
2. Using the **Select Grid Cell** tool, hold down the *Shift* key on the keyboard and click three cells roughly in the same position as that shown in Figure 13.
3. Release the *Shift* key. The three cells should be highlighted.

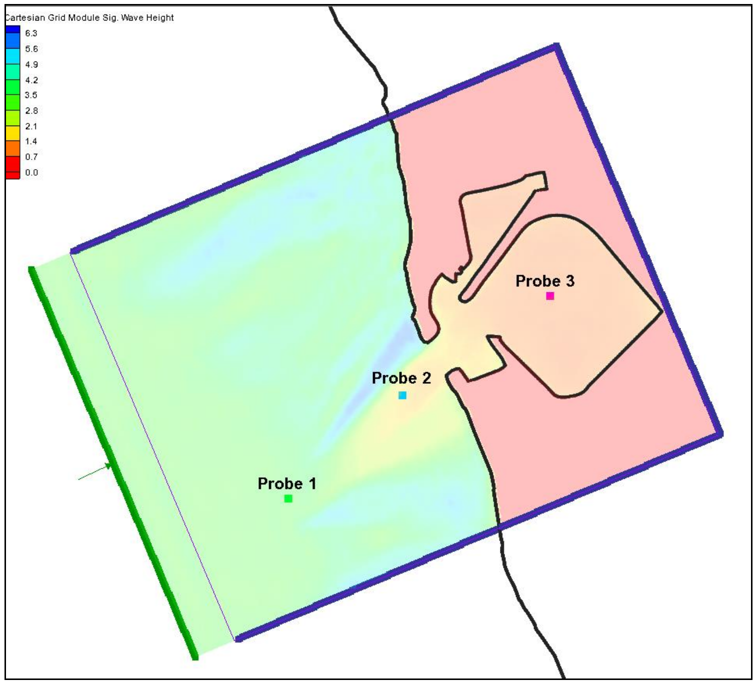


Figure 13 Probe positions

4. Right-click on the "BarbersPoint" simulation and select **Probe Manager** to bring up the *BOUSS-2D Probe Manager* dialog.
5. Click the **Create Probes at Selected Cells** button to create three probes at the selected cells.
6. Turn on *WSE (Water Surface Elevation)* for all three probes.
7. Turn on *Pressure* for all three probes.
8. Click on **Options** button next to the *Pressure* column for "Probe #1" to bring up the *Pressure Probe Options* dialog.
9. Enter an *Elevation above sea bed* of "1.0" and "5.0" then click **OK** to close the *Pressure Probe Options* dialog.
10. Repeat steps 8–9 for "Probe #2" and "Probe #3".
11. Turn on *Velocity* for the three probes.
12. Click on **Options** button next to the Velocity column for "Probe #1" to open the *UV Probe Options* dialog.
13. Enter an *Elevation above sea bed* of "2.0" and "6.0", then click **Ok** to close the *UV Probe Options* dialog. This will allow viewing of the velocity changes at 2.0 m and 6.0 m above sea level throughout the simulation run.
14. Repeat steps 12–13 for "Probe #2" and "Probe #3".

15. Click **OK** to exit the *BOUSS-2D Probe Manager* dialog.

12.1 Changing Display

The probes might not be easily visible at this point if the scatter set and Contours are turned on.

1. Select *Display* | **Display Options** to bring up the *Display Options* dialog.
2. Select *Cartesian Grid* from the list on the left and turn on *Cells*.
3. Turn off the *Ocean cell.*
4. Turn on *Probe without time series data*.
5. Turn on *Probe with time series data*.
6. Select the *Contours* tab and make sure the *Contour method* is set to “Color Fill”.
7. Set *Transparency* to “75%”.
8. Click **OK** to exit *Display Options* dialog.

Once BOUSS-2D is run and the solutions imported, the probes will have time series data and be seen as filled squares.

12.2 Saving New Project and Simulation Run

Now that the new probes have been created, it is time to run BOUSS-2D again. First, save a new SMS project by doing the following

1. Select *File* | **Save As…** to open the *Save As* dialog.
2. Enter a *File name* of “BarbersPoint_probes.sms” then click **Save**.
3. Right-click on the “BarbersPoint” simulation and select **Save Project, Export and Run BOUSS-2D** to start the *BOUSS-2D* model wrapper.
4. The simulation will take about 15 minutes to run. Once it is done, turn on the *Load Solution Files* option.
5. Click **Exit** to close the *BOUSS-2* model wrapper.

12.3 Reading in the Probe files and Viewing solutions

The solution files that were opened previously will not open the Probe files. Therefore these files must be opened manually.

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Browse to the folders where the project files are saved and select the four files with the extension “.ts1” then click **Open**.

3. Once the files are loaded, select the three probes using the **Select Grid Cell** tool and carefully select the appropriate cells.
4. Right-click on the display and select **Eta Time Series Plot** from the menu to bring up a plot window. Since all three probes are selected, three plots can be seen in the Time Series dialog. The plots should look like Figure 14.

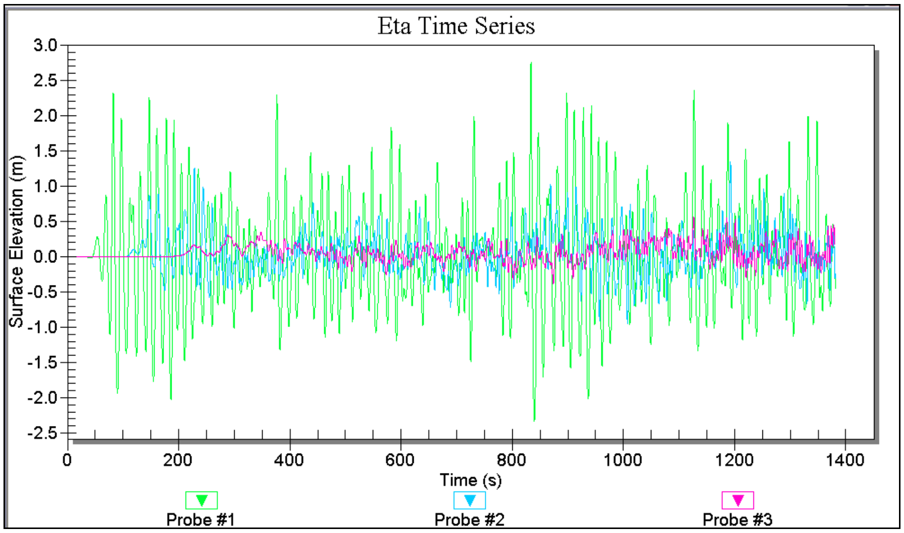


Figure 14 Water Surface Elevation Probe Plots

5. With the probes still selected, right-click on the display again and select *Zero-crossing Analysis* | **ETA Time Series**. This will bring up the *Zero-crossing Analysis* dialog.

The Zero-crossing Analysis will contain information for each probe.

- HAV (m) – Average wavelength for the full time series.
- H13 (m) – Average of the highest 3 % of Water Surface Elevation.
- H110 (m) – Average of the highest 10 % of Water Surface Elevation.
- HMAX (m) – Highest Water Surface Elevation.
- TAV (s) – Average time periods for the full time series.
- T13 (s) – Average of the highest 3 % of time periods.
- T110 (s) – Average of the highest 10 % of time periods.

13 Conclusion

This concludes the *BOUSS-2D* overview tutorial. Continue to experiment with the SMS interface or quit the program.