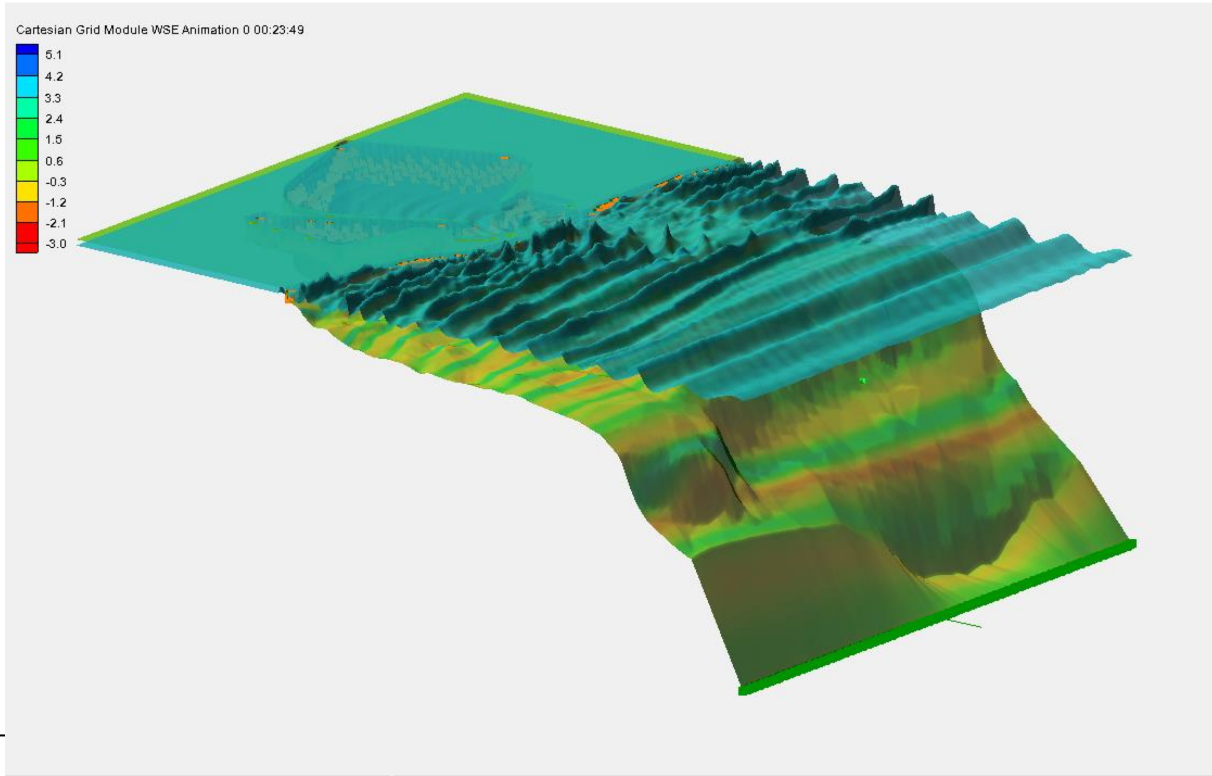
SMS

v. 12.2

SMS 12.2 Tutorial

BOUSS-2D



Objectives

Learn 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 using data from the area around Kalaeloa Barbers Point Harbor in Hawaii

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 runup 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 southwest corner of the island of Oahu, Hawaii.

2 Getting Started

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

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Browse to the *data files* folder for this tutorial and select “topo1.jpg”.
3. Click **Open** to import the image and exit the *Open* dialog.

The directory also includes a WLD file (“topo1.wld”). This indicates the image has been registered with geographic coordinates.

4. Repeat steps 1–3 to open “topo2.jpg”.
5. Select *File* | **Open…** to bring up the *Open* dialog again.
6. Select “bp_bathy_filtered.pts” and click **Open** to exit the *Open* dialog and bring up the *Step 1 of 2* page of the *File Import Wizard* dialog.

This file includes depth values obtained from a local survey in the Kalaeloa Barbers Point Harbor and the immediate coastal region outside the harbor.

7. In the *File import options* section, select *Delimited* and turn on *Space*.
8. Enter “2” for *Start import at row* and turn off *Heading row*.

9. Click **Next** to go to the *Step 2 of 2* page of the *File Import Wizard* dialog.
10. Click **Finish** to accept the defaults and close the *File Import Wizard* dialog.
11. Click **Display Options** to bring up the *Display Options* dialog.
12. Select "Scatter" from the list on that left.
13. On the *Scatter* tab, turn on *Points* and click **OK** to close the *Display Options* dialog.
14. Click **Open** to bring up the *Open* dialog.
15. Select "bp_coast.cst" and click **Open** to import the file and exit the *Open* dialog.

This file contains the coastline definition for the entire island of Oahu. After importing the bathymetric data, the project should appear similar to Figure 1.

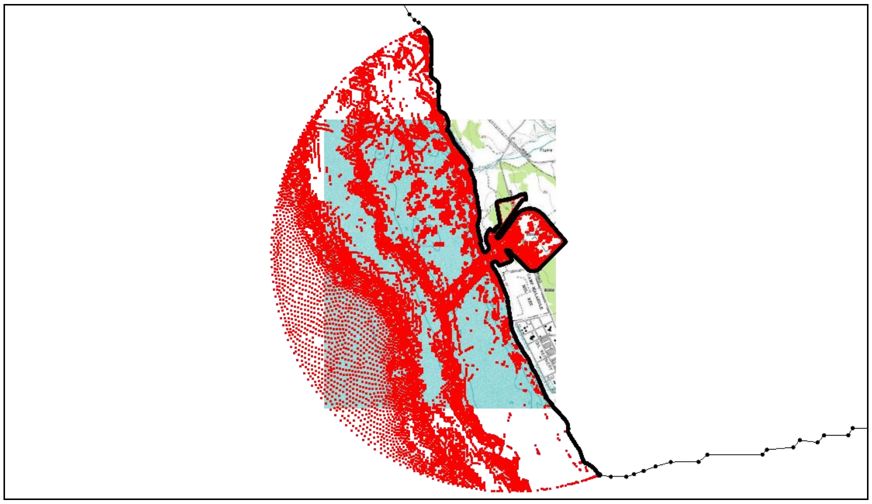


Figure 1 Bathymetry and images for project area

16. **Frame** the project

The project should appear similar to Figure 2.

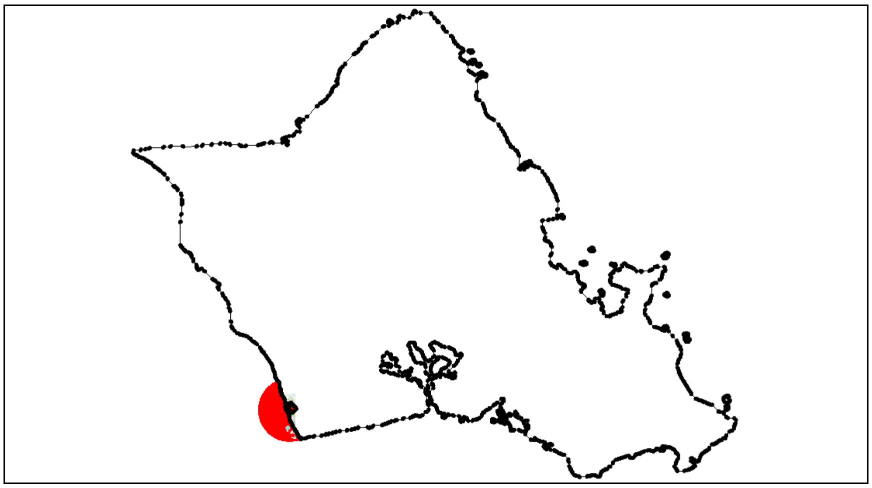


Figure 2 Coastline of Oahu with bathymetric data

3 Specifying Model Units

The images came from TerraServer[1] and are therefore registered to the Transverse Mercator NAD 83 coordinate frame. The bathymetry has been transformed to be relative to this coordinate frame. Importing 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. In the *Horizontal* section, verify that *Global projection* is selected and the field below that states "UTM, Zone: 4 (162˚West - 156˚W - Northern Hemisphere), NAD83, meters".
3. In the *Vertical* section, select "Meters" from the *Units* drop-down.
4. Click **OK** to exit the *Display Projection* dialog.
5. Right-click on " Area Property" and select **Projection…** to open the *Object Projection* dialog.
6. Repeat steps 2–3, then click **OK** close the *Object Projection* dialog.
7. Repeat steps 2–3 for " bp_bathy_filtered", then click **OK** close the *Object Projection* dialog.
8. **Frame** the project.

4 Trimming the Coastline

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

1. **Zoom** into the area being modeled (where the maps and the scattered point data are located).
2. Switch to the **Map Module** .
3. Right-click on the " Area Property" and select *Type* | *Models* | *BOUSS-2D* | **BOUSS-2D**.
4. Using the **Create Feature Arc** tool, create an arc as shown in Figure 3, beginning with P1 and continuing through P3.
5. Using the **Select Feature Arc** tool, select the coastline away from area of interest and hit the *Delete* key to eliminate this arc (the part to the right of P3 and above P1).
6. Click **Yes** when asked to confirm the deletion.

[1] See http://www.terraserver.com/ for more details.

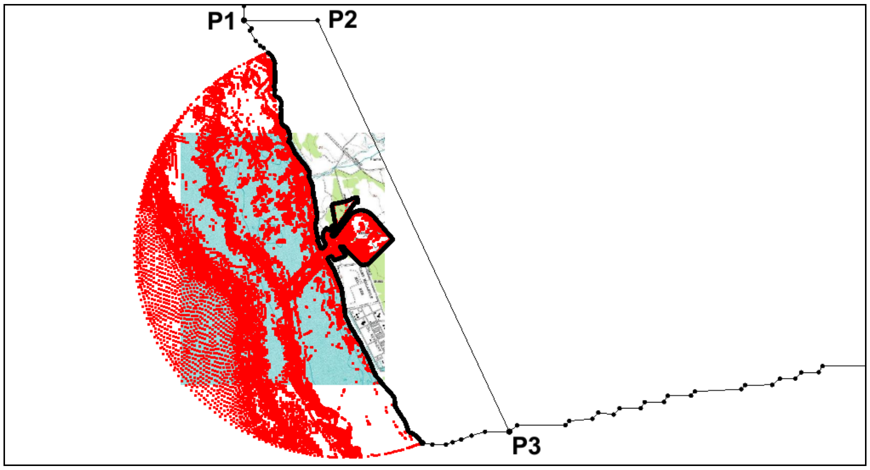


Figure 3 Arc bisecting area around simulation from island

7. **Frame** the project to show various islands which did not get deleted with the main island shoreline arc.
8. Using the **Select Feature Arc** tool, drag a box around the various island arcs and *Delete* them. Repeat until all the island arcs are deleted
9. **Frame** the project when done.

The project should appear similar to Figure 4.

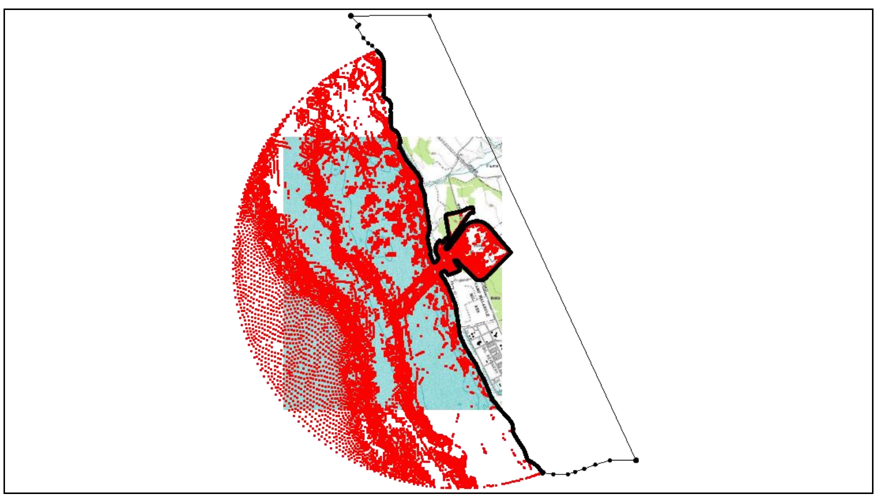


Figure 4 Area of interest after extra coastline arcs deleted

10. Build a polygon to represent the land around Kalaeloa Barbers Point Harbor by selecting *Feature Objects* | **Build Polygons**.
11. Using the **Select Polygon** tool, select the newly-created land polygon.
12. Select *Feature Objects* | **Select/Delete Data…** to bring up the *Select/Delete Data* dialog (Figure 5).
13. In the *Function Type* section, select *Delete*.
14. In the *Data Domain* section, select *Inside the polygon(s)*.

15. In the *Choose Data to Select* section, select *Scatter* and select “Triangles” from the drop-down.

This will delete triangles inside the polygon.

16. Turn on “bp_bathy_filtered” under “Scatter Data” in the dataset tree.
17. Click **OK** to close the *Select/Delete Data* dialog.

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

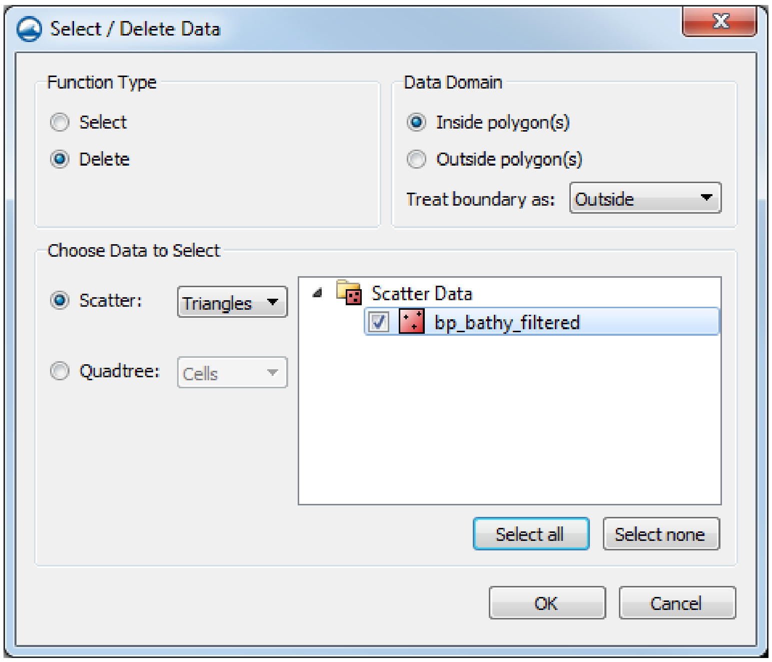


Figure 5 Select / Delete Data dialog

5 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 6.

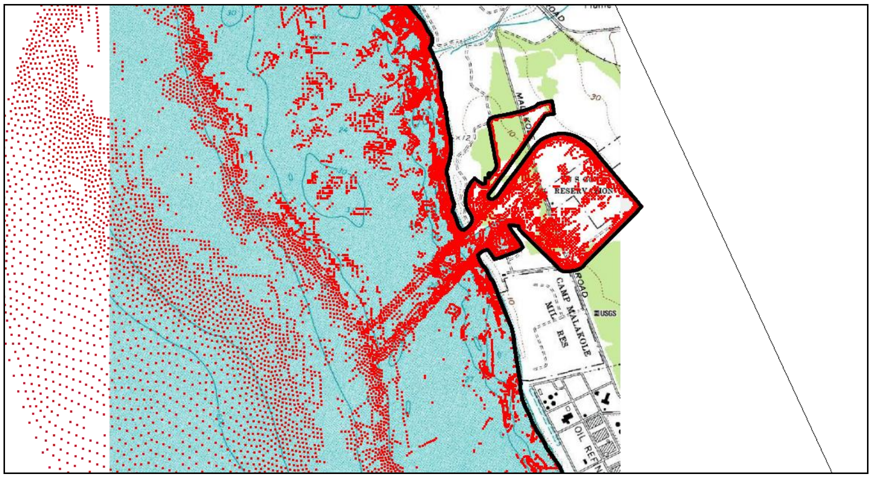


Figure 6 Zoomed view of harbor area

2. Select "Area Property" to make it active.
3. Using the **Create 2-D Grid Frame** tool, click a grid approximately as shown in Figure 7. The grid does not have to be exactly the same since it will be modified in the *Map → 2D Grid* dialog.

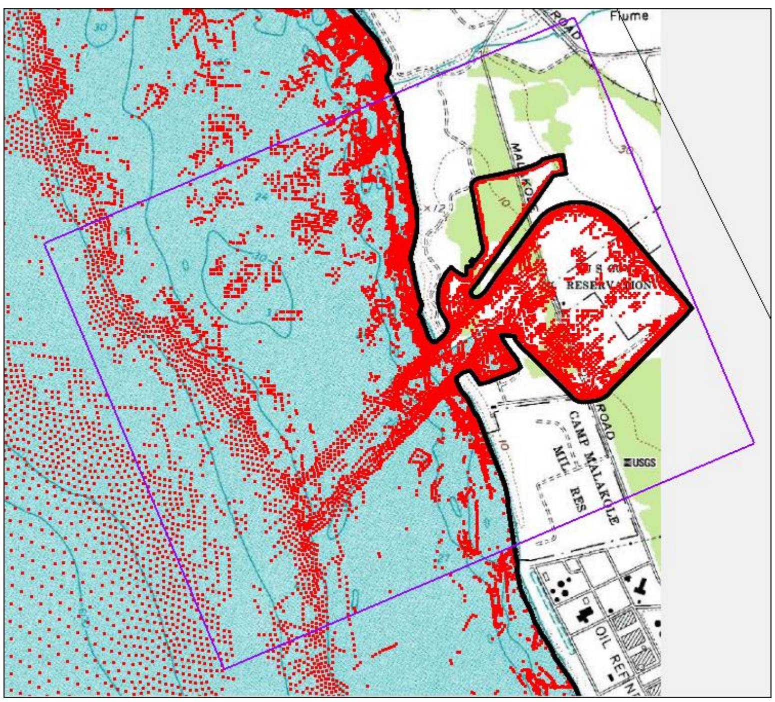


Figure 7 Grid frame

4. Right-click "Area Property" and select *Convert* | **Map → 2D Grid** to bring up the *Map → 2D Grid* dialog (Figure 8).
5. In the *Origin, Orientation and Dimensions* section, enter "590100.0" as the *Origin X*.
6. Enter "2356750.0" as the *Origin Y*.
7. Enter "23.0" as the *Angle*.

8. Enter "2500.0" as the *I Size*.
9. Enter "2000.0" as the *J Size*.
10. In the *Elevation options* section, select "Scatter Set" from the *Source* drop-down and click **Select…** to bring up the *Interpolation* dialog.
11. In the *Interpolation Options* section, enter "1.0" as the *Single Value*.

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.

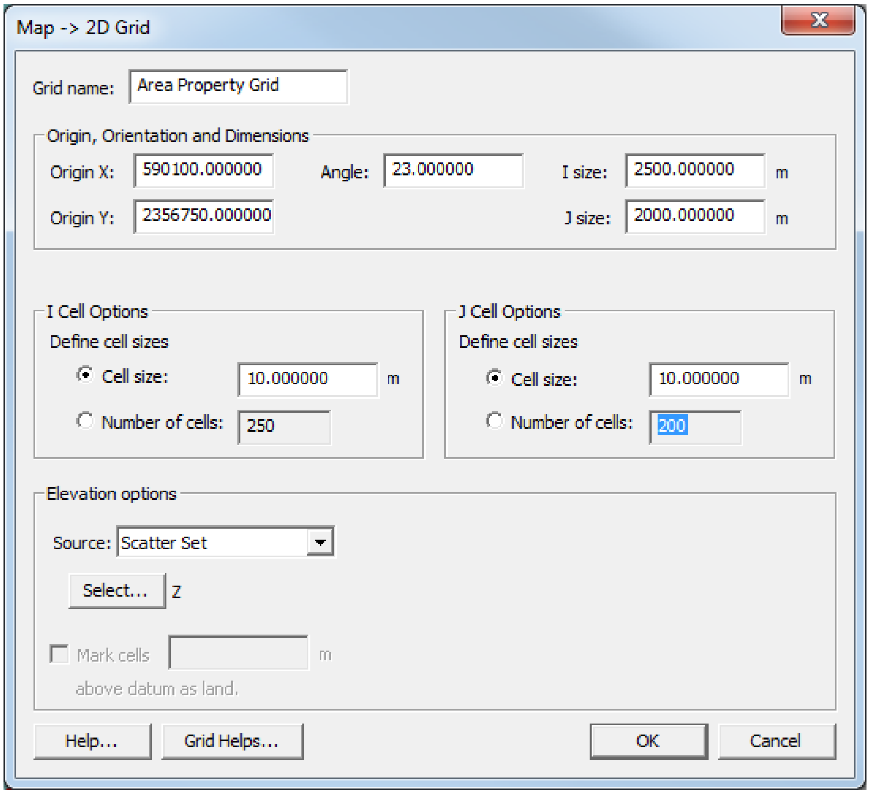


Figure 8 Parameters to create the grid

12. Click **OK** to close the *Interpolation* dialog.

For the next part, keep in mind that an appropriate cell size depends on the wavelength of the waves being modeled.

13. Click **Grid Helps…** to open the *BOUSS-2D Map → 2D Grid Helps* dialog.
14. Turn on *T (wave period)* and enter "15.0".
15. Click **OK** to close the *BOUSS-2D Map → 2D Grid Helps* dialog.

Notice that the recommended cell size is about 11.7 meters. Smaller cells increase the definition of the model, but also increase computation time. For the purposes of this tutorial, a smaller cell size will be used.

16. Enter "10.0" as the *Cell size* in both the *I Cell Options* and *J Cell Options* sections.
17. Click **OK** to close the *Map → 2D Grid* dialog and create the grid.

18. Right-click on the newly created " Area Property Grid" and select **Rename**.
19. Enter "10m Grid" and press *Enter* to set the new name.

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.

20. Turn off " Scatter Data" and " Map Data" in the Project Explorer.
21. Select *Display* | **Display Options…** to open the *Display Options* dialog.
22. Select "Cartesian Grid" from the list on the left.
23. On the *Cartesian Grid* tab, turn off *Cells* and turn on *Contours*.
24. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
25. Enter "25" as the *Transparency*.
26. Click **OK** to close the *Display Options* dialog.

The display should appear similar to Figure 9.

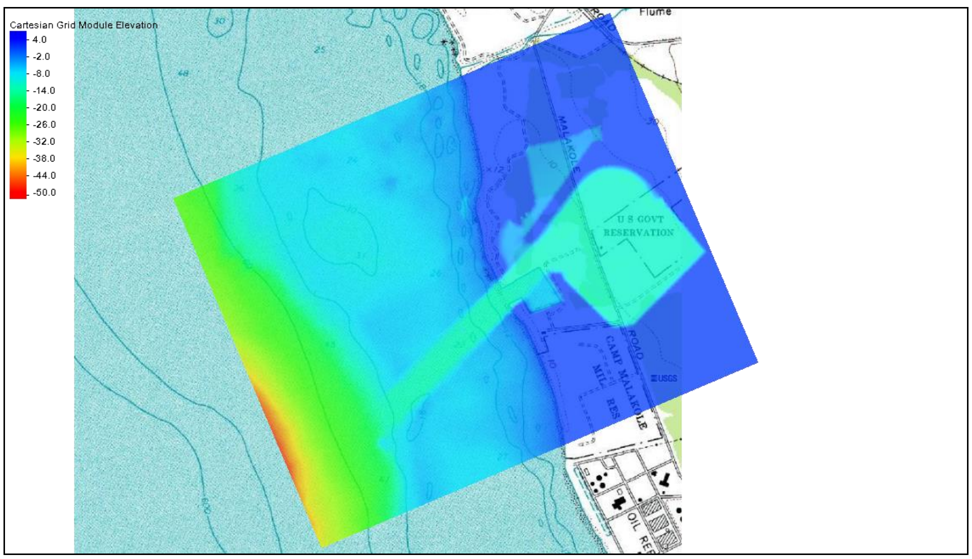


Figure 9 Resulting grid contours

6 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. In the *Coverage type* section, select *Models* | *BOUSS-2D* | **Wave Maker**.
3. Enter "Wave Maker" as the *Coverage Name*.
4. Click **OK** to close the *New Coverage* dialog.

5. Select "Wave Maker" to make it active.
6. Using the **Create Feature Point** tool, create a point in the new coverage as indicated in Figure 10.

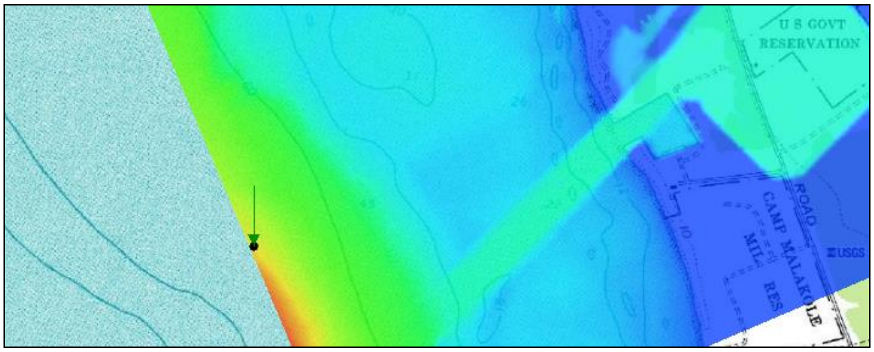


Figure 10 Wave Maker map coverage with feature point

7 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. In the *Wave Simulation Parameters* section, select "Irregular Unidirectional" from the *Type* drop-down.
4. Enter "750.0" as the *Series duration* and press *Tab*.
5. Click **OK** to acknowledge the message about changing the period for all wave makers.
6. In the *Spectral Parameters* section, select "JONSWAP Spectrum" from the *Type* drop-down.
7. Select "Specify hs and Tp" from the *Option* drop-down.

This allows the significant wave height and peak period values ($H_{s\,\&}$ T_p) to be set as follows:

8. In the spreadsheet, enter "3.0" as *Sig. Wave Height (m)*.
9. Enter "15.0" as *Peak Wave Period (s)*.
10. In the *Directional Parameters,* select "Meteorologic" from the *Projection* drop-down.
11. Enter "245.0" as the *Wave angle*.
12. Leave all other parameters at their defaults.

13. Click **OK** to close the *BOUSS-2D Wave Generator Properties* dialog.
14. If asked, click the **Yes** button to force a constant elevation.

As SMS generates the wave maker, this indicates that the offshore edge of the grid is not of constant depth. The wave maker should appear similar to Figure 11.

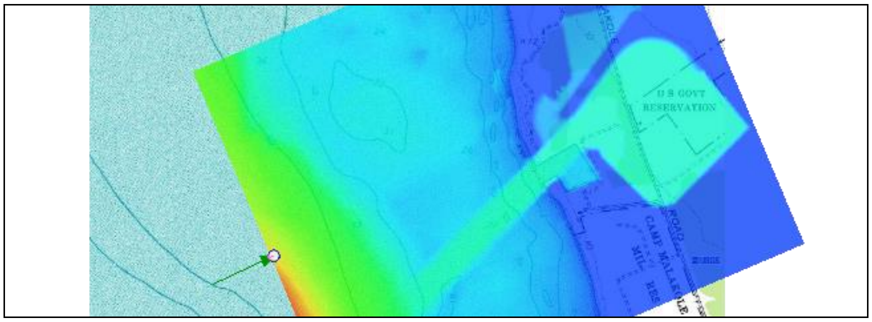


Figure 11 Arrow indicates direction of waves from the wave maker

8 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.
2. In the *Coverage Type* section, select *Models* | *BOUSS-2D* | **Roughness**.
3. Enter "Roughness" as the *Coverage Name*.
4. Click **OK** to close the *New Coverage* dialog.
5. Select the new " Roughness" coverage to make it active.
6. Using the **Create Feature Arc** tool, click out five enclosed arcs as shown in Figure 12.
7. Select *Feature Object* | **Build Polygons**.
8. Using the **Select Feature Polygon** tool, double-click on polygon 1 (as indicated in Figure 12) to bring up the *Roughness* dialog.
9. Enter "60.0" as the *Chézy coefficient* and click **OK** to close the *Roughness* dialog.
10. Repeat step 8–9 for polygon 2, entering "30.0" as the *Chezy coefficient*.
11. Repeat step 8–9 for polygon 3, entering "25.0" as the *Chezy coefficient*.
12. Repeat step 8–9 for polygon 4, entering "40.0" as the *Chezy coefficient*.
13. Repeat step 8–9 for polygon 5, entering "30.0" as the *Chezy coefficient*.
14. Right-click " Roughness" and select *Convert* | **Map → Active Grid** to bring up the *Map → Active Grid* dialog.
15. Enter "30.0" as the *Default Value*.
16. Enter "Roughness" as the *Dataset Name*.

17. Click **OK** to close the *Map → Active Grid* dialog.

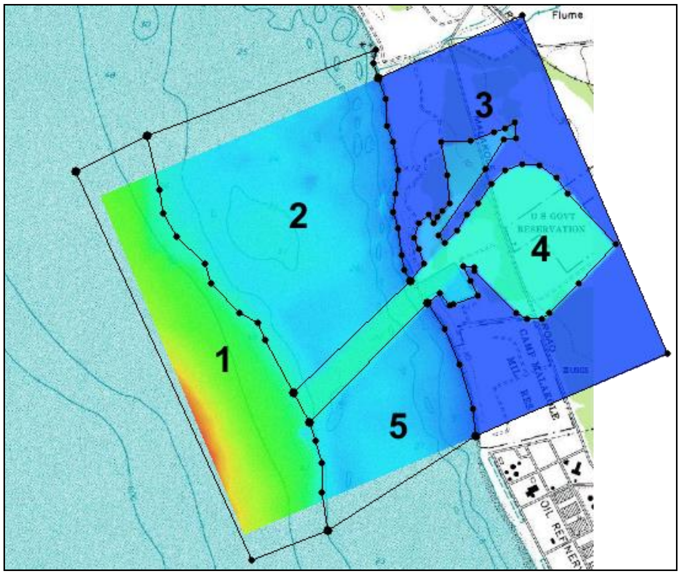


Figure 12 Polygons

9 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 an empty space in the Project Explorer and select *New Simulation* | **BOUSS-2D**.
2. Right-click on the newly created " Simulation" item and select **Rename**.
3. Enter "BarbersPoint" and press *Enter* to set the new name.
4. Drag both " 10m Grid" and " Wave Maker" under " BarbersPoint".
5. Right-click on " BarbersPoint" and select **Model Control…** to bring up the *BOUSS-2D Model Control* dialog.
6. Enter "Barbers Point Sample Run" as the *Project title*.
7. In the *Time Control* section, enter "1500.0" as the *Duration*.

This value should be greater than the computed default (the number in the *Recommended* column) in that field.

8. Enter "0.25" as the *Time step*.

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. In the *Parameters* section, turn on *Enable wave runup* and enter "0.018" as the *Min. flooding depth*.
10. In the *Output Options* section, turn on all options under *Time independent*.

11. Under *Animation Output*, turn on *Output WSE*, *Output Velocity* and *Override Defaults*.

For this case, first generate a series of solutions corresponding to the last five wave cycles (75 seconds) by changing the settings as follows:

12. Enter "1425.0" for *Begin output*.
13. Enter "1500.0" for *End output*.
14. Enter "1.0" as the *Step.*

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*.

15. Click **OK** to close the *BOUSS-2D Model Control* dialog.

10 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. Turn on " Area Property" and select " Elevation" under " 10m Grid" in the Project Explorer.
2. Right-click on " BarbersPoint" and select **Generate arcs along Land Boundary…** to bring up the *Generate Arcs* dialog.
3. Select "Grid" from the *Generate from* drop-down.
4. Click **(none selected)** 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. In the *Destination coverage* section, select *Create New Coverage*
7. Select "Damping" from the *Type* drop-down.
8. Enter"0" as the *Elevation.*
9. Click **OK** to close the *Generate Arcs* dialog.

This builds a new coverage called "Damping" that contains an arc along the boundary (Figure 13).

10. Select " Damping" to make it active.

11. Using the **Select Feature Arc** tool, double-click on the arc to bring up the *Damping Properties* dialog.
12. Enter "20.0" as the *Width* of "20.0" (the combined width of two cells).
13. Enter "0.2" as the *Coefficient*.
14. 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.

15. Drag " Damping" under " BarbersPoint" if it is not already linked to the simulation.
16. Click anywhere outside of the damping boundary arc to deselect it.

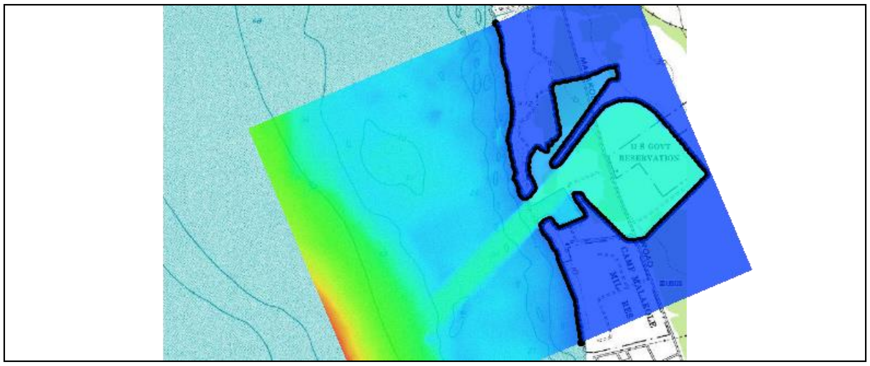


Figure 13 Boundary arc along the shoreline

11 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" as the *File name*.
3. Click **Save** to save the project under the new name and exit the *Save As* dialog.

This saves all the data files for execution.

4. Right-click on " BarbersPoint" and select **Export BOUSS-2D**.
5. Right-click on " BarbersPoint" and select **Run BOUSS-2D** to bring up the *BOUSS-2D* model wrapper dialog.
6. A *Model Checker* dialog may appear, warning that the selected run time is longer than the recommended time. Click **Run Model** to exit the *Model Checker* 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 may take several minutes to run with a 0.25 sec time step, depending on the speed of the computer being used. The model run time increases or decreases linearly based on the number of computational cells.

7. After the model run is complete, turn on *Load solution* and click **Exit** to close the *BOUSS-2D* model wrapper dialog.
8. If the *Dataset Time Information* dialog appears, choose to use seconds for all datasets and click **OK** to exit the *Dataset Time Information* dialog.

12 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:

- Wave breaking animation (Breaking Animation)
- Maximum Runup Height (Max. Runup Height)
- Mean wave level (Mean Water Level)
- Mean wave directions (Mean Wave Direction)
- Significant wave height (Sig. Wave Height)
- 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 *Display* | **Display Options…** to open the *Display Options* dialog.
2. Select *Cartesian Grid* from the list on the left.
3. On the *Cartesian Grid* tab, turn off *Cells* and turn on *Contours* and *Functional surface*.
4. Click **Options…** right under the *Functional surface* option to bring up the *Functional Surface Options* dialog.
5. In the *Dataset* section, select *User defined dataset* to bring up the *Select Dataset* dialog
6. Select "WSE Animation" from the list of datasets and click **Select** to close the *Select Dataset* dialog.
7. In the *Z Offset* section, select "Display surface above geometry" from the drop-down.

8. In the *Z Magnification* section, turn on *Override global value* and enter "50.0" as the *Magnification value*.
9. In the *Display Attributes* section, select *Use solid color* and click the larger *Color* button to bring up the *Color* dialog.
10. Select **cyan** on the top row under *Basic colors* (fifth box from the left) and click **OK** to close the *Color* dialog.
11. Click **OK** to close the *Functional Surface Options* dialog.
12. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
13. Select *General* from the list on the left.
14. On the *General* tab, turn off "Auto z-mag" in the *Drawing Options* section.
15. Enter "20.0" as the *Z magnification*.

This amplifies the variation in the z-direction due to the very small wave heights compared to the size of the domain.

16. On the *Lighting* tab, turn on *Enable lights*.
17. Turn on *Smooth edges* in the *Surface attributes for all lights* section
18. 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 14). This will give good lighting contrast for the 3D view.

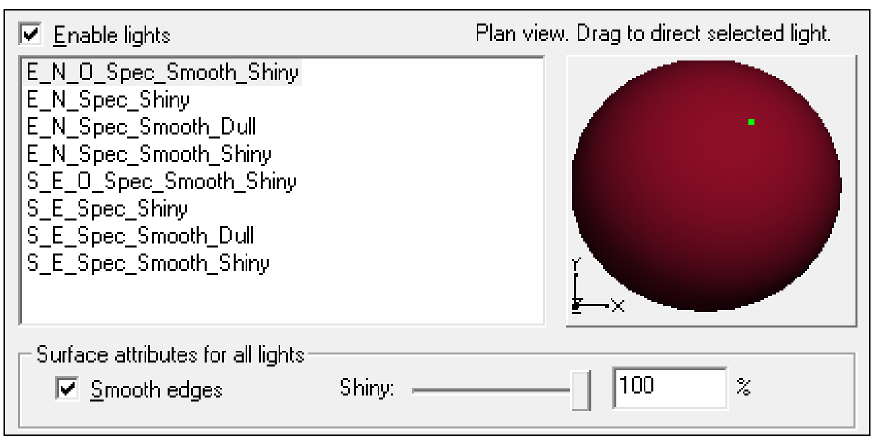


Figure 14 Lighting tab showing lighting direction

19. On the *View* tab, select *View angle* and enter "40.0" as the *Bearing* and "25.0" as the *Dip*.
20. Click **OK** to close the *Display Options* dialog.
21. 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 15 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.

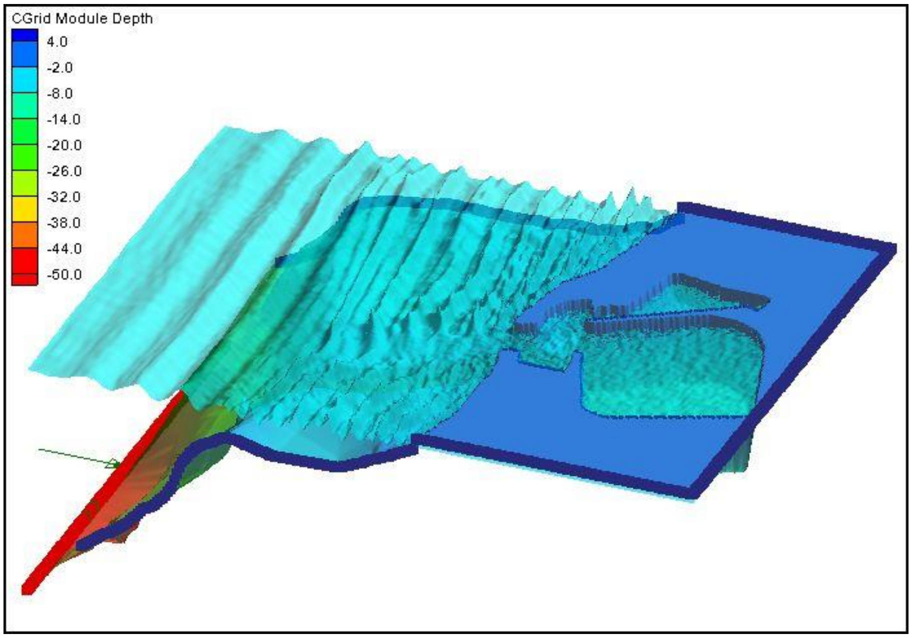


Figure 15 Water level functional surface over the bathymetry (magnified 20x)

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.

13 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. Select " 10m Grid" in the Project Explorer.
2. Using the **Select Grid Cell** tool, hold down the *Shift* key and select three cells in roughly the same position as those shown in Figure 16.
3. Right-click " BarbersPoint" and select **Probe Manager…** to bring up the *BOUSS-2D Probe Manager* dialog.
4. Click **Create Probes at Selected Cells** to create a probe at each of the selected cells.
5. Turn on *WSE (Water Surface Elevation)* for all three probes.
6. Turn on *Pressure* for all three probes.
7. Click **Options…** to the right of the *Pressure* column for "Probe #1" to bring up the *Pressure Probe Options* dialog.

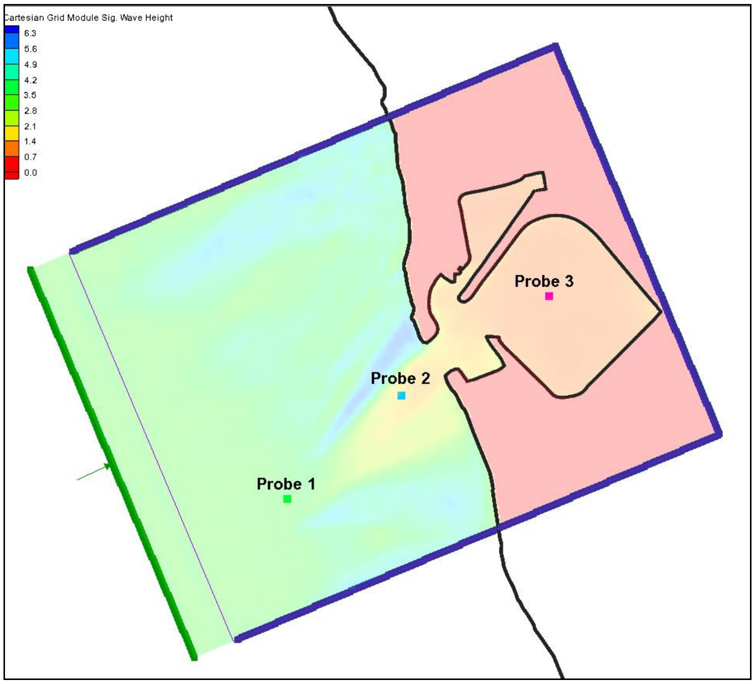


Figure 16 Probe positions

8. Enter "1.0" in the *Elevation above sea bed* column and press *Enter* to create a new blank line.
9. Enter "5.0" on the new blank line and press *Enter*.
10. Click **OK** to close the *Pressure Probe Options* dialog.
11. Repeat steps 7–10 for "Probe #2" and "Probe #3".
12. Turn on *Velocity* for the three probes.
13. Click **Options…** to the right of the *Velocity* column for "Probe #1" to open the *UV Probe Options* dialog.
14. Enter "2.0" in the *Elevation above sea bed* column and press *Enter* to create a new blank line.
15. Enter "6.0" in the *Elevation above sea bed* column and press *Enter*.
16. Click **OK** to close the *UV Probe Options* dialog.

This allows viewing of the velocity changes at 2.0 m and 6.0 m above sea level throughout the simulation run.

17. Repeat steps 13–16 for "Probe #2" and "Probe #3".
18. Click **OK** to exit the *BOUSS-2D Probe Manager* dialog.

13.1 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.
3. On the *Cartesian Grid* tab, turn on *Cells* and turn off *Ocean cell.*
4. Turn on *Probe without time series data*.
5. Turn on *Probe with time series data*.
6. On the *Contours* tab, make sure "Color Fill" is selected from the first drop-down in the *Contour method* section.
7. Enter "75" as the *Transparency*.
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.

13.2 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 "BarbersPoint_probes.sms" as the *File name*.
3. Click **Save** to save the project under the new name and close the *Save As* dialog.
4. Right-click " BarbersPoint" and select **Save Project, Export and Run BOUSS-2D** to open the *BOUSS-2D* model wrapper dialog.

Depending on the speed of the computer being used, the simulation may take several minutes to finish.

5. Once BOUSS-2D is finished, turn on *Load Solution Files* and click **Exit** to close the *BOUSS-2* model wrapper dialog.

13.3 Importing 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 *data files\BOUSS-2D\BarbersPoint* folder and select to following files:
    - "barberspoint_ts_eta.ts1"
    - "barberspoint_ts_pressure.ts1"
    - "barberspoint_ts_u.ts1"
    - "barberspoint_ts_v.ts1"
3. Click **Open** to import the files and exit the *Open* dialog.
4. Once the files are loaded, select the three probes using the **Select Grid Cell** tool and carefully select the appropriate cells.

5. 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 17.
6. 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.

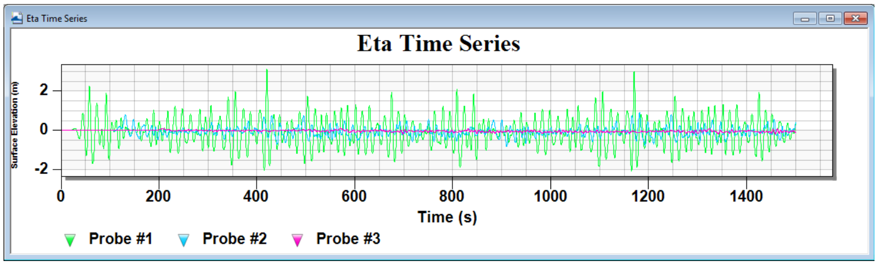


Figure 17 Water surface elevation probe plots

14 Conclusion

This concludes the “BOUSS-2D” overview tutorial. Feel free to continue experimenting with the SMS interface, or exit the program.

