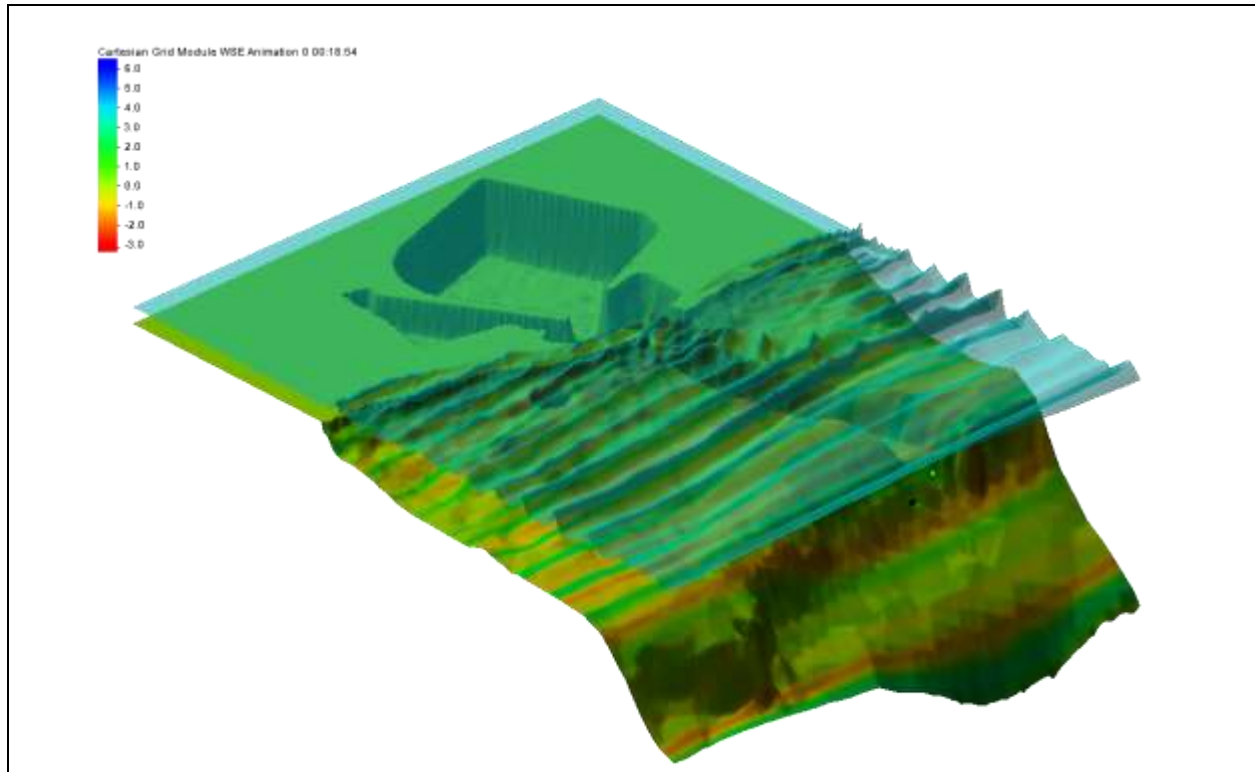


SMS 13.1 Tutorial

BOUSS-2D Advanced



Objectives

Learn to use the BOUSS-2D interface advanced features, roughness and probes.

Prerequisites

- Overview Tutorial
- BOUSS-2D
- Cell-Centered Grid

Requirements

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

Time

- 20–30 minutes

AQUAVEO™



1	Introduction	2
2	Getting Started	2
2.1	Saving the Project with a New Name	3
3	Creating the Roughness Coverage	3
3.1	Creating the New Coverage.....	3
3.2	Creating the Roughness Area Arcs.....	4
3.3	Deleting the Coastline Arc	4
3.4	Setting the Chézy Coefficients	4
4	Creating a New Grid	5
4.1	Converting the Roughness Coverage	5
5	Creating a New Simulation	5
6	Running the Simulation	6
7	Visualizing Simulation Results	6
7.1	Displaying a Functional Surface.....	7
8	Adding Probes.....	9
8.1	Creating a New Simulation	9
8.2	Adding Probes	9
8.3	Changing Display	11
8.4	Running the Simulation	11
8.5	Importing the Probe Files and Viewing Solutions	12
9	Conclusion.....	13

1 Introduction

The basic features of BOUSS-2D were discussed in the “Creating a Cell-Centered Grid” and “BOUSS-2D” tutorials. BOUSS-2D also allows for taking into account the ocean floor roughness and for placing probes at various depths to visualize the water surface elevation, pressure, and velocities time series at that specific point on the grid.

The data used for this tutorial includes images, bathymetry data, and coastline data for Kalaeloa Barbers Point Harbor, located on the southwest corner of the island of Oahu, Hawaii.

2 Getting Started

The cell-centered grid used for this tutorial was previously created. 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 “BarbersPoint.sms”.
3. Click **Open** to import the project and exit the *Open* dialog.

The project should appear similar to Figure 1. For more information on how the grid was generated, see the “Creating a Cell-Centered Grid” tutorial.

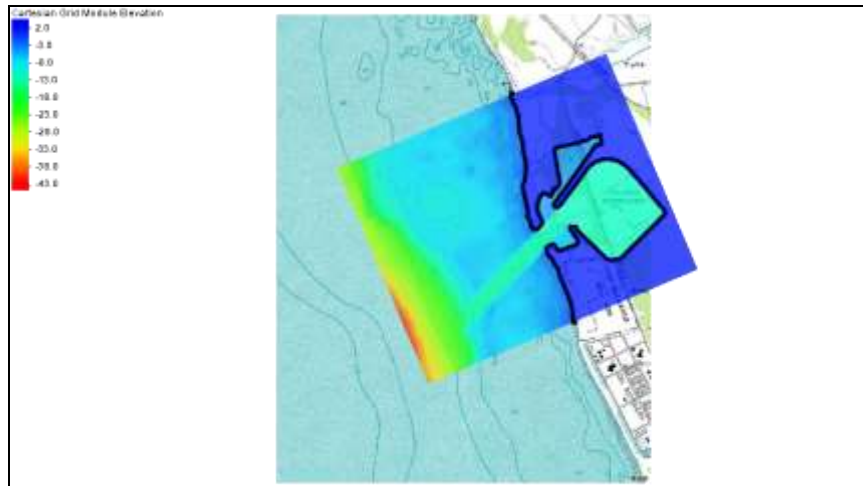



Figure 1 Initial project

2.1 Saving the Project with a New Name

Before continuing, it is best to save the project under a new name in order to preserve the starting project for future use.





1. Select *File* | **Save As...** to bring up a *Save As* dialog.
2. Enter the *File name* “BarbersPoint_roughness.sms” and click **Save**.

3 Creating the Roughness Coverage

There are two ways to create the Roughness coverage: create a new coverage from scratch, or duplicate an existing coverage. By duplicating the “ Area Property” coverage, the existing coastline arc can be used as a base to create the roughness polygons.




3.1 Creating the New Coverage

Create the coverage by doing the following:

1. Right-click on “ Area Property” and select **Duplicate** to create a new “ Area Property (2)” coverage.
2. Right-click on “ Area Property (2)” and select **Rename**.
3. Enter “Roughness” and press *Enter* to set the new name.
4. Right-click on “ Roughness” and select *Type* | *Models* | **BOUSS-2D | Roughness**.
5. Click **OK** when warned about gridframes being deleted.

3.2 Creating the Roughness Area Arcs

Now that the coverage has been created, finish creating the arcs needed for the roughness polygons by doing the following:

1. Select “ Roughness” to make it active.
2. Using the **Create Feature Arc**  tool, click out the borders for sections 1, 2, and 5, connecting them to the existing coastline arc as shown in Figure 2.
3. Using the **Create Feature Arc**  tool, create the outside border for section 3, connecting it to the existing coastline arc as shown in Figure 2.
4. Select *Feature Object* | **Build Polygons**.

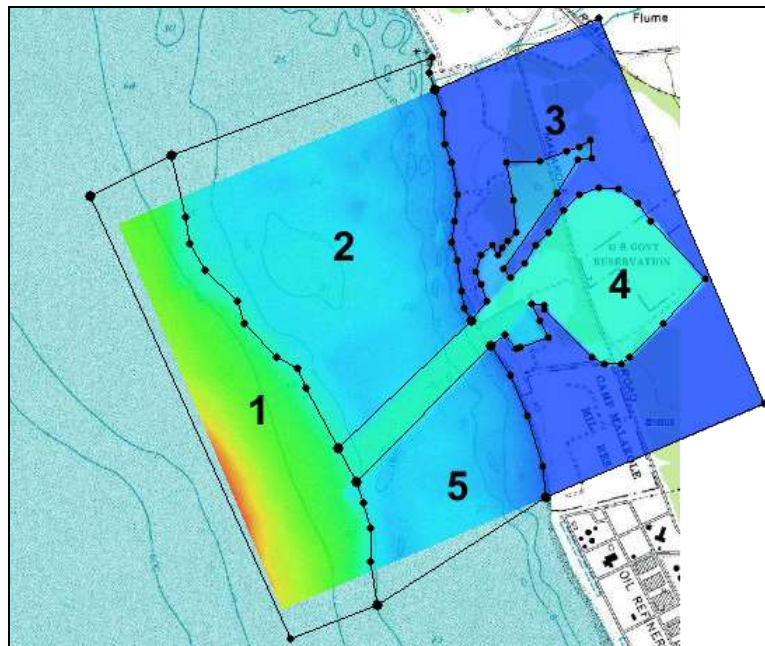





Figure 2 Polygons


3.3 Deleting the Coastline Arc

The coastline arc outside of the grid is not needed, so delete it by doing the following:

1. **Frame**  the project.
2. Using the **Select Feature Arc**  tool while holding down the *Shift* key, select the four coastline arcs outside of the roughness polygons and press *Delete*.
3. Click **Yes** when asked to confirm the deletion.
4. **Frame**  the project.





3.4 Setting the Chézy Coefficients

Now to set the Chézy coefficient for each polygon:

1. Using the **Select Feature Polygon**  tool, double-click on polygon 1 (as indicated in Figure 2) to bring up the *Roughness* dialog.
2. Enter “60.0” as the *Chezy coefficient* and click **OK** to close the *Roughness* dialog.
3. Repeat step 1–2 for polygon 2, entering “30.0” as the *Chezy coefficient*.
4. Repeat step 1–2 for polygon 3, entering “25.0” as the *Chezy coefficient*.
5. Repeat step 1–2 for polygon 4, entering “40.0” as the *Chezy coefficient*.
6. Repeat step 1–2 for polygon 5, entering “30.0” as the *Chezy coefficient*.


4 Creating a New Grid



As with coverages, there are multiple ways to create a new 2D grid. In this case, duplicate the existing grid to save time.

1. Right-click on “ 10m Grid” and select **Duplicate** to create a new “ 10m Grid (copy)” grid.
2. Right-click on “ 10m Grid (copy)” and select **Rename**.
3. Enter “10m Grid Roughness” and press *Enter* to set the new name.
4. Select “ 10m Grid Roughness” to make it active.

4.1 Converting the Roughness Coverage




Now that the new grid has been created, create a Roughness dataset under it by converting the roughness coverage.





1. Right-click “ Roughness” and select *Convert* | **Map** → **Active Grid** to bring up the *Map* → *Active Grid* dialog.
2. Enter “30.0” as the *Default Value*.
3. Enter “Roughness” as the *Dataset Name*.
4. Click **OK** to close the *Map* → *Active Grid* dialog.

A new “ Roughness” datasets should appear under “ 10m Grid Roughness” in the Project Explorer.

5 Creating a New Simulation

Instead of using the existing simulation, it is recommended to create a new simulation in order to model the area while retaining the ability to compare the results of multiple simulations. This allows differences to be easily compared for evaluation.

1. Right-click on “ BarbersPoint” in the Project Explorer and select **Duplicate** to create a new “ BarbersPoint (2)” simulation.
2. Right-click on “ BarbersPoint (2)” and select **Rename**.



3. Enter “BP_roughness” and press *Enter* to set the new name.
4. Right-click on “ 10m Grid” under “ BP_roughness” and select **Delete**.
5. Click **Yes** in the dialog asking to delete selected items.
6. Drag “ 10m Grid Roughness” under “ BP_roughness”.

The new coverage, grid, and simulation have been created. Now to save the project before proceeding:

7. Click **Save** .


6 Running the Simulation



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

1. Right-click on “ BP_roughness” and select **Export BOUSS-2D**.
2. Right-click on “ BP_roughness” and select **Run BOUSS-2D** to bring up the *BOUSS-2D* model wrapper dialog.
3. If a *Model Checker* dialog appears with a warning that the selected run time is longer than the recommended time, click **Run Model** to exit the *Model Checker* dialog.

If using a standard 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.

Depending on the speed of the computer being used, the model run may take several minutes to complete. An estimate of the remaining time will display in the *BOUSS-2D* model wrapper dialog. The model run time increases or decreases linearly based on the number of computational cells.

4. When the model run finishes, turn on *Load solution* and click **Exit** to close the *BOUSS-2D* dialog and bring up the *Select Tree Item for Datasets* dialog.
5. If the *Dataset Time Information* dialog appears, choose to use seconds for all datasets and click **OK** to exit the *Dataset Time Information* dialog and bring up the *Select Tree Item for Datasets* dialog.
6. Select “ 10m Grid Roughness” from the tree list and click **OK** to close the *Select Tree Item for Datasets* dialog.

A set of scalar and vector datasets should appear in a “ bp_roughness” folder under “ 10m Grid Roughness” in the Project Explorer.

7 Visualizing Simulation Results

The model run created eight solution datasets that include spatially varying results at the grid nodes. Six of these are scalar datasets:

- Breaking Animation (wave breaking animation)

- Mean Water Level
- Mean Wave Direction
- Sig. Wave Height (significant wave height)
- Velocity Animation Mag (velocity magnitude animation)
- WSE Animation (water surface elevation 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 “bp_roughness_sol.h5” at the end of the run. SMS creates a folder in the Project Explorer containing all of the datasets (in this case, “BP_roughness”).

7.1 Displaying a Functional Surface

To display a functional surface of the water, do the following:

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...** 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 “30.0” 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 as shown in Figure 3. This gives good lighting contrast for the 3D view.

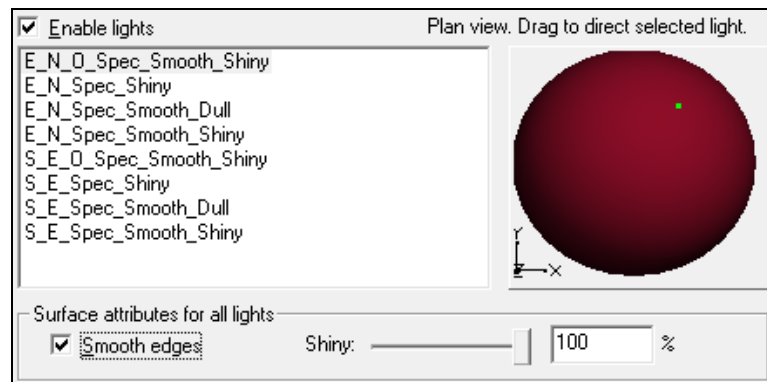




Figure 3 Lighting tab showing lighting direction

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 under “ 10m Grid Roughness” in the Project Explorer to show the contour on the bottom of the ocean.
22. Step through the solution time steps to see the changes in the surface.

The system may take several minutes to update the display, depending on the capabilities of the computer. Figure 4 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.

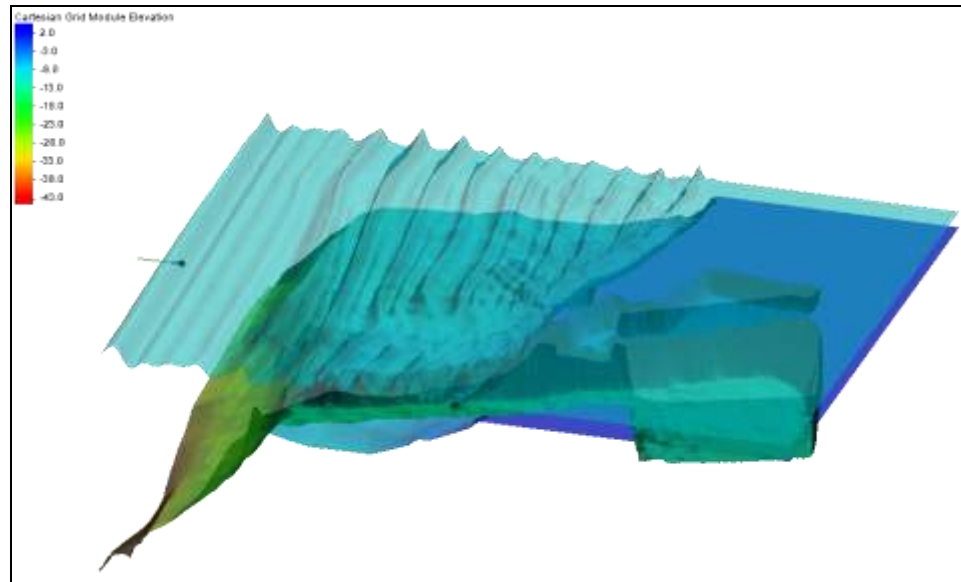


Figure 4 Water level functional surface over the bathymetry (magnified 20x)

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

8 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.

8.1 Creating a New Simulation


Before adding the probes, it is recommended to create a second simulation in the project. This way the original simulation is saved and can be compared to the new simulation.

1. Right-click “BP_roughness” and select **Duplicate**.
2. Right-click on “BP_roughness (2)” and select **Rename**.
3. Enter “BP_probes” and press *Enter* to set the new name.

8.2 Adding Probes

Now add the probes to the grid by doing the following:

1. Select “10m Grid Roughness” in the Project Explorer.
2. Select “Elevation” under “10m Grid Roughness”.
3. Switch to **Plan View**.

4. 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 5.
5. Right-click “BP_probes” and select **Probe Manager...** to bring up the *BOUSS-2D Probe Manager* dialog.
6. Click **Create Probes at Selected Cells** to create a probe at each of the selected cells.
7. Turn on *WSE (Water Surface Elevation)* for all three probes.
8. Turn on *Pressure* for all three probes.
9. Click **Options...** to the right of the *Pressure* column for “Probe #1” to bring up the *Pressure Probe Options* dialog.
10. Enter “1.0” in the *Elevation above Seabed (m)* column and press *Enter* to create a new blank line.
11. Enter “5.0” on the new blank line and press *Enter*.
12. Click **OK** to close the *Pressure Probe Options* dialog.
13. Repeat steps 9–12 for “Probe #2” and “Probe #3”.
14. Turn on *Velocity* for the three probes.
15. Click **Options...** to the right of the *Velocity* column for “Probe #1” to open the *UV Probe Options* dialog.
16. Enter “2.0” in the *Elevation above Seabed (m)* column and press *Enter* to create a new blank line.
17. Enter “6.0” in the *Elevation above Seabed (m)* column and press *Enter*.
18. 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.

19. Repeat steps 15–18 for “Probe #2” and “Probe #3”.
20. Click **OK** to exit the *BOUSS-2D Probe Manager* dialog.

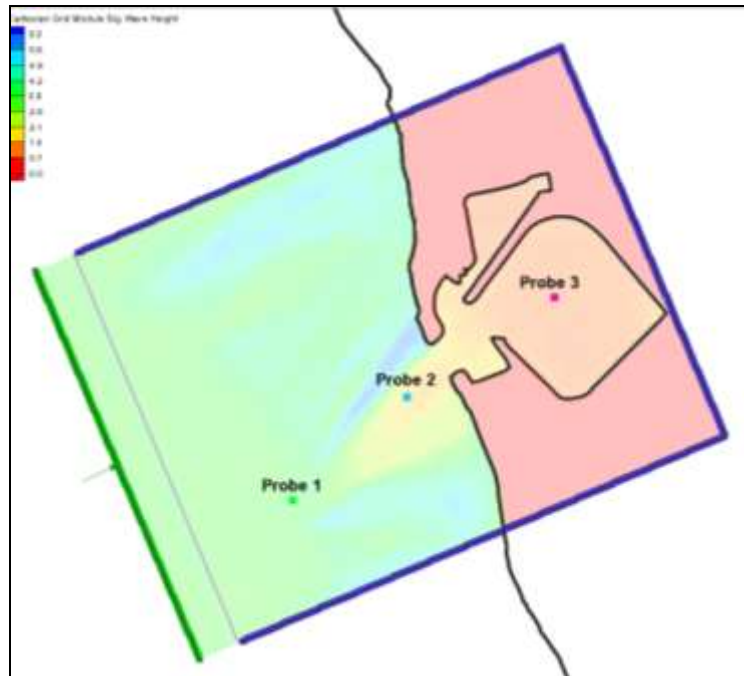


Figure 5 Probe positions

8.3 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.

8.4 Running the Simulation

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. Right-click “BP_probes” 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.

2. Once BOUSS-2D is finished, turn on *Load Solution* and click **Exit** to close the *BOUSS-2D* model wrapper dialog and bring up the *Select Tree Item for Datasets* dialog.
3. Select “ 10m Grid Roughness” and select **OK** to exit the *Select Tree Item for Datasets*.

8.5 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\BP_probes* folder and select the following files:
 - “bp_probes_ts_eta.ts1”
 - “bp_probes_ts_pressure.ts1”
 - “bp_probes_ts_u.ts1”
 - “bp_probes_ts_v.ts1”
3. Click **Open** to import the files and exit the *Open* dialog.
4. Select **OK** to close the dialog stating where the props were added.
5. Once the files are loaded, select the three probes using the **Select Grid Cell**  tool and carefully select the appropriate cells while holding the *Shift* key.
6. 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 appear similar to Figure 6 below.

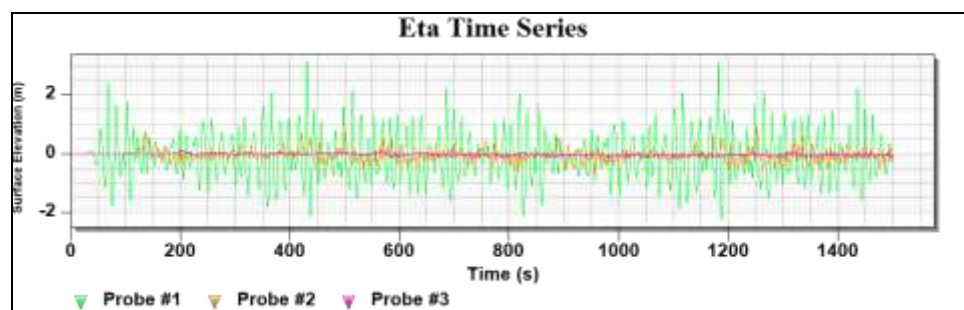
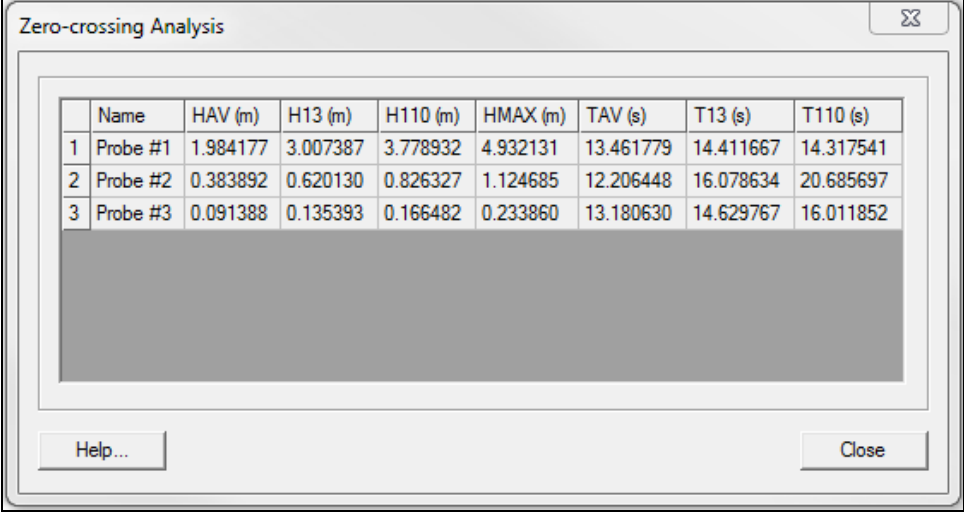


Figure 6 Water surface elevation probe plots

7. 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 (Figure 7).

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.



The image shows a software dialog box titled "Zero-crossing Analysis". It contains a table with 9 columns: Name, HAV (m), H13 (m), H110 (m), HMAX (m), TAV (s), T13 (s), and T110 (s). There are three rows of data, each for a different probe. Below the table is a large empty rectangular area. At the bottom of the dialog are "Help..." and "Close" buttons.

	Name	HAV (m)	H13 (m)	H110 (m)	HMAX (m)	TAV (s)	T13 (s)	T110 (s)
1	Probe #1	1.984177	3.007387	3.778932	4.932131	13.461779	14.411667	14.317541
2	Probe #2	0.383892	0.620130	0.826327	1.124685	12.206448	16.078634	20.685697
3	Probe #3	0.091388	0.135393	0.166482	0.233860	13.180630	14.629767	16.011852

Figure 7 Zero-crossing Analysis dialog

9 Conclusion

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