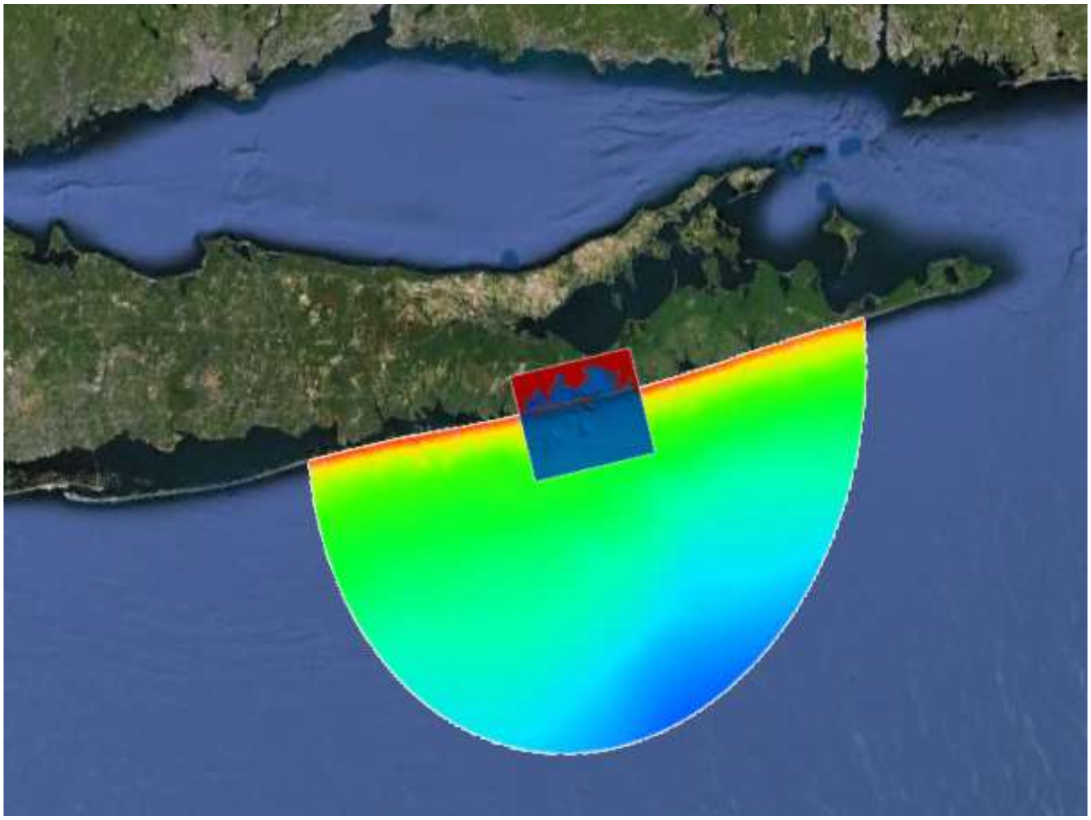
v. 11.2

SMS 11.2 Tutorial

STWAVE Analysis



Objectives

This workshop gives a brief introduction to the STWAVE modules. Data from the Shinnecock Inlet, Long Island, New York, have been set up as an example. This example will use a scatter set generated in another tutorial. An STWAVE grid will be created over a small section of the scatter set data.

Prerequisites

- Overview Tutorial

Requirements

- Map Module
- STWAVE
- Cartesian Grid Module
- Scatter Module

Time

- 30-60 minutes

1 Converting ADCIRC to Scatter

1.1 Reading in the ADCIRC files

First, open the scatter set data of the area around Shinnecock Inlet on the south shore of Long Island, New York. For convenience, the scatter set data, and an XMDF version of the current file are supplied in the “data files” folder for this tutorial. To open the files:

1. Select *File* | **Open...** to bring up the *Open* dialog, and select the file “shinfinal.h5.”
2. Push **Open** to read in the data file.
3. Open the file “shincurrents.h5” in the same manner as in step 1 and 2.

1.2 Coordinate Conversions

The coordinate of the project are currently set to geographic coordinate but this project requires viewing the data from a state plane coordinate for New York Long Island. To do this:

1. Click *Display* and select **Projection...** to bring up the *Display Projection* dialog.
2. Click **Set Projection** to bring up the *Select Pojection* window.
3. In the *Select Projection* window, set the *Projection* to “State Plane Coordinate System.” The *Datum* should be “NAD83.” The *Planar Units* should be set to “METERS” with the *Zone* set to “New York–Long Island (FIPS 3104).” Click **OK.**

4. Back in the *Display Projection* dialog, make sure the *Vertical Units* are set to "Meters."
5. Click **OK** to close the *Display Projection* dialog.
6. Next, right-click on the scatter set in the Project Explorer and select **Projection**.
7. In the *Object Projection* dialog, make sure that *Global projection* is toggled on and click on **Set Projection**.
8. In the *Select Projection* dialog, make sure that the *Projection* is set to "Geographic (Latitude/Longitude)." Make sure that the *Datum* is set to "NAD83" and the *Planar Units* are in "Arc Degrees." Click **OK** twice.

2 Creating the Cartesian Grid

Now it's time to create a Cartesian grid for running STWAVE. The grid frame is created in the Map module. The Map module contains tools for creating GIS objects such as points, arcs, and polygons. It is also used for creating a frame, which will be filled in by a Cartesian grid.

2.1 Creating the Cartesian Grid Frame

To create the grid frame:

1. Right-click on the "Area Property" default coverage in the Project Explorer. In the *Type* menu, select *Models* and in that menu select **STWAVE**.
2. Right-click on "Area Property" again and select **Rename**. Then enter the name "STWAVE Bounds" as the new coverage name.
3. Right-click on the "STWAVE Bounds" coverage and click on **Projection**.
4. In the *Object Projection* dialog, select the *Global projection* radio button, then click **Set Projection.**
5. In the *Select Projection* window, set the *Projection* to "State Plane Coordinate System." The *Datum* should be "NAD83." The *Planar Units* should be set to "METERS" with the *Zone* set to "New York - Long Island (FIPS 3104)." Click **OK**.
6. Back in the *Object Projection* dialog, the *Vertical Units* should also be set to "Meters." Click **OK**.
7. Select the **Create 2-D Grid Frame** tool from the *Toolbox*.
8. Zoom in to the harbor a little and click out three corners of the grid in the order shown in Figure 1 to create the grid frame. The first two points the user clicks define the i-direction, which is the direction of the incoming waves, and the last two points the user clicks are placed on the land.

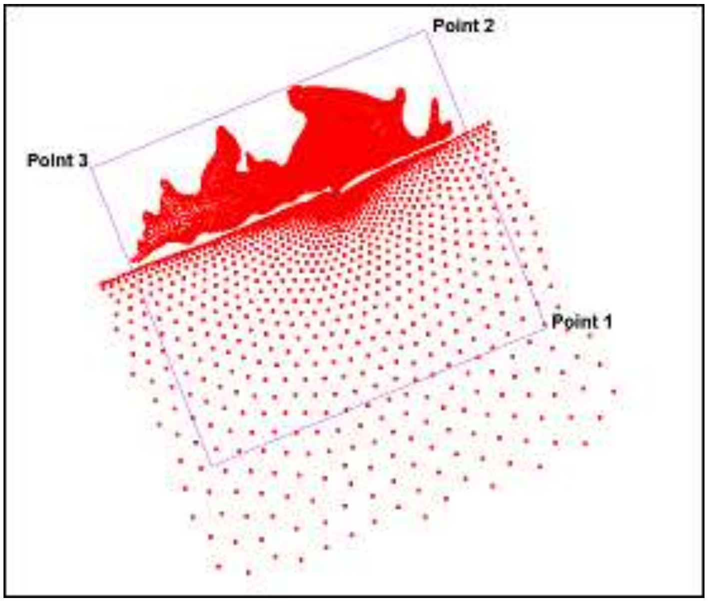


Figure 1 Creating the Cartesian Grid Frame

9. Switch to the **Select Grid Frame** tool and select the box in the middle of the grid frame (it may be hard to see behind the scatter points). The origin should be in the bottom right corner of the grid, which will be shown by the two arrows appearing at that corner.
10. A user can drag and resize the grid frame by dragging the corners or edges until the grid frame fits over the desired area. Dragging a corner or side resizes the frame. Dragging the middle point moves the entire frame. A user can rotate the frame around the origin by dragging the circle located just outside the grid.
11. The user can also right-click on the grid frame and select *Properties*. This brings up attributes of the grid frame in a *Grid Frame Properties* dialog.
12. For consistency purposes, set the *Origin X* to "438,000," the *Origin Y* to "70,000," and the *Angle* to "112." Also make the grid size about "15,000 "meters in the *I size* direction and about "17,000" meters in the *J size* direction. Click **OK** to exit the *Grid Frame Properties* dialog.
13. Click outside the grid frame to unselect the grid when finished.

2.2 Mapping to the Grid

Next, users will fill the interior of the grid. While the grid is filling, the depth and current values will be interpolated from the scatter set and mapped to each cell. To do this:

1. Go to *Feature Objects* | **Map→2D Grid**.
2. In the *Map →2D Grid* dialog, make sure the X, Y, and Angle values are set to "438,000," "70,000," and "112" respectively. Make sure the *I size* value is "15000" m and the *J size* value is "17000" m.

3. Make sure *Cell Size* is selected in the *Cell Options* and change the cell size in both directions to “100” m.
4. In the *Depth Options* section, make sure that the *Source* is set to “Scatter Set” and then click on the select button.
5. In the *Interpolation* dialog, choose “elevation” in the *Scatter Set to Interpolate From* section. Click **OK.**
6. Turn the *Map Vector* toggle on. Make sure the *Interpolated* option is selected and click the button underneath it.
7. In the *Interpolation* dialog, select the “Velocity (64) “scatterset. Make sure A*ll Time Steps* is selected. Select **OK** to exit the *Interpolation* dialog.
8. Push **OK** to create the Cartesian grid.
9. Right-click on the “STWAVE Bounds” grid item in the *Project Explorer* and select ***Rename***. Edit the name of the grid to “100m”. This indicates the resolution of the grid.

   Note on interpolation: It is easiest to interpolate currents when creating the 2D Grid even if the project won’t be using currents until a later simulation. Users can choose whether to use currents in the STWAVE model control or not. When interpolating, a user can specify a single time step or multiple steps. Single times come from any time in the dataset. For multiple steps, a user can specify to match all the steps from the dataset, or can specify a beginning and ending time step and a time step size.

A Cartesian grid has been created from the grid frame. To view the grid only:

1. Turn off the Scatter Data and Map Data in the Project Explorer by unchecking their respective toggle boxes in the tree.
2. **Frame** the display.
3. Click on the “100m” item to work on the STWAVE simulation.
4. Select the **Select Grid Cell** tool from the Cartesian Grid toolbox and select cells within the grid over the offshore edge of the grid to get an idea of the depth of the water. The average is approximately 32 m.

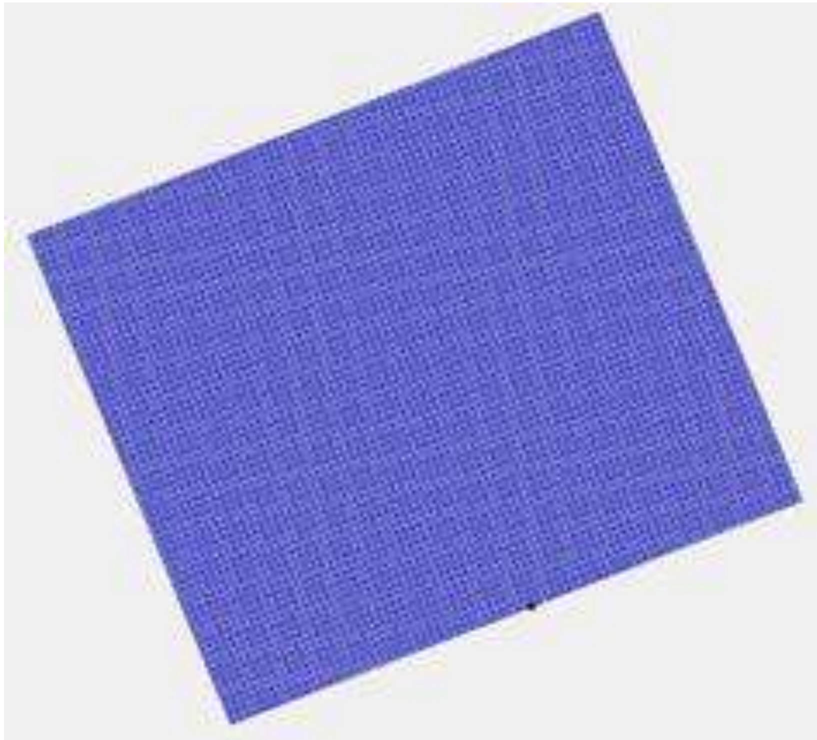
3 Editing the Grid and Running STWAVE

3.1 Generating Spectral Energy Distribution

Next generate the Spectral Energy distribution.

1. Right-click on Map Data in the project explorer and select **New Coverage**.
2. In the *New Coverage* dialog, select the *Coverage Type* as “Spectral” under the “Generic” section.
3. Set the name of the coverage to be "Spectral" and click **OK**.

4. Using the **Create Feature Point** tool, create a node along the middle of the bottom grid boundary as shown in the figure.



5. Use the **Select Feature Point** tool and double-click on the node. The *Spectral Energy* dialog should come up.
6. Click the **Create Grid** button to bring up the *Spectral Grid Attributes* dialog.
7. Set the *Grid angle* to "112.0" degrees. This will match the angle of the STWAVE grid.
8. Make sure that the *Spectral energy grid plane type* is set to "Full - local" and select **OK**.
9. Now the user should see the *Create Spectral Energy Grid* dialog. In the *Frequency Distribution* section of the dialog, change the *Number* to "40" and click **OK** to create a new spectral energy grid.
10. The new spectral energy grid will appear in the Spectral Energy tree control. Select the "Spectral_Grid*"* to see an example displayed in the *Spectral Viewer*.
11. Click the **Generate Spectra** button.
12. In the *Generate Spectra* dialog, enter the following parameters into the spreadsheet.

Time	Angle	Hs	Tp	Gamma	nm
1	25	1	20	8	30

13. This tutorial assumes that the wave gauge is approximately at the offshore edge of the grid. Therefore, set the *Seaward Boundary Depth* for the spectra to “32” m. If the gauge was in deeper water, specify the actual depth of the gauge.
14. Push the **Generate** button. The new spectrum, labeled “1.00000,” will appear below the grid in the Spectral Manager tree control. The "1.00000" represents the time offset from the reference time in hours. The reference time is displayed below the tree control.
15. Select the spectrum “1.00000”. The contours show the energy distribution. Select cell corners to view/edit their energies.
16. Push **Done** to exit the *Spectral Energy* dialog.

3.2 Model Control

In the Model Control, STWAVE inputs can be set. To view the Wind parameters:

1. Select the grid "STWAVE Bounds Grid" in the Project Explorer to make the STWAVE grid active. Users can also select the **Cartesian Grid** module macro.
2. Select *STWAVE* | **Model Control** to bring up the *STWAVE Model Control* dialog.
3. In the dialog, select "Spectral coverage" from the *Boundary condition* source combo box.
4. Select the **Spectral Grid...** button to bring up the *Spectral Grid Properties* dialog.
5. In the *Frequency Distribution* section of the dialog, change the *Number* to “40.”
6. Select **OK**.
7. Select *Half Plane* for the *STWAVE Plane Mode*.
8. Click the **Boundary Control** button to bring up the *Spectral Events* dialog.
9. In the *Spectral Events* dialog, click on the button that corresponds to Side 1 in the *Edge Boundary Type* section. At this time, the button will be labeled “(none selected)”.
10. In the *Select spectral coverage* dialog, select the spectral coverage and click **OK**. This assigns the spectral data contained in the coverage to the boundary.
11. In the *Spectral Events* dialog, click on the **Populate From Coverage** button near the bottom on the dialog. This will create an event for every time entry that is defined in the spectral coverage. In this case, there is one event created with a time of 1.00.
12. This case will not use wind or tide, so click **OK** to close the dialog.
13. Change the *Source Terms* to “Propagation Only” (since the wind was set to zero, this is a redundant setting, but it will speed up the model a little).

14. Click **OK** to exit the dialog.

3.3 Selecting Monitoring Stations

The final step is to select cells to act as monitoring stations:

1. Select the **Select Grid Cell** tool.
2. When selecting a cell, the i and j location can be seen at the bottom of the screen in the status portion of the *Edit Window*. SMS can also select cells by choosing their i and j coordinates.
3. Make sure no cells are selected and choose *Data* | **Find Cell…** to bring up the *Find Cell* dialog.
4. Enter "110" for *I* and "60" for *J* and click **OK**. A cell in the bay should now be selected. Users can also select cells by entering the nearest x and y values or entering the cell ID.
5. Select *STWAVE* | **Assign Cell Attributes…** to bring up the *Cell Attributes* dialog.
6. In the dialog, change the *Cell Type* to *Monitoring Station* and click **OK**.
7. Repeat steps 3 through 6 to assign a monitoring station in the inlet and the ocean. The i and j coordinates for the inlet cell are 92 and 66 respectively, and the i and j coordinates for the ocean cell are 50 and 70 respectively.

3.4 Saving the Simulation

To save the simulation:

1. Select *File* | **Save As**. The *Save As* dialog will appear. Make sure the *Save as type* is set to "Project Files (sms)," and enter the file name "shin_half."
2. Click the **Save** button.

3.5 Running STWAVE

To run STWAVE:

1. Right-Click on the "STWAVE Bounds Grid" in the Project Explorer.
2. Choose **Export STWAVE Files**
3. Right-click on the "STWAVE Bounds Grid" again, and choose **Launch STWAVE**
4. If a message such as "stwave.exe – not found" is given, click the File Browse button to manually find the STWAVE executable.
5. A *STWAVE* dialog will come up showing the progress of the STWAVE run. Click **Exit** when STWAVE finishes. The solution will be loaded into SMS when **Exit** is clicked. The solution will be stored in the solution file

(*shin_half__100m__sol.h5*. The model directory is composed of the project name (shin_half) and the grid name (100m) separated by a double underscore.

SMS has several visualization options to view the solution.

3.6 Visualizing the STWAVE Solution

To see the solution results:

1. Select *Display* | **Display Options** to bring up the Diaplay Options dialog.
2. Under the *Cartesian Grid* tab turn the *Contours* and *Vectors* toggles on.
3. Under the *Contours* tab for the *Contour Method* select "Color fill." Also set the *Transparency* to "20" percent.
4. Under the *Vectors* tab, make sure the *Shaft Length* is set to "Define min and max length" and set the *Min length* to "10" and the *Max length* to "35."
5. Under *Vector Display Placement and Filter,* change the *Display* to "on a grid." Leave the spacing at "30" pixels in each direction and set the *Offset* to "10."
6. Push **OK** to exit the *Display Options* dialog.
7. Select the different scalar and vector datasets of the 100m grid model in the Project Explorer to view their contours and vectors. Zoom into the inlet and pan along the coast to view variations.

Notice that the waves do not cover the entire bay. STWAVE is limited on how fast the waves will spread out after going through the inlet.

3.7 Visualizing the Spectral Energy

The spectral energy is recorded at each monitoring station in the grid frame and written by STWAVE to an observation file (*.obse). The observation file is automatically read in at the end of an STWAVE run. The data in this file is put into a spectral coverage. In this case the coverage is named "100m.obse" or "STWAVE Bounds Grid.obse". To view the spectral energy:

1. Select the coverage "100m.obse" in the Project Explorer to make it the active coverage. Notice that there are nodes at each of the monitoring locations.
2. To view the data in each location, double-click on the desired node using the **Select Feature Point** tool. This will bring up the *Spectral Energy* dialog.
3. Look at the spectral energy at each monitoring station using the *Spectral Viewer*. The ocean station is not much different than the input spectral energy (labeled as *1*). The energy increases in the inlet and changes direction. The energy in the bay is very low compared to the inlet.
4. A user can also look at the spectral energies of the monitoring stations with a current. Notice that the current dampens the energy in the inlet but slightly increases the energy in the bay.
5. When done, click **Done** to exit the *Spectral Energy* dialog.

3.8 Visualizing Current Effects

The next step is to see the effects when a current is added at the inlet from the receding tide. To do this:

1. Select *STWAVE* | **Model Control…** to bring up the *STWAVE Model Control* dialog.
2. Set the *Current interaction* to "Dataset*.*"
3. Click on the **Select** button to the right of *Current Interaction.*
4. In the Select Dataset dialog, select "Velocity (64)_interp" as the vector dataset. Click **Select** to close the dialog.
5. In the *STWAVE Model Control* dialog, click on the **Boundary Control** button to bring up the *Spectral Events* dialog..
6. In the *Events* section, set the time to "1" hour in the first row, and "2" hours in the second row. These will be the snaps that SMS will put the transient current dataset to.

   Note on current dataset: STWAVE simulates one or more boundary condition scenarios referred to as "snaps." In the original run, SMS used a single snap that represented a specified boundary condition. For this model run, SMS is going to run two snaps one at hour 1 and the other at hour 2. When using snaps based upon times, the user provides a current dataset that is transient. SMS will have STWAVE use the current dataset values for the time specified in the *Boundary Control* dialog. If the snap time step is between two time steps in the current dataset, SMS will interpolate values. If the snap time step is before all time steps in the current dataset, SMS will use the values from the first time step. Similarly if the snap time step is after all time steps, SMS will use the values in the last time step. If STWAVE is not using times and is instead representing independent boundary conditions, a steady state current dataset is chosen for each snap. The surge and wind datasets work similarly.
7. Save the project as "shin_curr.sms" and rerun STWAVE as previously done.
8. Select the different scalar and vector datasets of this simulation to view the contours and vectors. If viewing the current field, notice the current is a fairly strong ebb current. Notice the difference in wave height that the current makes to the results.

4 Re-running ST-WAVE in Full Plane Mode

Now to see the effects of running the STWAVE module in the full-plane mode.

4.1 Open the Previous Half Plane Simulation

By opening the previous solution, the tutorial avoids repeating the grid set up. To open the half plane simulation:

1. Select *File* | **Open** and open the file “shin_half.sms” that was saved previously. Click **Yes** to delete the current simulation. The grid should appear on the screen.
2. Select *STWAVE* | **Model Control** to open the STWAVE Model Control dialog.
3. In the dialog, set the *STWAVE Plane Mode* option to *Full Plane*. Check to ensure that the *Source terms* are still set to “Propagation Only.”
4. Click **OK** to exit the *STWAVE Model Control* dialog.

4.2 Save and Run STWAVE in Full Plane Mode

To save the simulation:

1. Select *File* | **Save As**. The *Save As* dialog will open. Make sure the *Save as type* is set to “Project Files (sms),” and enter the file name “shin_full.”
2. Click the **Save** button.
3. Run STWAVE in the same manner as for half-plane. These steps are found in section 3.5 of this tutorial.
4. Click **Exit** when the model finishes.

4.3 Visualizing the Full Plane Solution

Repeat step 3.6 to visualize the STWAVE solution for the full plane option. Notice that the direction of waves in the back bay is much more varied in the full plane option.

5 Conclusion

This concludes the STWAVE Analysis tutorial. The model contains many more features and capabilities that have not been explored in this document. Refer to the STWAVE Users manual and SMS help file found in *Help* | **SMS Help…** for more information.