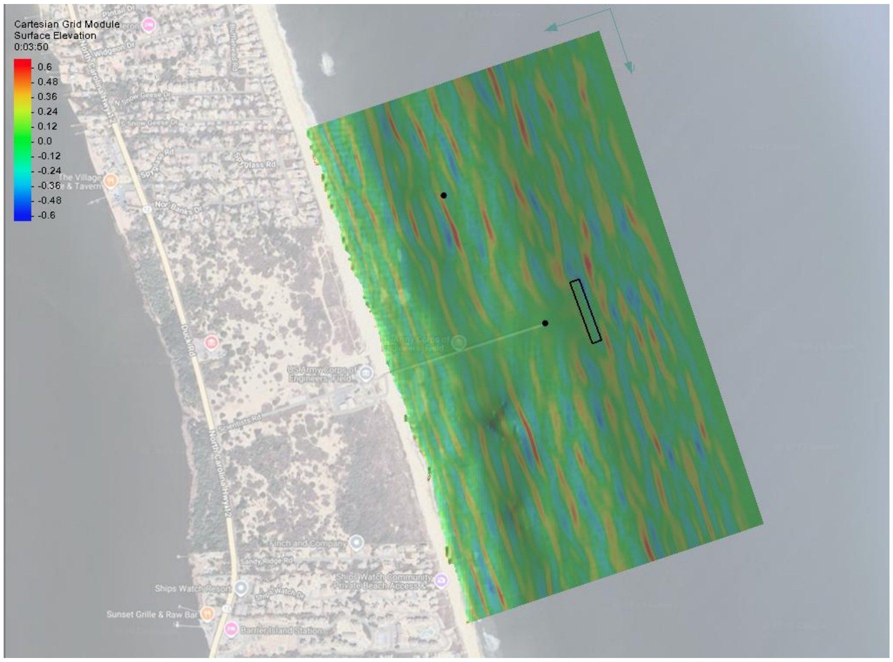
SMS 14.0 Tutorial

FUNWAVE

Overview of the FUNWAVE Interface in SMS



Objectives

Learn how to set up a FUNWAVE simulation and export the simulation files.

Prerequisite Tutorials

- SMS Overview
- Grid Generation

Required Components

- SMS Core

Time

- 35–45 minutes

1 Introduction

This tutorial covers the steps of creating the data files for FUNWAVE-TVD® (FUNWAVE) simulations in the Surface-water Modeling System (SMS).

FUNWAVE is a fully nonlinear phase-resolving Boussinesq wave model for modeling of surf zone–scale optical properties in a Boussinesq model framework, and modeling of Tsunami waves in both a global/coastal scale for prediction of coastal inundation and a basin scale for wave propagation.

A FUNWAVE simulation consists of the following components:

- A Cartesian grid
- A coverage defining output stations
- Datasets as needed for spatial inputs
- A set of model parameters

The approach used for the definition of this tutorial includes using the WaterWorks On Demand (WWOD) platform. This requires an account on WWOD, which has an installation of FUNWAVE. Browse to https://taylorengineering.com/project/usace-model-modernization-waterworks-ondemand-platform/ for more information.

This tutorial begins with a bathy/topo DEM of the area around the USACE Field Research Facility (FRF) at Duck, North Carolina, a background image, and annotation data.

2 Getting Started

Start by importing a project file containing background data:

1. Launch the SMS application.
2. From the *Menu* bar, select the *File* | **Open…** menu item to bring up the *Open* dialog.
3. From the *Files of type* drop-down, select “Project Files (*.sms)”.
4. Browse to the *data_files* folder for this tutorial and select “start.sms”.
5. Click the **Open** button to import the project and exit the *Open* dialog.

The project should appear similar to Figure 1.

Note that the project includes a display projection set to UTM Zone 18. FUNWAVE requires a Cartesian projection.

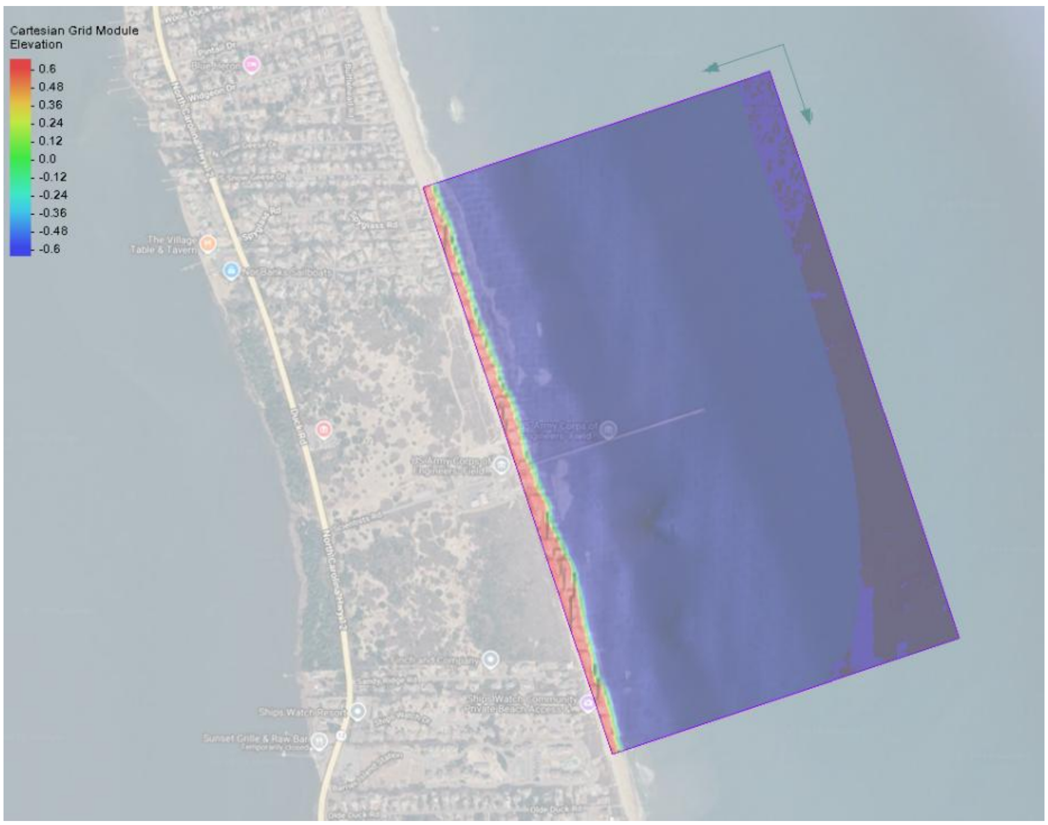


Figure 1 Initial project

2.1 Saving a Project File

Save the project with a new name.

1. Click the *File* | **Save As…** menu command to bring up the *Save As* dialog.
2. For the *File Name*, enter “existing.sms”.
3. Click the **Save** button to save the file under the new name and close the *Save As* dialog.

3 Create the Output Stations

It is the responsibility of the modeler to define where extra output is desired from the FUNWAVE simulation. This is specified as a set of output stations (points). The data saved by FUNWAVE at these locations can be used to evaluate wave conditions at a single location. They can also be used in conjunction with other stations to compute a variety of information such as wave run up.

3.1 Create an Output Station Coverage

FUNWAVE output points can be created in a “FUNWAVE: Output Stations” coverage. To create a coverage for this purpose:

1. In the Project Explorer, right-click on the “ Map Data” folder and select the **New Coverage** context menu item to bring up the *New Coverage* dialog.
2. In the *Coverage Type* section, open the “Models” entry and select “FUNWAVE: Output Stations”.
3. For the *Coverage Name,* enter “Output Stations”.
4. Click the **OK** button to close the *New Coverage* dialog and create the coverage.

3.2 Create the Station Points

To create output stations:

1. In the Project Explorer, select the “ Output Stations” coverage to make it active.
2. Select the **Create Feature Point** dynamic tool and click near the locations shown in Figure 2.

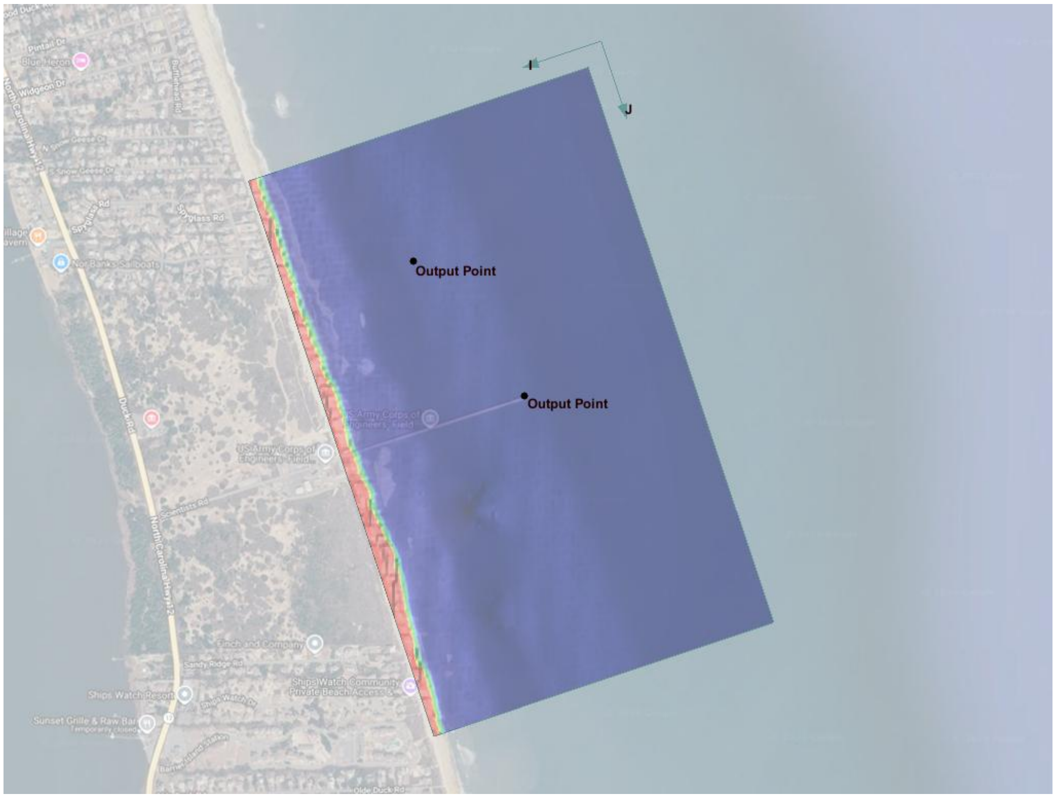


Figure 2 Output stations in the domain

4 Assemble the Simulation

An SMS project can include multiple simulations. Each simulation combines a grid with model parameters.

4.1 Create the Simulation

The first step in defining a simulation is creating a blank simulation to hold the components. To do this:

1. In the Project Explorer, in a blank area, right-click and select the *New Simulation* **| FUNWAVE** context menu item.
2. The " Simulation Data" folder appears and populates with the " FUNWAVE Simulations" model simulation folder and the " Sim" simulation item.
3. Right-click on the " Sim" simulation item, select the **Rename** context menu item.
4. Rename the simulation to "existing" and press the *Enter* key to set the new name.

4.2 Add Components to the Simulation

The simulation is assembled by adding components to it. To do this:

1. In the Project Explorer, from the " Cartesian Grid Data" folder, drag the " 2m grid" grid dataset and drop onto the " existing" simulation. A black drop location line will appear.
2. From the " Map Data" folder, drag the " Output Stations" map coverage and drop onto the " existing" simulation.

Note: Components can also be added to a simulation by right clicking on the component, selecting "Apply to" and selecting the desired simulation.

4.3 Setting Model Parameters

The run options or model parameters are assigned to the simulation. The interface sets most parameters to a default. Only modified parameters are discussed here:

1. In the Project Explorer, right-click on " existing" and select the **Model Control...** context menu item. The *Model Control* dialog will open.

4.4 Setting Project Parameters

1. Select the *Project* tab. The associated parameters appear on the right.
2. For the *Project title,* enter "FRF existing conditions".
3. For *Number of processors to use in X (PX),* enter 5.
4. For *Number of processors to use in Y (PY),* enter 8.

Note: For this case the values of PX and PY are determined to make approximately square subdomains while limiting the number of compute threads to less than 44. A single compute node on WWOD has 44 threads.

4.5 Setting Time Parameters

For this tutorial, the peak wave period is 12.5 seconds. The total time would normally be at least 50 wave cycles, because of Nyquist sampling theory. For speed, we will only use 20 cycles for this tutorial. The other determining input is the desired number of timesteps for evaluating the wave conditions. For this simulation, we be outputting instantaneous data for the last 2 wave cycles (25 seconds) with sampling rate of 1/10th a wave cycle (20 timesteps).

1. Select the *Time* tab. The duration of the simulation is principally controlled by the input wave conditions.
2. For the *Simulation time in second (TOTAL_TIME),* enter "250.0" seconds. (20 wave cycles x 12.5 seconds = 250.)
3. For the *Output interval in seconds (PLOT_INTV),* enter "1.25" seconds. This is 1/10th of the wave period.
4. For the *Time interval of screen print (SCREEN_INTV),* enter "2.5" seconds. This is the frequency with which FUNWAVE outputs diagnostic information. It could match the plot interval, or as in this case, a little less frequent. It is still frequent enough to give multiple outputs for each wave cycle.
5. For the *Time interval of gauge output (PLOT_INTV_STATION),* enter "0.25" seconds. This is the higher frequency output for the stations. It should not be less than 0.1 seconds for stability.

6. For the *Output field start time in seconds (PLOT_START_TIME),* enter “225.0” seconds. This is 2 wave cycles (25 seconds) before the end of the simulation, allowing us to review instantaneous data for the last 2 cycles.

4.6 Setting Geometry Parameters

1. Select the *Geometry* tab.
2. For the *Depth dataset (DEPTH_FILE),* click the **Select** button to open the *Select Dataset* dialog.
3. Turn on the “Elevation” dataset checkbox.
4. Click the **OK** button to close the *Select Dataset* dialog.

4.7 Setting Numerical Parameters

1. Select the *Numerical Parameters* tab.
2. For the *Minimum water depth (m) for wetting and drying scheme (MinDepth),* enter “0.01”. This allows for more precision.

4.8 Setting Wave-maker Parameters

1. Select the *Wave-maker* tab.
2. From the *Wave-maker type (WAVEMAKER)* drop-down menu, select “Wei and Kirby 1999 TMA spectrum wavemaker (internal) (WK_IRR)”.
3. For the *Time ramp (s) (Time_ramp),* enter “1.0”.
4. For the *Width parameter δ (Delta_WK),* enter “1.0”.
5. For the *Water depth (m) (DEP_WK),* enter “8.0”.

This is the approximate depth of water at the wave maker and the maximum depth on this edited grid.

6. For the *X coordinate (m) (Xc_WK),* enter “160” (2 wave lengths in from the edge).
7. For the *Y coordinate (m) (Yc_WK),* enter “800” (half the grid height).
8. For the *Peak frequency (1/s) (FreqPeak),* enter “0.08” (a 12.5 second wave).
9. For the *Low frequency cutoff (1/s) (FreqMin),* enter “0.03”.
10. For the *High frequency cutoff (1/s) (FreqMax),* enter “0.3”.

Note: this corresponds to a period range from 0.33 – 33.3 seconds with the peak period at 12.5 seconds.

11. For the *Hmo (m) (Hmo)*, enter “1.0”. This indicates that the average wave height generated will be 1.0 m.

4.9 Setting Boundary Conditions Parameters

1. Select the *Boundary Conditions* tab.
2. Turn on the *Enable periodic boundary condition in the y-direction (PERIODIC)* checkbox. This indicates that oblique wave energy that exits the top or bottom of the domain will enter at the opposite end of the domain.

4.10 Setting Sponge Layer Parameters

1. Select the *Sponge layer* tab.
2. Turn on the *Enable L-D type sponge (DIRECT_SPONGE)* checkbox.
3. Turn on the *Enable friction type sponge (FRICTION_SPONGE)* checkbox.
4. For the *Maximum Cd (CDsponge),* enter “1.0”.
5. For the *Width (m) of sponge layer at west boundary (Sponge_west_width),* enter “100.0”.

In this case, the directions refer to local coordinates for the grid, so west is behind the wave maker. Adding a sponge width here damps the waves coming from the back of the wave maker.

4.11 Setting Output Parameters

1. Select the *Output* tab.
2. Turn on the *Enable ETA output (ETA)* checkbox.
3. Turn on the *Enable Velocity output (U and V)* checkbox.
4. Turn on the *Enable output of wetting-drying Mask (MASK)* checkbox.
5. Select the **OK** button to save these settings and close the *Model Control* dialog.

5 Running the FUNWAVE Simulation

Export the simulation from SMS, upload to WWOD, execute FUNWAVE on WWOD, and import the FUNWAVE simulation results into SMS.

This tutorial includes a solution file that can be copied into SMS. Described in the *Loading the FUNWAVE Solution into SMS* section.

5.1 Exporting Simulation Files

SMS has now gathered all the information related to a FUNWAVE simulation. We now need to tell SMS to save the input files for the analysis. To do this and review the files:

1. From the *Menu* bar, select the *File* | **Save Project** menu item.
2. Right-click on the " existing" simulation and select the **Save Simulation** context menu item.
3. In the Project Explorer, right-click on the " Project" folder and select the **Open Project Folder** context menu command. This will open an operating system File Explorer browser window in the location of the files, for this tutorial the “FUNWAVE_Intro\data_files\” folder.
4. In the *File Explorer* window, opened in the previous step, browse to the "existing_models\FUNWAVE\existing" folder.

5.2 Renaming the Input File

1. Right-click on the “input.funwave” file and select the **Copy** context menu command.

2. Right-click on a blank space in the *File Explorer* window and select the **Paste** context menu command.
3. Right-click on the “input – Copy.funwave” file and select the **Rename** context menu command.
4. Rename the file to “input.txt” and press the *Enter* key to set the new name. FUNWAVE requires the input file to be named like this.
5. In the rename confirmation dialog, click the **Yes** button.

5.3 Copying the Submit File

1. From the *data_files* folder for this tutorial, copy the “submit.pbs” file.
2. In the *data_files\existing_models\FUNWAVE\existing* folder, paste the “submit.pbs” file. This is a bash script that will be used to run the simulation.

5.4 Using WaterWorks On Demand (WWOD)

The approach used for this tutorial includes using the WaterWorks On Demand (WWOD) platform https://taylorengineering.com/project/usace-model-modernization-waterworks-ondemand-platform/. This requires an account on WWOD, which has an installation of FUNWAVE.

5.5 Upload Simulation Files to WWOD

Upload the simulation files created by SMS to WWOD.

1. Log into WWOD.
2. From the *WWOD* dashboard, click on the *Files* link at the top of the screen and select the **Home Directory** option.
3. Use the **New Directory** button to create a directory or select an existing directory to perform the FUNWAVE run.
4. Click on the **Upload** button to bring up the upload dialog.
5. Click on the **Browse folders** link. This will open a *windows file selector* dialog.
6. Navigate to the FUNWAVE simulation folder for the project on the local machine. For this tutorial, the “data_files\existing_models\FUNWAVE” directory.
7. Select the “existing” folder and click the **Upload** button.
8. Click the **Upload files** button to upload the files to WWOD.

5.6 Starting Linux Desktop on WWOD

This section describes how to execute FUNWAVE on the WWOD system.

1. In the WWOD dashboard, click on the *Apps* drop-down at the top of the screen and select *All Apps.*
2. Scroll through the entries and select “Linux Desktop.” This will open the setup window.
3. For the *Session Target,* select “With GPU”.
4. For the *Maximum duration in hours of your remote session*, for the purposes of this tutorial enter “8”. For most simulations, this will need to be longer.

5. Click the **Launch** button.
6. WWOD will open a status list of interactive sessions with the code server app showing a status of *Queued.*
7. Wait for the notice to change from *Queued* to *Running.* When this happens click the **Launch Linux Desktop** button that appears. This will launch a new web browser tab for the virtual server.

5.7 Executing FUNWAVE on WWOD

1. From the dock at the bottom of the display, use the *Application Finder* to launch a new *Terminal Emulator*. This will bring up a terminal window.
2. Navigate to the directory containing the FUNWAVE input files. For example, enter “cd FUNWAVE/existing”.
3. Enter the command “qsub submit.pbs” and press the *Enter* key to run the script. This will launch the FUNWAVE simulation. This simulation will take a few minutes to run on WWOD. For this first simulation, WWOD may take a few minutes longer to launch the first process.
4. To monitor the progress of the simulation, use the “qstat” command. There will be a job with the name of “funwave” in the list. Rerun the “qstat” command from time to time. The simulation is done when the *funwave* job disappears from the list.

The *submit.pbs* script has a few parameters set for this tutorial that should be changed for other simulations. This includes the *max walltime* on line 6 and the *N_PROCS* variable on line 11.

On WWOD, *max walltime* refers to the maximum amount of real-world clock time that the job is allowed to run on the compute-node before termination. Since this tutorial is small and light this is set to 8 hours. A larger simulation may need a longer time period, though this may increase queue wait time.

On WWOD, *N_PROCS* refers to the number of processor cores that can be used in parallel to run the simulation. This is set in the Model Control dialog, by entering values for PX and PY. For this case, the values of PX (5) and PY (8) are determined to make approximately square subdomains and multiplying (5 * 8) = 40 them together while limiting the number of compute threads to less than 44, a single compute node on WWOD. Using more cores than a node has can slow performance due to communications overhead.

5.8 Copy Solution to Local Machine

After the FUNWAVE simulation is complete, the solution files will need to be downloaded back to the local machine. To do this:

1. From the *WWOD* dashboard, click on the *Files* drop-down menu at the top of the screen and select “Home Directory”.
2. Browse to the directory the simulation was uploaded to. For this tutorial, “FUNWAVE”.
3. To the left of the “existing” folder, turn on the checkbox.
4. Click the **Download** button at the top of the panel. This compresses the folder as a ZIP file and downloads the file.

5.9 Loading the FUNWAVE Solution into SMS

To load the solution computed by FUNWAVE for review in SMS. This solution includes field data for grid as well as point data at the stations.

Copy the solution files to the simulation directory.

1. In the *File Explorer*, browse to the *Downloads* folder and locate the “existing.zip” file. If needed, an already completed solution file is available in the “FUNWAVE_Intro\Solution” folder.
2. Right-click on the “existing.zip” file and click on the **Extract All…** context menu command.
3. Copy all of the contents from the solution folder and paste into the simulation folder created by SMS. For this tutorial, from the “Downloads\existing” folder to the “data_files\existing_models\FUNWAVE\existing” folder.

To load the solution into SMS:

4. In the Project Explorer, right-click on the “existing” simulation and select the **Read Solution** context menu item.
5. The *Loading FUNWAVE Solution Files* dialog appears as the files are read into SMS.
6. A *Solution* folder is created under the existing simulation.
7. The *Time steps* window also appears.

The solution consists of water surface elevations and wave velocities (directions) for 20 timesteps at the end of the simulation. These timesteps start at 225 seconds (3:45 minutes) into the simulation and are saved about every 1.5 seconds by FUNWAVE.

The solution also includes time series for the entire simulation at the output stations with a time interval of about 0.25 seconds.

6 Field datasets

The file data is visualized using contour options on the grid in SMS. Earlier in this tutorial, we used the same options to visualize the bathy/topo data for the site.

To examine the field data for this solution:

1. Select the *Display* | **Display Options…** menu command to open the *Display Options* dialog.
2. From the list on the left, select the *Cartesian Grid* option
3. On the *Cartesian Grid* tab, turn off the *Cells*, *Cell id*, and *Cell ij* checkboxes.
4. Turn on the *IJ triad* checkbox.
5. Turn on the *Contours* checkbox.
6. On the *Contours* tab, in the *Contour method* section, from the drop-down menu select *Color fill.*
7. For the *Transparency,* enter “60” %.
8. Click the **OK** button to close the *Display Options* dialog.
9. In the *Time steps* window, step through the various time steps and see the changes in the *Graphics Window* which should look similar to Figure 3.

Experiment with different contour ranges in the *Display Options* dialog. (Note the range of water elevations.)

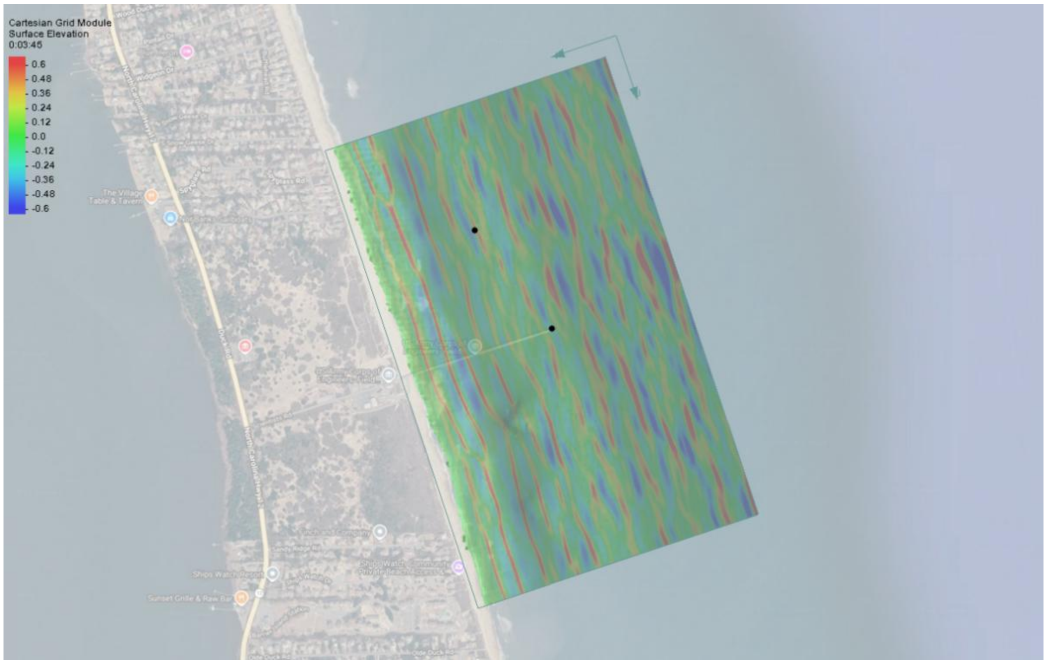


Figure 3 FUNWAVE Output Stations solution

6.1 Station data

To view the data at a station:

1. In the Project Explorer, make the *Output Station* coverage active.
2. Using the *Select Feature Point* dynamic tool, double click on one of the output station points.
3. The *FUNWAVE Solution Plots* dialog appears.
4. From the list on the left, in the *Plots* section, turn on the *WSE_m* plot checkbox. The plot appears under the *Plot* tab of the right side of the dialog. The dialog is resizable, refer Figure 4.
5. On the right side of the dialog, select the *Table: PT1 – existing/output* tab and review the data values in the spreadsheet. Note how long it takes for the wave to get to the selected point, *WSE_m* is close to zero initially.
6. Click the **Close** button to close the *FUNWAVE Solution Plots* dialog.

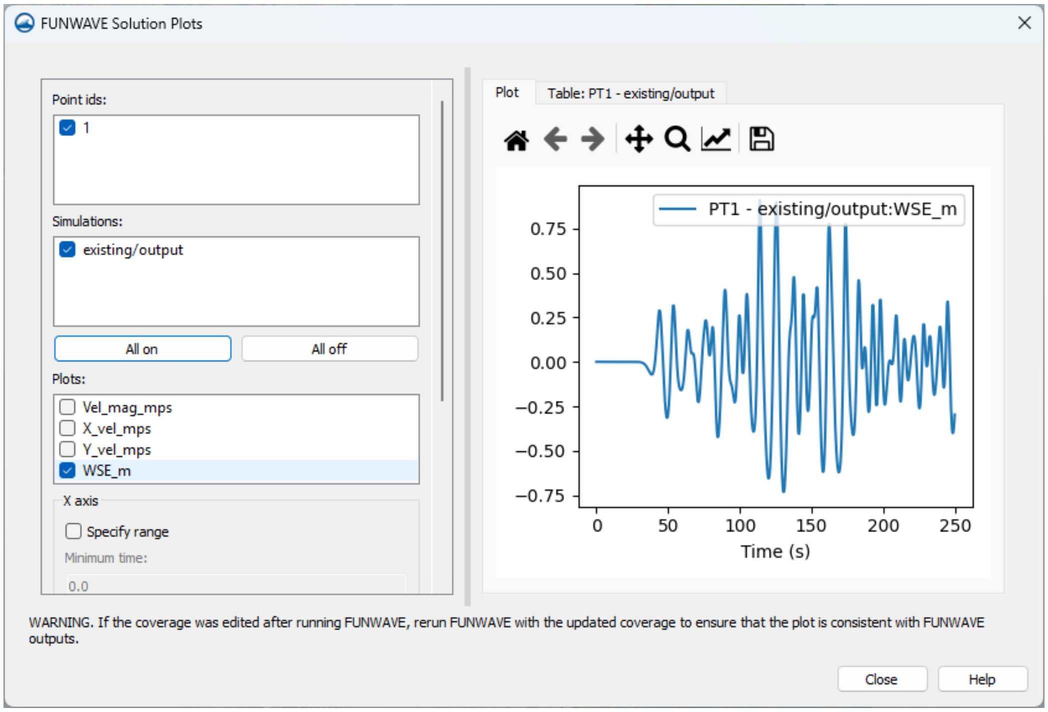


Figure 4 FUNWAVE Solution Plots dialog

Both points may be selected and then right-clicked on to bring up the plot dialog and compare the water level at different locations.

7 Conclusion

In this tutorial demonstrated the setup of a FUNWAVE simulation and exported the files for the analysis. The tutorial covered the following:

- Defining FUNWAVE boundary conditions and model parameters
- Creating and assembling the FUNWAVE simulation
- Exporting the FUNWAVE files
- Running the FUNWAVE model on WWOD
- Importing the FUNWAVE solution into SMS
- Reviewing FUNWAVE results

Documentation for the USACE version of FUNWAVE can be found at https://fengyanshi.github.io/build/html/index.html#.

