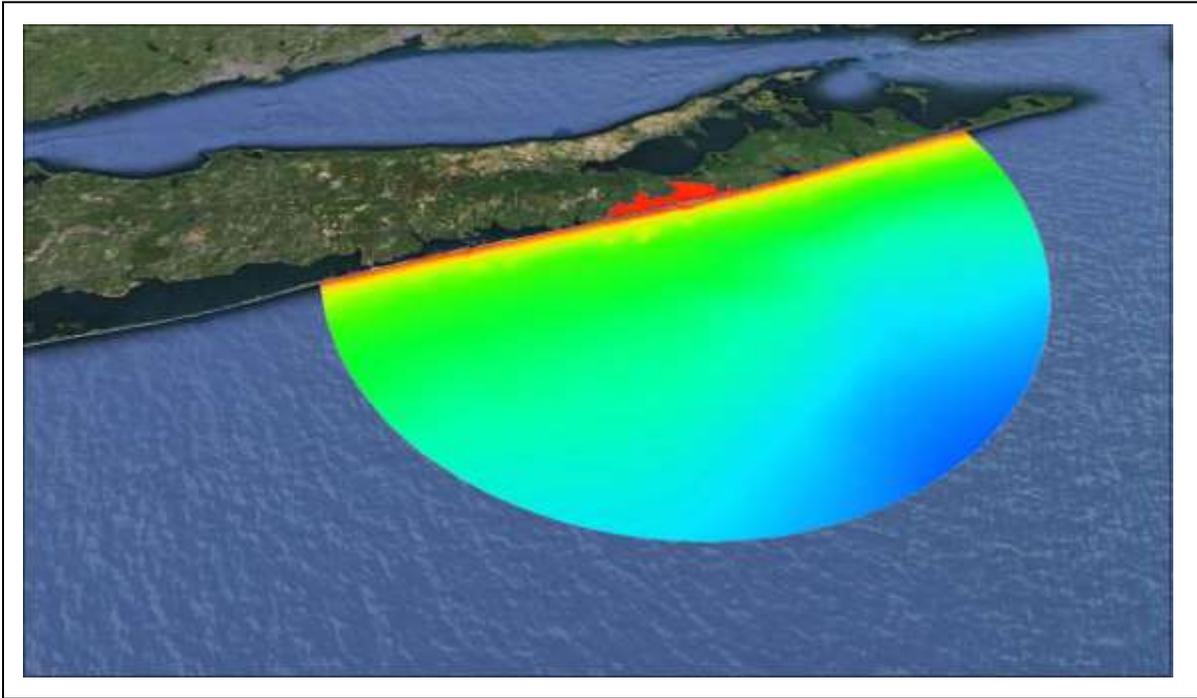


SMS 11.2 Tutorial

ADCIRC Analysis



Objectives

This lesson reviews how to prepare a mesh for analysis and run a solution for ADCIRC. It will cover preparation of the necessary input files for the ADCIRC circulation model and visualization of the output. It will start by reading in a coastline file and then a SHOALS file.

The data used for this tutorial are from Shinnecock Bay off of Long Island in New York. All files for this tutorial are found in ADCIRC Data Files directory.

Prerequisites

- Overview Tutorial

Requirements

- ADCIRC
- Map Module
- Mesh Module
- Scatter Module
- LeProvost Tidal Database

Time

- 60-90 minutes

AQUAVEO™



1 Reading in a Coastline File

For this tutorial, first read in a coastline file, which has already been set up for. This sample coastline will form the boundary for the mesh. To set up the coverage for ADCIRC and open the coastline file:

1. Change the coverage type to ADCIRC by right-clicking on the “area property” coverage, selecting *Type*, and choosing *Models | ADCIRC*.
2. Right-click on the “Area Property” coverage and select **Rename**. Give the coverage the name “coastline”.
3. Select *File | Open* to bring up the *Open* dialog.
4. Select the file “shin.cst” in the Data Files Folder for this tutorial and click the **Open** button.

Coastline files include lists of two-dimensional polylines that may be closed or open. The open polylines are converted to feature arcs and are interpreted as open sections of coastline. Closed polylines are converted to arcs and are assigned the attributes of islands.

1.1 Defining the domain

First, it’s necessary to assign a boundary type to the coastline arc. Then the user can define the region to be modeled. To do this:

1. Make sure the Map  module is active, if not already selected. Make certain the “coastline” coverage is still set as ADCIRC .
2. Choose the **Select Feature Arc**  tool from the toolbar and click on the coastline arc to select it.
3. Select *Feature Objects | Define Domain*. The *Domain Options* dialog should appear.
4. In the dialog, select the *Semi-circular* option and click **OK**.
5. **Frame**  the display.

A semi-circular arc is created to define the region.

1.2 Assigning Boundary Types

Boundary types for arcs are specified in the Map module. Boundary types are prescribed by setting attributes to feature arcs. To set the boundary types:

1. Choose the **Select Feature Arc**  tool.
2. Double-click the arc representing the ocean boundary, shown in Figure 1.

3. In the *ADCIRC Arc / Nodestring Attributes* dialog, assign this arc to be of type *Ocean*.
4. Click the **OK** button to close the dialog.

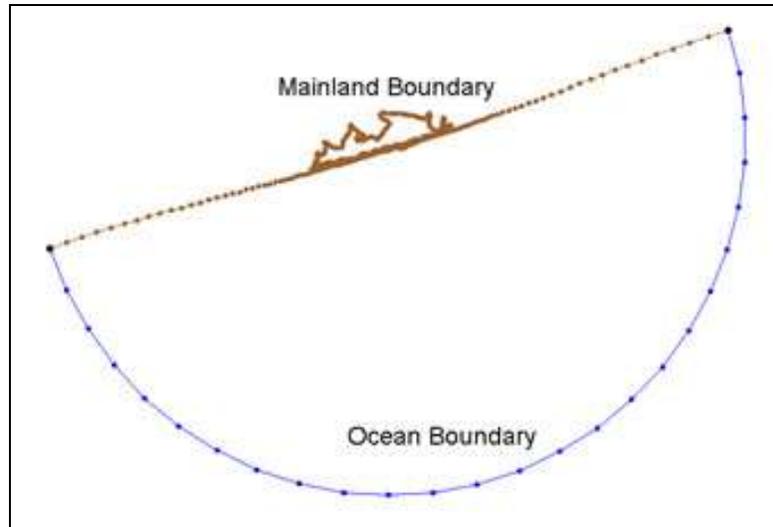


Figure 1 Feature arcs after boundary types have been assigned

*Note: When the Coastline file is read into a coverage that is already set as a ADCIRC type, it will automatically be read in as a Mainland boundary arc. If the coverage is set to ADCIRC after reading in the coastline file, the user will have to double-click on the coastline arc and set it to be a mainland boundary.

2 Editing the Coastline File

Now that the coastline file has been read in and a corresponding map object created, several modifications must be made to the data before the SHOALS file is read in. Since the SHOALS file is in UTM coordinates, zone 18(78W to 72 W northern Hemisphere), the data should also be set to those coordinates. In order to do this the display projection must be set to same global projection. To convert the display coordinates:

1. Choose *Display / Projection...* to bring the *Display Projection* dialog.
2. In the *Display Projection* dialog, toggle on *Global projection* to bring up the *Select Projection* dialog or click **Set Projection** if the dialog doesn't automatically appear.
3. In the *Select Projection* dialog, set the *Projection* to "UTM", and set the *Zone* to "18 (78W to 72W – Northern Hemisphere)" and the *Datum* as "NAD 27". When done, click **OK**.
4. Ensure that the *Vertical Units* is set to "Meters".
5. Click the **OK** button to exit the dialog.

Next, to set the Map object coordinates:

6. Right-click on the "coastline" map object and select **Projection**. The *Object Projection* dialog should appear.
7. Toggle on the *Global Projections* option then click **Set Projection**.
8. In the *Select Projection* dialog set *Projection* to "Geographic (Latitude/Longitude)" then click **OK**.
9. Click **OK** to close the *Object Projection* dialog.
10. Right-click the "coastline" coverage again and select **Work in Object Projection**.
11. Right-click the "coastline" coverage again and select **Reproject**.
12. Click **Yes** if a warning appears about round-off errors.
13. In the *Reproject Object* dialog, the *Current projection* should be set to "Geographic (Latitude/Longitude), NAD27, arc degrees". The *New projection* should be set to "UTM, Zone: 18 (78°W – 72°W – Northern Hemisphere), NAD27, meters". If this is not correct in either case, use *Set* toggle and the **Set Projection** button to correct the projections in the *Select Projection* dialog that will appear. Click **OK**.
14. Click the **Frame** macro to see the points.

The coastline data has now been converted to a global projection of UTM coordinates with zone 18 (78w to 72W--Northern Hemisphere) and Datum set at NAD27. However, the display projection is now in Geographic (Latitude/Longitude). Before reading in the SHOALS data it's necessary to change it back to UTM. To do this:

15. Right-click on the "coastline" coverage and select **Work in Object Projection**.

3 Reading in a SHOALS File

Now to read in a SHOALS file: "shin.pts". This file contains data at various locations along the coastline and throughout the region being modeled.

1. Choose *File* | **Open** to bring up the *Open* dialog.
2. Select the file "shin.pts" and click **Open**.
3. In the *Open File Format* dialog, toggle on *Use Import Wizard* and click **OK**. The *File Import Wizard* dialog will open allowing specifications on how the data will be read into SMS.
4. For Step 1 of the dialog, make certain the *Space* option is toggled on. If the *Space* toggle is on for this file, then the first line in the *File preview* box is the file header. The next line shows the name of each respective column of

data. In this case, the file has three data columns. The first column is the *X* Coordinate, the second column is the *Y* Coordinate, and the third column is the *depth/bathymetry*.

5. Click the **Next** > button to move on to Step 2 of the *File Import Wizard*.
6. The second step of the *File Import Wizard* allows changing other specifications for reading in the SHOALS file. For now, accept the defaults in this step and click the **Finish** button.

Figure 2 shows the plot of the points read in from the “shin.pts” file. (If the scatter set does not appear at first, make sure that *Points* is toggled on in the *Display Options* under *Scatter*).

Once the scatter set is read in, set its projection. To do this:

1. Right-click on the “shin” scatter set in the project explorer and select **Projection**.
2. In the *Object Projection* dialog, and toggle on *Global projection*. The projection will already be set to UTM since the *Display Projection* is currently set to UTM. If not, click on the **Select Projection** button to change the projection to UTM. When done, click **OK** to exit the dialog.

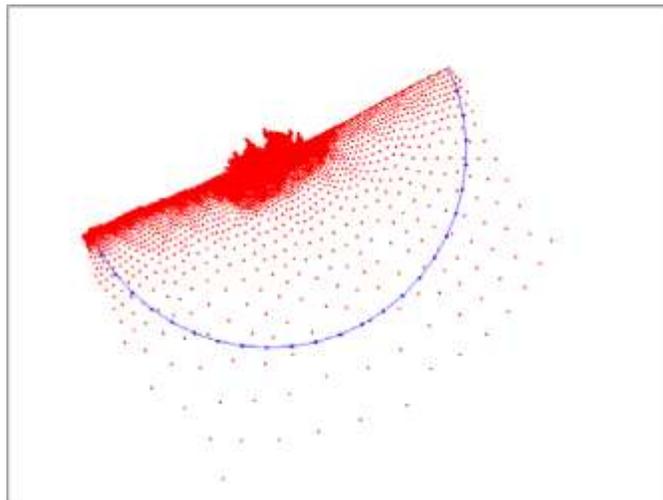


Figure 2 Display of “shin.pts”

4 Shallow Wavelength Functions

The next step before building a finite-element mesh is to create several functions for creating the finite element mesh. For this tutorial, the mesh will be generated according to the wavelength at each node. Large elements will be created in regions of long wavelengths. Conversely, smaller elements are needed closer to the shore to correctly model the smaller wavelengths.

To create this shallow wavelength function from the bathymetric data:

1. In the Scatter  module, select *Data* / **Dataset Toolbox**.
2. In the *Dataset Toolbox* dialog, select the *Wavelength and Celerity* tool in the *Coastal* section. This enables the wavelength options on the dialog.
3. Make sure the options for creating a *Wavelength* and *Celerity* function are checked. Leave the Period at "20.0" seconds and enter a name "20 sec" in the *Output base name* edit field.
4. Click the **Compute** button to create the datasets and the **Done** button to close the dialog.

Two functions are created: celerity and wavelength at each node using the shallow water wavelength equation. The celerity is calculated as:

- $Celerity = (Gravity * Nodal\ Elevation)^{0.5}$.

The wavelength is calculated as:

- $Wavelength = Period * Celerity$.

5 Creating Size Functions

Now having created the wavelength function, there are a few more conversions before creating the mesh. A size function is a multiple that guides the size of elements to be created in SMS. Any dataset may be used for this purpose. If generating a mesh using the original wavelength function alone, SMS would create a decent mesh to work with, but this tutorial requires a mesh whose density radiates out from a point in the inlet. This allows more accurate results in the inlet for the outcome of the ADCIRC run. Therefore, it's necessary to create a size function based on the wavelength to attain this end. The final size function used for modeling applications varies and is found through trial and error to give a nicely formed mesh. This example illustrates one method of building a size function.

5.1 Finding the Central Point for the Mesh

Since the mesh will be generated in a radial fashion, the distance from a central point must be found. The first step is to locate the central point and then use the *Data Calculator* to compute the distances of all points from this center point. To do this:

1. Still in the Scatter  module, zoom in to the area of the inlet shown in Figure 3 with the **Zoom** tool until the screen looks like Figure 4.
2. Click on one of scatter points in the middle of the inlet using the **Select Scatterpoints**  tool. Make note of this point's X and Y coordinates in the Edit Window at the top of the screen.
3. Frame the data by clicking the **Frame**  tool.

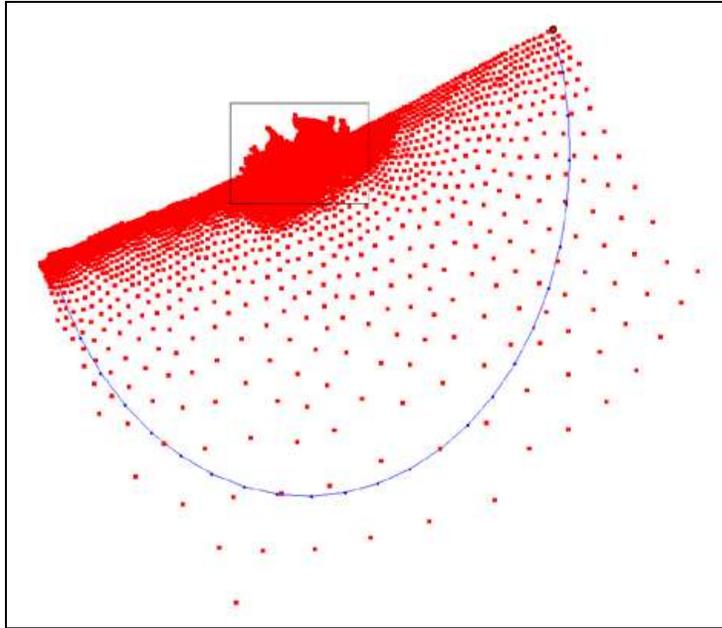


Figure 3 Inlet location to zoom in on

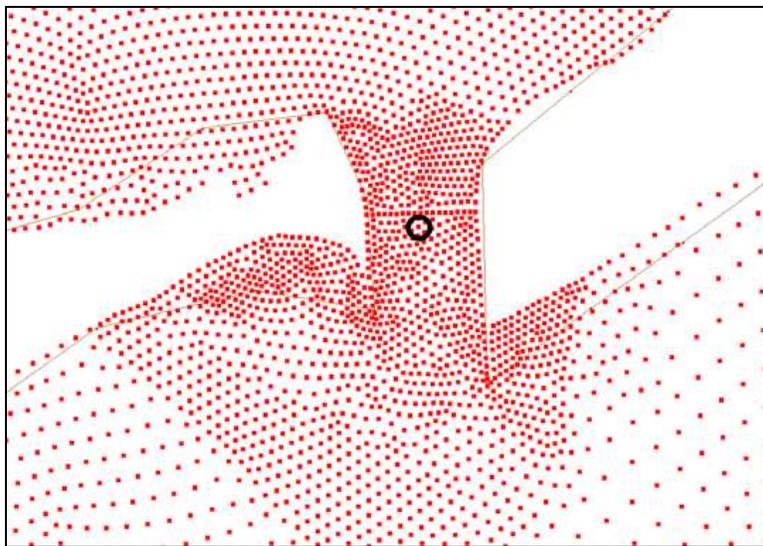


Figure 4 Choose a center point

For now, turn off the scatterpoint display. However, users may turn it back on at any time during the tutorial if desired. To turn off the visibility of the “shin.pts” data unselect the box next to the “shin” dataset in the Project Explorer.

Next to use the *Data Calculator* to compute new datasets by performing operations with scalar values and existing datasets. The *Data Calculator* will be used to create the size function.

5.2 Distance Function

For consistency, use the (x,y) location of (712768.675, 4523969.712) as the center scatterpoint for the mesh.

1. Select *Data* | **Data Calculator....** This brings up the *Dataset Toolbox*, with the data calculator active.
2. Click the **sqrt** button.
3. In the *Expression* field, using the keyboard replace “??” so the expression looks like:
 - $\text{sqrt}((d4 - 712768)^2 + (d5 - 4523950)^2)$

This expression takes the *x* and *y* locations of each scatter point, which correspond to the “d4” and “d5” datasets respectively, and computes its distance to the point designated as the mesh center.

4. In the *Output dataset name* field, enter the name of “distance” for the dataset and click the **Compute** button.

5.3 Initial Size Function

1. Highlight the “20 sec_Wavelength” dataset and click the **Add to Expression** button. The letter and number “d2” should appear in the *Expression* field.
2. In the *Expression* field, make the equation look like “d2*7”.
3. Enter the name “size” for this dataset in the *Output dataset name* area and click the **Compute** button. This creates a function of 7 times the wavelength.

5.4 Scale Function

The last separate function before computing the final size function will be a scale factor out from the center point. It will take on the following format:

- $\text{scale} = (\text{distance}/\text{max distance})^{0.5}$.

This scale function will range between 0 and 1, 0 being at the center point and 1 at the farthest point from the center of the mesh. This will allow the mesh to radiate out in density from the middle of the inlet. Taking the square root of the scale factor forces the elements to grow larger more quickly as one moves away from the center. To compute this function:

1. Highlight the “distance” function in the Datasets window and click the **Dataset Info...** button.
2. In the *Dataset Info* dialog, notice that the *Maximum* value is 65607.9.
3. Click the **X** in the corner of the dialog window to close the *Dataset Info* dialog.

4. Enter “sqrt(d6 / 65607.9)” in the *Expression* field. This assumes that d6 is the distance function.
5. Enter the name “scale” and click the **Compute** button.

5.5 Final Size Function

Now to create the final size function that the mesh will be based on.

1. Click the **max** button.
2. Replace “??,??” so the equation reads “max(50, (d7*d8))”. This will multiply the scale factor (which varies from very small by the center of the domain up to one at the edges) by the size (which is seven times that of the wavelength). The result will be a value that varies from very small to seven times that of the wavelength that is truncated to a minimum size of 50 meters to prevent infinitely small elements from being created around the mesh center.
3. Enter the name “radial size” in the *Output dataset name* field and click the **Compute** button.

The *Data Calculator* gives many options for building the size function. The size function created in this tutorial was created through several steps. This was done to show the many possibilities that exist for defining the size function, and ultimately for defining the finite element mesh. Other options that could be used for this or other meshes include:

- Use the wavelength multiplied by a scale factor (without using distance).
- Don’t take the square root of the scale factor for a denser mesh.
- Use a value other than 50 meters as the minimum size for a denser mesh in the channel, etc.

5.6 Smooth Size Function

The final step in creating a size function is to smooth the size function. Smoothing modifies the size function so the size function values do not change too quickly. Size functions that change too quickly can create poor transitions in element size.

1. In the *Dataset Toolbox*, select *Smooth datasets* from the tool list on the left.
2. Select the scatter data set named “radial size”.
3. Change the *Area change limit* to “0.5” un the *Elements* section. This will modify the size function so the elements created by the size function are at most twice as big or half as small as their adjacent elements.
4. Enter the *Output dataset name*: “radial size smoothed 0.5” and click the **Compute** button.
5. Click **Done** to exit the *Dataset Toolbox*.

(Note: If desired, the differences between the dataset “radial size” and “radial size smoothed 0.5” can be visualized by using the *Data Calculator* to subtract “radial size”

from “radial size smoothed 0.5” and contouring the resulting dataset.)

6 Creating Polygons

A polygon is defined by a closed loop of feature arcs and can consist of a single feature arc or multiple feature arcs, as long as a closed loop is formed. For initial mesh generation, polygons are a means for defining the mesh domain.

6.1 Building Polygons

To create polygons from the arcs on the screen:

1. Switch to the *Map*  module.
2. Make sure that no arcs are currently selected.
3. Select *Feature Objects* / **Build Polygons**.

Now a polygon has been created out of all the arcs.

6.2 Polygon Attributes

Next, each polygon (this case only has one) must be assigned proper attributes.

1. Choose the **Select Feature Polygon**  tool and click inside the polygon.
2. Select *Feature Objects* / **Attributes**. (Double-clicking inside the polygon will perform this same step.) The *2D Mesh Polygon Properties* dialog will open.

6.3 Assigning the Mesh Type

1. In the *2D Mesh Polygon Properties* dialog, select “Scalar Paving Density” as the *Mesh Type*.
2. Click the **Scatter Options...** button below the *Mesh Type*. The *Interpolation* dialog should appear.
3. In the *Interpolation* dialog, in the dataset tree under *Scatter Set To Interpolate From*, make sure the “radial size smoothed 0.5” function is highlighted.
4. In the *Extrapolation* section, set the *Single Value* to “50”.
5. Turn on the *Truncate values* option and set the *Min* to “50” and the *Max* to “5000”.
6. Click the **OK** button to return to the *2D Mesh Polygon Properties* dialog. If a warning message appears, click **OK**.

6.4 Assigning the Bathymetry Type

Next, the bathymetry type is selected. In this case the imported bathymetry is in the form

of a scatter set.

1. Select “Scatter Set” as the *Bathymetry Type*.
2. Click the **Scatter Options...** button below the *Bathymetry Type* option.
3. In the *Interpolation* dialog, highlight “elevation” under *Scatter Set To Interpolate From*, leave the *Single Value* at “0.000”, and make sure the *Truncate values* option is turned off.
4. Click the **OK** button.

6.5 Assigning the Polygon Type

1. Make sure the *Polygon Type* is set to “Ocean”.
2. Click the **OK** button to close the *2D Mesh Polygon Properties* dialog.

7 Creating the Mesh

Once the polygon attributes are set, the mesh can be generated automatically based on the options that were selected. To generate the mesh:

1. Select Feature Objects | *Map* → *2D Mesh*.
2. In the *2D Mesh Options* dialog, turn off the toggle for *Copy coverage before meshing* and click **OK**.
3. In the *Mesh Name* dialog, enter the name “ADCIRC Mesh” and click **OK**.

7.1 Mesh Display Options

After SMS has completed generation of the mesh, users should be able to view the bathymetry, nodes, and elements. To set the display:

1. Switch to the Mesh  module.
2. Select *Display* | **Display Options...** or select the  macro.
3. In the *Display Options* dialog under the *2D Mesh* section, make sure the *Nodes* and *Contours* are turned off and the *Elements* are turned on.
4. Click the **OK** button to close the *Display Options* dialog.

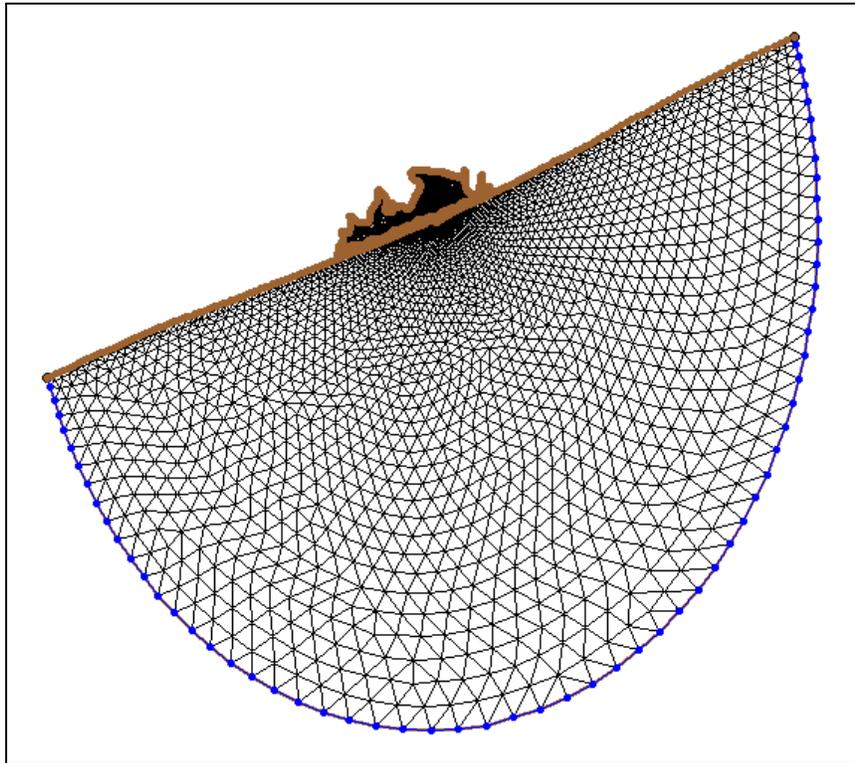


Figure 5 View of elements after automatic mesh generation

Figure 5 shows the final mesh. Notice how the elements are smaller closer to the coast and within the inlet. Once the mesh has been created and refined, final preparations must be done in order to run ADCIRC. These items are renumbering of the mesh nodes and saving the grid.

7.2 Minimizing Mesh Bandwidth

Before running ADCIRC, the mesh nodes must be renumbered to minimize the bandwidth of the mesh. This allows the ADCIRC model to run efficiently. SMS has done this automatically as the mesh was generated. However, if you edit the mesh, you would need to renumber again. To do this:

1. Select the **Select Nodestring**  tool and select the nodestring along the ocean boundary.
2. Right-click and select **Renumber Nodestrings**.

The nodes have now been renumbered for the entire mesh starting with those along the ocean boundary.

8 Building the ADCIRC Control File

The control file specifies values corresponding to different parameters for ADCIRC runs. These parameters include specifications for tidal forcing, selection of terms to include,

hot start options, model timing, numerical settings, and output control. In order for ADCIRC to run properly, the mesh must be converted to Latitude/Longitude coordinates.

8.1 Converting back to Lat/Lon

The model control expects the coordinates to be in latitude/longitude. The initial conversion was made to UTM coordinates for the meshing. The size function was calculated in meters, so the mesh could not be created while the coordinates were in degrees without performing more conversions (i.e. degrees ↔ meters). To convert back to Geographic coordinates:

1. Select *Display / Projection* to bring up the *Display Projection* dialog.
2. In the dialog select **Set Projection** to bring up the *Select Projection* dialog.
3. In the *Select Projection* dialog, change the display Projection to “Geographic (Latitude/Longitude)”.
4. Click **OK** twice to exit the *Display Projection* dialog.
5. Now right-click on the “ADCIRC Mesh” object in the Project Explorer and select **Reproject**.
6. Click **Yes** if a warning message appears.
7. In the *New Projection* side, the projection should be “Geographic (Latitude/Longitude), NAD27, arc degrees”. If it is not, use the **Set Projection** button to change the projection in the *Select Projection* dialog.
8. Make sure the *Vertical* units are in “Meters”.
9. Click the **OK** button.

8.2 Main Model Control Screen

Before setting the model control, it is necessary to make sure that SMS points to the right location of the LeProvost tidal database, as SMS will need to access the database at some point in the model control. To do this:

1. Select *Edit / Preferences* to bring up the *SMS Preferences* dialog.
2. In the dialog, click on the *File Locations* tab.
3. In the *Other Files* section of the dialog, set the *LeProvost tidal database* to point to the location where the LeProvost tidal database files are stored. They may be found in the data files folder of this tutorial. Do this by:
 - a. Clicking the **Browse** button next to *LeProvost tidal database*.
 - b. Use the browser to locate the directory and click **Choose**.
4. Click **OK** to close the *SMS Preferences* dialog.

To set up the model control for ADCIRC:

1. Click on the “ADCIRC Mesh” item in the project explorer to make sure it is active.
2. Select *ADCIRC* | **Model Control** to bring up the *ADCIRC Model Control* dialog.
3. In the *General* tab, turn on the following options under *Terms* in the center of the dialog: *Finite amplitude terms*, *Wetting/Drying*, *Advective terms*, and *Time derivative terms*.
4. Click the **Options...** button below the *Wetting/Drying* option.
5. Make sure the following values are entered in the *Wetting/Drying Parameters* dialog:
 - *Minimum Water Depth*..... 0.05
 - *Minimum Velocity for Wetting*..... 0.02
6. Click **OK** to return to the *Model Control* dialog.
7. Enter a value of “3.0” for the *Lateral Viscosity* in the *Generalized Properties* section on the right side of the dialog.
8. Change the *Bottom Stress/Friction* method to “Constant Quadratic” and set the *Friction coefficient* to “0.005”.

8.3 Time Control

Next, values for the *Timing* must be set. To set these values:

1. Click on the *Timing* tab of the *ADCIRC Model Control* dialog.
2. Set the following values:
 - *Ramp function value*: “1.0” days. (This is a time period for the model to ramp from no circulation to full tidal amplitude. This enhances stability. The results from these calculations do not match physical reality and are normally not even saved.)
 - *Time Step*: “2.0” seconds
 - *Run Time*: “1.5” days (Normal simulations last for several days up to a full lunar month. This is set to 36 hours just to get past the ramp time and show a tidal cycle.)

8.4 Output Files

ADCIRC will generate two global output files, water-surface elevation and velocity. To set the time for the two files:

1. Click on the *Files* tab of the *ADCIRC Model Control* dialog.
2. In the *Output Files Created by ADCIRC* section scroll down to *Elevation/Bathymetry Time Series (Global)* and turn the *Output* checkbox on.

3. Make sure the *Start (day)* is set to “1.0” and set the *End (day)* to “1.5” and the *Frequency (min)* to “30”.
4. Repeat steps 2 and 3 for the *Velocity Time Series (Global)*.

8.5 Tidal Forces

For this run of ADCIRC, tidal forcing will be used. To define the tidal constituents that ADCIRC will apply at the ocean boundaries:

1. Click on the *Tidal/Harmonics* tab of the *ADCIRC Model Control* dialog.
2. Check the *Use forcing constituent* and *Use potential constituents* boxes in the *Tidal Constituents* section.
3. Click the **New** button next to the checkboxes to bring up the *New Constituent* dialog.
4. In the *New Constituent* dialog, make sure the *LeProvost* constituent database is selected.
5. Select the *K1*, *M2*, *N2*, *O1*, and *S2* constituents in the *Constituents* section.
6. Set the *Starting Day* as 0.0 hours on February 1, 2000 (*Hour*: “0.0”, *Day*: “1”, *Month*: “2”, and *Year*: “2000”). This is the date from which the tides will start.
7. Click the **OK** button to return to the *ADCIRC Model Control* dialog.

SMS takes each constituent, extracts the values it needs from the *LeProvost* constituent database, and places them into the spreadsheet in the lower left corner. The amplitude and phase values may then be adjusted for each node.

8. If a message appears indicating that SMS cannot find one of the constituent files, click **OK** and find the file. The file should be located with the tutorial files.
9. Click the **OK** button to exit the *ADCIRC Model Control* dialog.

8.6 Saving the Mesh and Control Files

To save the mesh and control files:

1. Select *File* | **Save New Project...**
2. In the *Save* dialog, enter the name “shinfinal.sms” and click the **Save** button.

9 Running ADCIRC

Now to run ADCIRC. Presently, ADCIRC uses a specific naming convention for its input and output files. Therefore, before ADCIRC can start, the basic input files must be present in the working directory, which SMS takes care of automatically. SMS makes a copy of the active mesh file and names it “fort.14”, then makes a copy of the model control information file and names it “fort.15”.

To run ADCIRC:

1. Select *ADCIRC* | **Model Control** to bring up the *ADCIRC Model Control* dialog.
2. Select the *General* Tab. Name the project "test" in the *Project title* and enter "test" for *Run ID* as well. Press **OK**.
3. Select *ADCIRC* | **Model Check**. This will bring up the *Model Checker* if any issues are found.
4. Resolve any issues before proceeding using the instructions in the *Model Checker* dialog. If there are no issues Click **OK**.
5. Select *ADCIRC* | **Run ADCIRC**.
6. If the name of the ADCIRC executable does not appear, click the folder icon , locate the ADCIRC executable, and click **OK**.

The ADCIRC model wrapper appears and gives status for 64800 timesteps while the model runs. On a typical desktop machine, this will take around 15 minutes. Once the ADCIRC run has completed, there will be several new files created. SMS copied the "shinfinal.grd" file (the mesh file saved when the project file was saved) to "fort.14" and "shinfinal.ctl" file to "fort.15", the filenames needed by ADCIRC. ADCIRC created the "fort.63" (global elevation) and the "fort.64" (global velocity) files. There are a couple of other files that hold basic output information, but we will only focus on the elevation and velocity files for the remainder of this tutorial.

7. When ADCIRC finishes running, click **Exit**.

10 Importing ADCIRC Global Output Files

Each output file from ADCIRC is imported into SMS as a "Dataset." There are two types of datasets, scalar and vector. The global elevation file is an example of a scalar dataset, while the global velocity file is a vector dataset. Users import the global elevation file and the global velocity file simultaneously. To do this:

1. Select *File* / **Open**.
2. In the *Open* dialog, hold the *Shift* key down and select both the "fort.63" and "fort.64" files.
3. Click the **Open** button.
4. Click **OK** in the *Convert to XMDF* dialog to convert both solution files.

SMS reads in the files and adds "Water Surface Elevation (63)" and "Depth-averaged Velocity (64) mag" as scalar datasets and "Depth-averaged Velocity (64)" as a vector dataset in the "Mesh Data" of the Project Explorer.

11 Viewing ADCIRC Output

Once both ADCIRC output files have been imported, the user must decide on how to

view the data. The Project Explorer may be used to select the desired scalar and vector datasets.

1. Activate the “Water Surface Elevation (63)” scalar dataset by clicking on it in the Project Explorer.

11.1 Scalar Dataset Options

A good way to view the output is to edit the contour display options. To change the contour properties:

1. Select the **Display Options**  macro to bring up the *Display Options* dialog.
2. In the *2D Mesh* tab, click the **All off** button to turn off current display options.
3. Turn on the *Contours*, and *Mesh boundary*.
4. Under the *Contours* tab, change the *Contour Method* to “Color Fill”.
5. For the “Number” under *Contour interval*, enter “25”.
6. Click **OK** to exit the dialog box, and SMS will redraw the screen similar to Figure 6 (it will look about the same for the Time step 1 01:00:00).

To view the data at different time steps, select the desired time steps in the *Time Steps* window. The *Time Step* value is in hours, minutes and seconds from the start of the ADCIRC run.

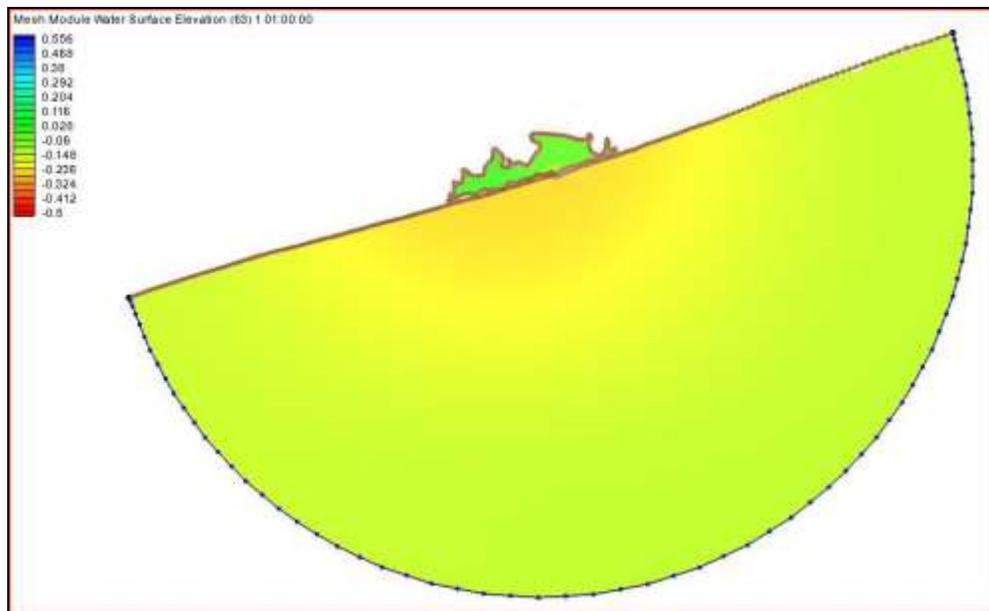


Figure 6 ADCIRC output from the “fort.63” file

11.2 Vector Dataset Options

Velocity vectors can be displayed in several different ways. This tutorial will first view them displayed at each node, and then on a normalized grid.

Vectors at each Node

1. Using the **Zoom**  tool, zoom in on the mesh so only the bay area is visible.
2. Select the **Display Options**  macro.
3. In the *Display Options* dialog under the *2D Mesh* tab, turn on the *Vectors* toggle.
4. In the *Vectors* tab, under the *Arrows Options* section, make sure the *Shaft length* is set to “Define min and max length”.
5. Change the *Minimum* length to “15” and the *Maximum* length to “40”.
6. Click the **OK** button to exit.

The screen should now look similar to that shown in Figure 7. Users can now visualize the flow at each node through Shinnecock Bay at this particular time step.

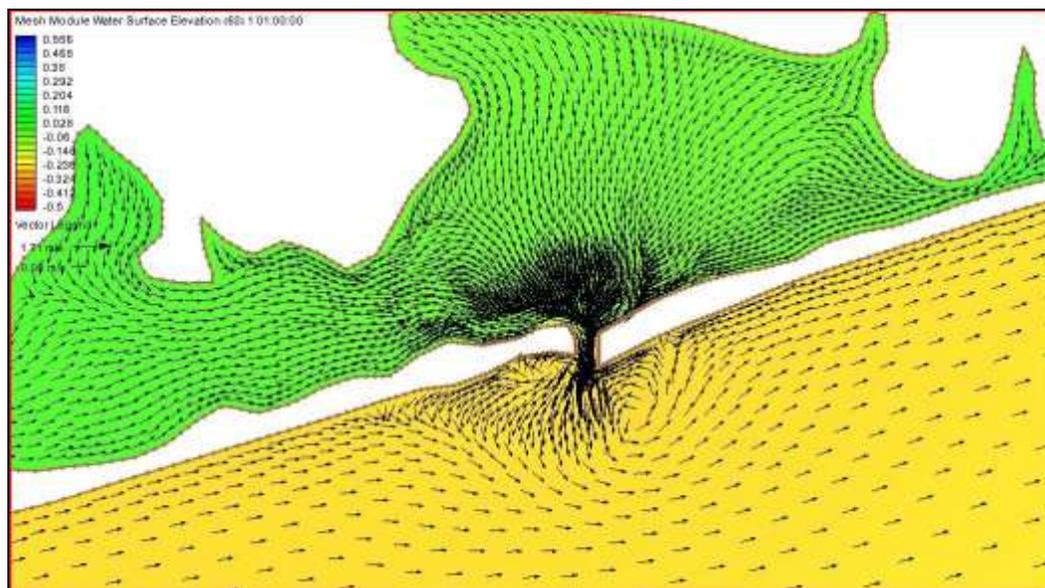


Figure 7 View of velocity vectors at each node

Vectors on a Normalized Grid

1. Click on the **Display Options**  macro.
2. In the *Display Options* dialog, go to the *Vectors* tab.
3. Under *Vector Display Placement and Filter*, change the *Display* to the “on a grid” option.
4. For both the *X spacing* and *Y spacing*, enter a value of “15” and click the **OK** button.

This method of displaying vectors is useful when displaying areas with both coarse and refined areas, such as the entire mesh in this case.

12 Film Loop Visualization

In addition to single time steps of contours and vectors, animations can be generated and saved. SMS enables the user to generate and save animations by using the Film Loop.

To create a film loop of the ADCIRC analysis:

1. Select *Data* | **Film Loop**.
2. In the *Film Loop Setup* dialog, leave the set defaults to create an AVI file and click the **Next>** button.
3. Continue accepting the default options. Click the **Next>** button, then the **Finish** button.

SMS now starts the film loop, adding one frame at a time. Once the last frame has been added to the loop, an AVI Application will open and the animation will start automatically.

Users may continue to experiment with the film loop features if desired. Click the **Close** button when finished. The film loop has been saved as sms.avi.

13 Conclusion

This concludes the ADCIRC Analysis tutorial.