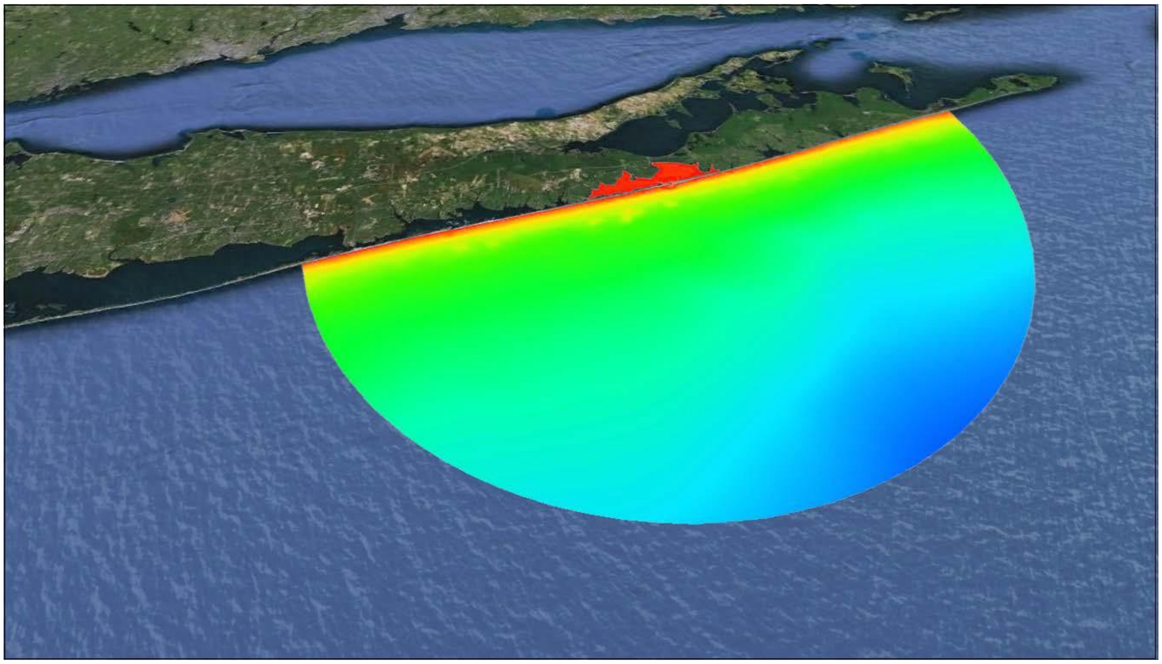
v. 13.0

SMS 13.0 Tutorial
ADCIRC Analysis



Objectives
This tutorial reviews how to setup and run an ADCIRC simulation. It will cover preparation of the necessary input files for the ADCIRC circulation model and visualization of the output.

Prerequisites
- Overview Tutorial
- Size Function
- Map Module
- Mesh Generation

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

Time
- 15–30 minutes

1 Introduction 2
2 Reading in the starting project 2
3 Assigning Boundary Conditions 3
3.1 Creating the Boundary Condition Coverage and Arcs 3
3.2 Setting the Boundary Condition Attributes 4
4 Setting up the ADCIRC Simulation 5
4.1 Creating the Simulation 5
4.2 Adding Components to the Simulation 5
4.3 Setting Model Parameters 6
4.4 Tidal Forces 7
5 Running ADCIRC 8
5.1 Aborting the Run (optional) 8
5.2 Loading the Solution 9
6 Conclusion 9

1 Introduction

The ADCIRC (Advanced Circulation) model is a finite element hydrodynamic model for coastal oceans, inlets, rivers and floodplains. This tutorial will go over the process of creating an ADCIRC model using SMS. It will start by reading in a project with the mesh already generated. Review the “Size Function” and “Mesh Generation” tutorials to get to this point.

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.

2 Reading in the starting project

An initial project has been created containing a generated mesh and relevant data. For information on how this mesh was created, see the “Mesh Generation” tutorial. To open the project to start this tutorial, with SMS open:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Browse to the data files folder for this tutorial and select the file “shin.sms.”
3. Click **Open** to import the project.

The project should appear in the Graphics Window as show in Figure 1.

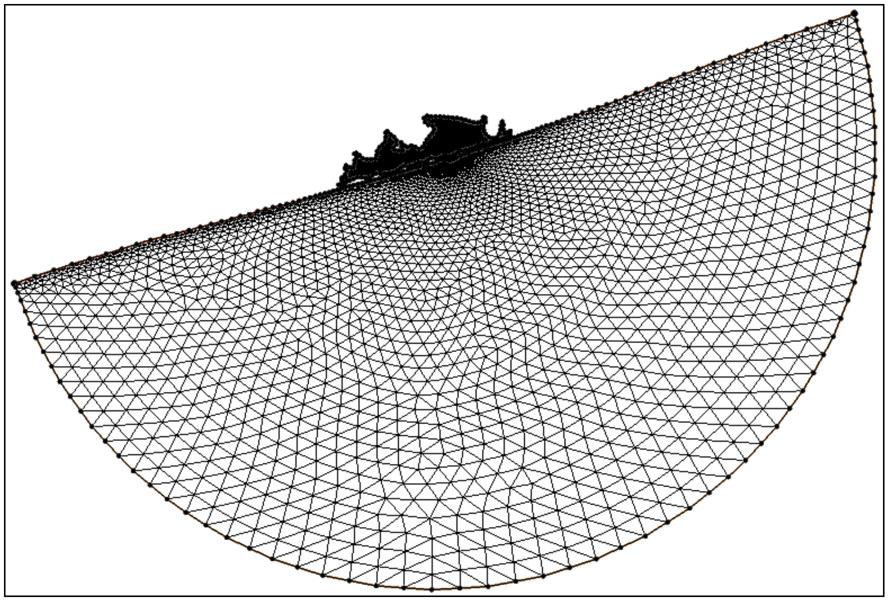


Figure 1 Initial mesh

The project includes:

- A generic mesh of Shinnecock Bay on Long Island, New York, and a small part the coastal region outside the bay.
- A scatter set representing bathymetric depths in the region.
- A map coverage which includes an arc representing the coastline and an arc representing the ocean boundary.
- The display projection has been set to “GCS_North_American_1927” (geographic). The mesh is also relative to this datum.
- The projection for the scatter set and map data is “NAD27 UTM Zone 18.”

3 Assigning Boundary Conditions

The ADCIRC model requires the user to define the boundary types for all edges of the domain. This is done by creating arcs around the domain and specifying the boundary type for each arc. The arcs need not be precisely located; they will be snapped to the mesh for each simulation being considered. If existing arcs exist they can be reused for this purpose.

3.1 Creating the Boundary Condition Coverage and Arcs

The boundary condition arcs must reside in an ADCIRC boundary condition coverage. In this case, a coastline and ocean arc already exist in the coastline coverage, so these will be reused. To create do this:

1. Right-click on the “ coastline” coverage in the Project Explorer and select **Duplicate**.

A new "coastline(2)" coverage will appear in the Project Explorer.

2. Right-click on the "coastline(2)" coverage and select **Rename**.
3. Enter the name "BC" and press *Enter*.
4. Right-click on the "BC" coverage in the Project Explorer and select *Type | Models | ADCIRC* | **Boundary Condition**.

This sets the coverage type so that it can be used in an ADCIRC simulation.

Note: refer to the "Map Module" tutorial to review the process of creating arcs. Remember that in this case, since the boundary condition arcs will be mapped (or snapped) to the mesh, they do not need to be precise. They can be simple approximations of the boundaries.

3.2 Setting the Boundary Condition Attributes

With the boundary condition coverage created, the arcs on the coverage need to be assigned attributes that define the boundary condition parameters for the simulation.

To set the boundary types:

1. Select the "BC" coverage in the Project Explorer to make it active
2. Using the **Select Feature Arc** tool, double-click the arc representing the ocean boundary as shown in Figure 2. The *ADCIRC Linear BC* dialog will appear.

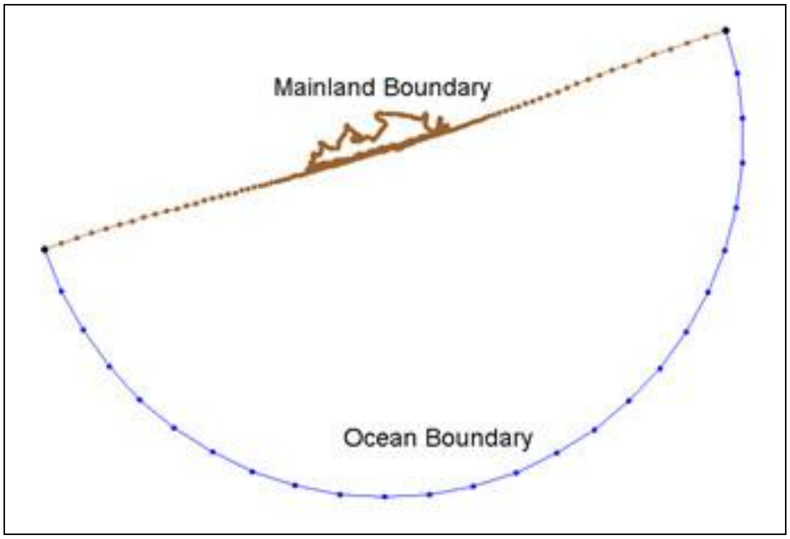


Figure 2 Feature arcs after boundary types have been assigned

3. Select the *Ocean* option under the *Type* section to assign this arc as an ocean boundary arc.
4. Click **OK** to close the *ADCIRC Linear BC* dialog.
5. Using the **Select Feature Arc** tool, double-click the arc representing the mainland boundary as shown in Figure 2 to open the *ADCIRC Linear BC* dialog.

6. Select the *Mainland* option under the *Type* section to assign this arc asa mainland boundary arc. Leave the *BC Options* set to "Natural."
7. Click the **OK** button to close the *ADCIRC Linear BC* dialog.

Both arcs have now been assigned the boundary conditions attributes needed for the simulation run.

Note: The natural boundary condition is much more numerically stable and is the recommended option.

4 Setting up the ADCIRC Simulation

Starting in SMS 13.0, the interface for ADCIRC uses a simulation based representation. The simulation based approach allows for reuse of meshes, boundary condition coverages and other model components. Multiple simulations can be created in a single SMS project file.

4.1 Creating the Simulation

To create the simulation:

1. Right-click in an empty area of the Project Explorer and select *New Simulation* | **ADCIRC**.

A new " ADCIRC Simulations" group will appear in the Project Explorer along with a new simulation named " Sim."

2. Right-click on the " Sim" simulation and select **Rename**.
3. Enter the name "Run1" for the simulation and press *Enter*.

4.2 Adding Components to the Simulation

An ADCIRC simulation consists of a mesh (basis of computations), boundary conditions, and model parameters. The components already exist in the Project Explorer, now they must be linked to the new simulation. Two methods exist to link the components. This section illustrates both.

To add the components:

1. In the Project Explorer, right-click on the " ADIRC Mesh" object and select *Link to* | **ADCIRC Simulations→Run1**.

Notice an " ADCIRC Mesh" item has been added under the " Run1" simulation. The " ADCIRC Mesh" item creates a link between the simulation and the mesh which tells the simulation to use the specified mesh during the model run. If multiple simulations were in the project, the same mesh can be linked to more than one simulation. Any changes to the mesh will automatically be updated in every simulation using that mesh.

Now to add a map coverage to the simulation.

2. Click-and-drag the " BC" coverage down to the " Run 1" simulation. As the cursor moves below the simulation name, a black line appears indicating this object can be added to this simulation. Let go of the mouse button to drop the " BC" coverage into the simulation.

The simulation will now make use of both the mesh geometry and the boundary condition map coverage.

4.3 Setting Model Parameters

To set up the ADCIRC simulation parameters:

1. Right-click on the " Run1" simulation and select **Model Control** to open the *ADCIRC Model Control* dialog.
2. Select the *General parameters* tab:
    a. Specify the *Project title* to be "Shinnecock sample."
    b. Specify the *Run ID* to be "ADCIRC v53"
    c. In the *Coordinates* section change the "Coriolis option:" to "Variable." This instructs ADCIRC to use the coordinates of the mesh to compute the influence of Coriolis forces.
    d. Leave all the other general parameters as the default settings. Refer to the ADCIRC documentation for descriptions of each parameter.
3. Select the *Model formulation* tab:
    a. In the *Nonlinear terms* section, set the *Finite amplitude terms* to be "With wetting/drying."
    b. Specify the *Minimum depth* to be 0.05
    c. Specify the *Minimum vel (m/s)* to be 0.02
    d. Turn on the *Advective terms – NOLICA* option.
    e. In the *Generalized wave continuity equation – GWCE* section, set the *Weighting factor – TAU0* to be "Pure primitive equation: TAU0 specified." Leave the default value of 0.005.
    f. Set the *Solver message level* to be "Fatal."
4. Select the *Timing* tab:
    a. Set the *Interpolation reference date* to "2/1/2000 12:00:00 am" (0.0 hours on February 1, 2000). This date is used by the interface to extract tidal or meteorological values from other data sources for use in a simulation.
    b. In the *Timing* section, set the *Time step (seconds)* to be "2.0."

   c. Set the *Length of run (days)* to be "1.5." 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.

   d. In the *Ramp Options – NRAMP* section set the *Number of hyperpoic tangent spin up ramps* to be "1."

   e. Set the *General ramp* duration to be "1.0" days.

5. Select the *Output* tab:

   a. In the *Water surface elevation* section set the *Start (days)* to be "1.0" (to be after the ramp) and the *End (days)* to be "1.5."

      Note: any value greater than the run duration will result in output being saved through the end of the simulation.

   b. In the *Velocity* section set the *Start (days)* to be "1.0" (after the ramp) and the *End (days)* to be "1.5."

6. Click **OK** to exit the *ADCIRC Model Control* dialog.

4.4 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. Right-click on the " BC" coverage and select the **Tidal Attributes** to open the *ADCIRC Tidal Attributes* dialog.
2. In the *Tidal forcing* tab, in the *Tidal database constituents* section, click on the **Insert Below** button. A row appears in the table
3. Change the *Source* for the new row to be "LeProvost" and select the *Constituent* for the new row to be "M2."
4. Repeat steps 2 and 3 for each of the following *Constituents*: K1, N2, O1 and S2.

When done, the *Tidal database constituents* table should contain 5 rows with 5 constituents.

5. Select the *Tidal potential input* tab.
6. Change the *Tidal potential* option to be "Used."
7. Click on the **Insert Below** button to add a row to the table.
8. Change the *Source* for the new row to be "LeProvost" and select the *Constituent* for the new row to be "M2."
9. Repeat steps 7 and 8 for each of the following *Constituents*: K1, N2, O1 and S2.

When done, the *Tidal potential constituents* table should contain 5 rows with 5 constituents.

10. Click **OK** to close the *ADCIRC Tidal Attributes* dialog.

SMS will extract the specified constituents at each boundary location when the fort.15 file is created during the "Export" process.

5 Running ADCIRC

Before running the simulation, 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 simulation run. To do this:

1. Select *Edit* | **Preferences** to bring up the *SMS Preferences* dialog.
2. Click on the *File Locations* tab. 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 (included with data files for this tutorial) and click **Choose**.
3. Click **OK** to close the *SMS Preferences* dialog.

The simulation is now ready to be run. To invoke the run:

4. Right-click on the " Run 1" simulation and select **Save, Export, and Launch ADCIRC**.

This command does the following:

- Perform model checks and report any detected issues for the user to address
- Save all the work done so far into the SMS project files
- Export the fort.14 and fort.15 files for the "Run 1" simulation into its own folder below the SMS simulation folder
- Launch the *Simulation Run Queue* and starts the "Run 1" simulation. Progress of the simulation can be monitored in the *Simulation Run Queue*.

This run will simulate 1.5 days (36 hours) of simulation with a 1.0 day (24 hour) ramp. The time step was specified as 2.0 seconds, so the simulation will include 64,800 time steps (1.5 days * 24 hours/day * 3600 seconds/hour * 0.5 time steps/second).Output will be saved for each point on the mesh every 30 minutes of simulation time from 24 hours to 36 hours (25 total output periods).

5.1 Aborting the Run (optional)

For this run of ADCIRC, tidal forcing will be used. On a typical laptop, running on a single core, this simulation of ADCIRC takes around 10 minutes.

A finished solution is provided to avoid waiting for the simulation to complete. Each output file from ADCIRC is imported into SMS as a dataset. If you wish to abort the run and then load the results:

1. Click on the **Abort** button in the *Simulation Run Queue.* This will terminate the simulation.
2. Click on the **Remove** button in the *Run Queue.* This will remove the simulation from the queue, meaning it is fully released for additional edits and manipulation in SMS.
3. The precomputed solution file(s) (fort.63, fort.64, maxele.63, maxvel.63, …) are in the output directory. To open these:
    a. Select *File* | **Open** to bring up the *Open* dialog.
    b. Browse to the output folder for the model run which will be called "shinfinal." Hold the *Shift* key down and select the desired solution files.
    c. Click the **Open** button to import the selected files

Alternately, a drag/drop process can be used to load the selected files. In either case, the datasets will be loaded into a simulation folder under the mesh.

5.2 Loading the Solution

Once the simulation runs to completion, do the following to load the solution:

1. Click on the **Load** button in the *Simulation Run Queue*. (This appeared in the place of the abort button when the simulation completed.)

This operation removes the simulation from the *Simulation Run Queue* and dismisses (closes) the *Simulation Run Queue*.

The solution datasets (" Elevation," " Velocity," " Maximum Elevation," " Maximum Velocity,"…) appear in a folder named for the simulation under the mesh. The number of datasets depends on the specified output options for the simulation.

6 Conclusion

Refer to the visualization tutorial to review/learn the methods used to generate images, curves, and animations for the solution data generated by ADCIRC.

This concludes the "ADCIRC Analysis" tutorial. Continue exploring the ADCRIC model in SMS or exit the program.

