SURFACE WATER MODELING SYSTEM

The LTEA Toolbox

1 Introduction

LTEA stands for the "Local Truncation Error Analysis". The name refers to an analysis methodology for measuring local truncation error in a numerical analysis. The LTEA Toolbox incorporated into the Surface-water Modeling System uses the LTEA algorithm as the heart of a utility which creates finite element meshes of varying resolution for ADCIRC analysis.

1.1 Before Using the Toolbox

Before the LTEA toolbox can be accessed, the user must load the geometric definition for an ADCIRC simulation. This definition consists of a bathymetric survey of the area to be modeled and an ADCIRC Coverage in the Map Module with arcs describing the shape of the modeling domain.

(Note: It should be observed that an existing ADCIRC mesh can be used as the basis for the analysis since it defines both the bathymetry and a domain boundary.)

Loading Data

To load the site data, the File | Open command is used. This tutorial makes use of a bathymetric survey and a coastline arc. (A modeler acquires this type of data from local surveys or from a data source such as those described in the GeoSpatial Data Acquisition page (http://xmswiki.com/xms/GSDA:GSDA) maintained by Aquaveo.

Two data files are provided for this exercise including:

- shin.cst – a coastline definition in Geographic NAD 27 coordinate.
- shin.pts – a set of survey points in UTM NAD 27 coordinates.

To get ready to use LTEA:

1. Select *File* | *Open* and select the file "shin.cst".

2. Right click on *default coverage* and select *rename*.
3. Change the name of coverage to “Shin Domain”.
4. Right click on “Shin Domain” and select *Type | Models | ADCIRC*.
5. Select *Edit | Projection* change to a global projection.
6. Under Projection, select Geographic (Latitude/Longitude),.
7. Change the Datum to NAD 27, Click *OK* and leave the *Vertical* Coordinates to local and change the *units* to meters. Click *OK* to close the *Current Projection* dialog.
8. Select the *Select Feature Arc*  tool.
9. Double click on the arc to open the *Attributes* dialog box.
10. Under Boundary Type, turn on the *Mainland* toggle. Click *OK* to exit dialog box.
11. Select *Feature Objects | Define Domain*, then set the arc as *Semi-circular,* then click *OK.*
12. Build polygons by using *Feature Object | Build Polygons*.
13. Double click on the newly created ocean arc to pull up the ADCIRC Arc/Nodestring Attributes.
14. Change the ocean arc to have the attribute *Ocean.* Click *OK.*
15. Go to *Edit | Reproject to* convert the coordinate system back to UTM – this is done for two reasons. First, the survey is in UTM and second, the Toolbox requires that data start in a rectangular system.
16. Click the *Global projection* (right side of the screen) and then click *Set Projection.*
17. Select UTM. Then verify the *Datum* is NAD27 and *zone* is 18 and click *OK twice*.
18. Open the survey file, *shin.pts*
19. Click *Next*, then *Finish* in the *File Import Wizard* dialog.

(Note: this coastline definition is less precise than the survey data. The user must define the level of detail for the coastline keeping in mind the impact on numerical analysis.)

1.2 Using the Toolbox

The LTEA toolbox consists of a wizard that guides the user through various steps in creating a mesh with varied resolution for an ADCIRC analysis.

Launching the Toolbox

To launch the toolbox:

1. Make sure the *Scatter Data* set is selected in the Project Explorer
2. Select *Data | Scatter -> Mesh*

3. Select the *Mesh Data* in the Project Explorer
4. Select the *ADCIRC | Mesh Generation Toolbox* command.
5. Select "Localized Truncation Error Analysis (LTEA)" and click Run

Step 1 - Specifying the Input

If a mesh already exists, the domain and bathymetry can be derived from it. In order to generate a simple mesh in order to run ADCIRC do the following:

1. Make sure the combo box labeled "Boundary" says "Shin Domain"
2. Select the button labeled "Select" next to "Bathymetry" and select the "Z" dataset. Click *Select.*
3. Make sure the *Create Linear Run Mesh* radio button is selected
4. Make sure the *Override Boundary Spacing* check box is checked

   By choosing this option the toolbox will space boundaries varying from the coastline spacing to the deepwater spacing shown in the edit fields below the checkbox. Currently these options are set to 100 on the coast to 10000 in Deep Ocean. **Error! Reference source not found.** shows how the boundary is spaced from the coastline to Deep Ocean. If great effort was put into spacing the boundary, you would want to leave this unchecked to preserve your efforts.
5. Check the *Save Linear Run Mesh* check box
6. Click the File button below *Save Linear Run Mesh*. Enter ltea_lin_mesh for the file name. Click *Save*
7. Click on the Continue> button

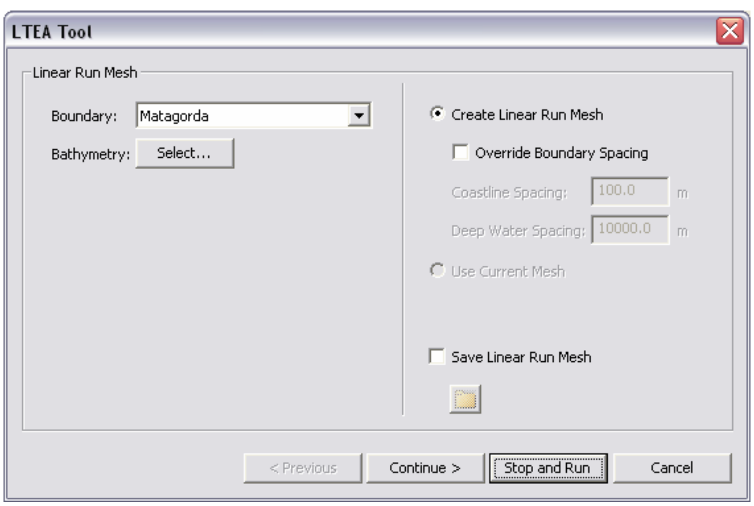

Figure 1 Initial mesh page of the toolbox

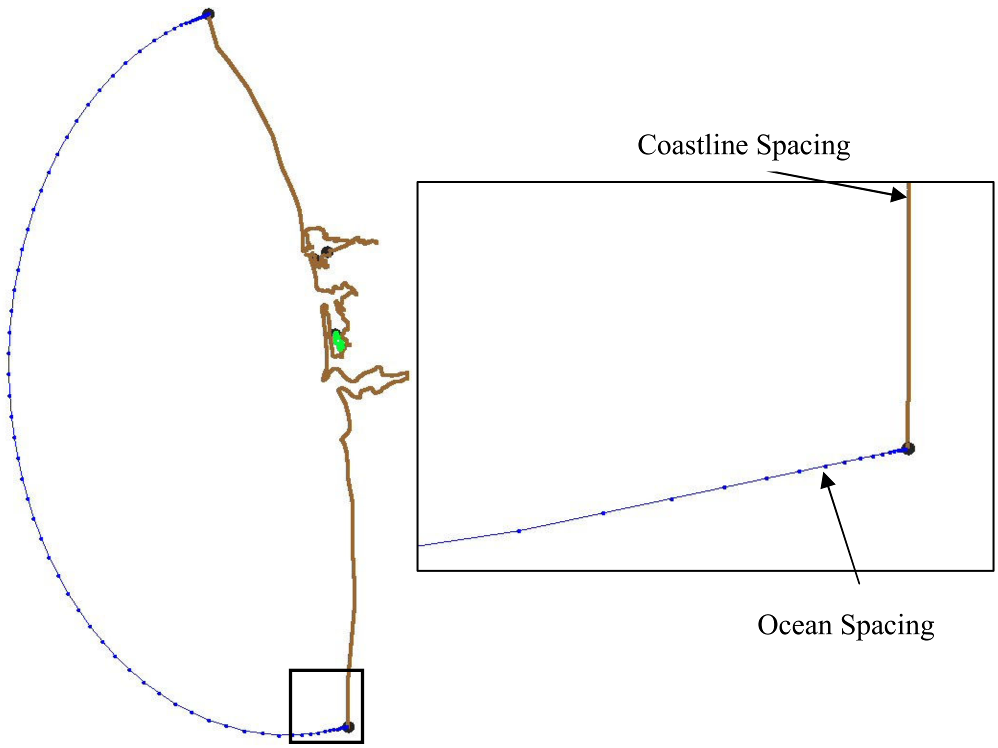

Figure 2 Automatic vertex spacing for initial grid (PICTURE OF WRONG DOMAIN)

Step 2 – Getting the harmonic solution

The LTEA analysis utilizes the harmonic solution data from an ADCIRC run. Normally, this solution comes from a linear run of the ADCIRC model on the linear mesh generated in the previous step.

Since no solution exists for a mesh in our test case, SMS has selected the "Run ADCIRC Linear Run" option and set the model parameters to standard defaults.

To set up the harmonic solution:

This brings up the "ADCIRC Linear Run" page. The term linear ADCIRC is used because several of the non linear terms in the ADCIRC solution schemes are turned off. The LTEA toolbox uses the following settings for the linear ADCIRC run:

- Finite Amplitude Terms are off
- Advective Terms are off
- Time Derivative Terms are off
- Coriolis Method is set to a constant value of 0
- Lateral Viscosity is set to 0
- Bottom Stress/Friction is set to Constant Linear
- Harmonic analysis is performed for the last 1.5 days of the simulation
- A single M2 tidal forcing constituent is extracted from the LeProvost database along the ocean boundaries. The Nodal Factor and Equilibrium Argument are set to 0.

- Minimum Water Depth is set to 1.25 times the maximum extracted tidal forcing amplitude
  We will leave the default values as they are. It should be noted that a time step of 15 seconds is used; this greatly reduces the run time compared to a non-Linear ADCIRC run.

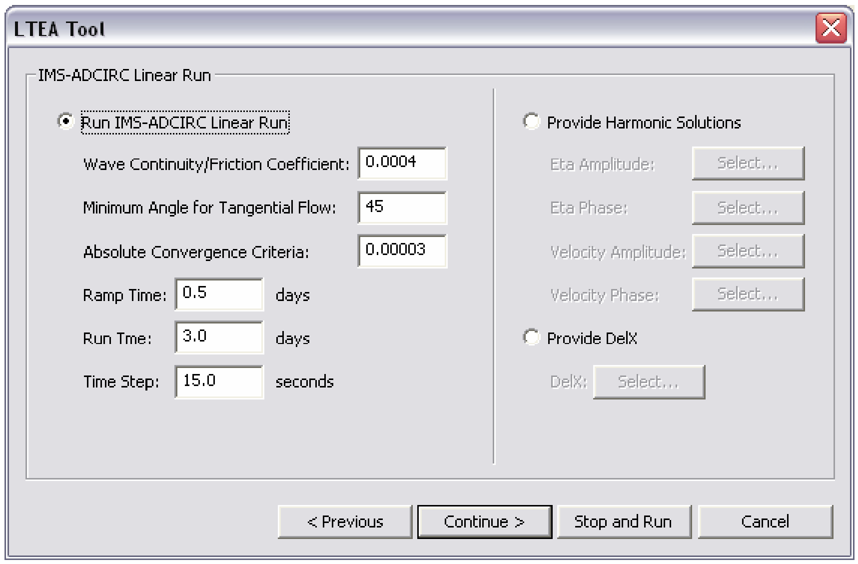

Figure 3 Linear ADCIRC run page of the toolbox

8. Click the “Continue>” button to move to the next step of the process.

(Note: If desired, the “Stop and Run” button would instruct SMS to close the toolbox and perform the linear run. This may take a few minutes. When it is done there will be several Datasets loaded into SMS and you could then restart the LTEA toolbox and skip through to the third step.)

Step 3 – LTEA Analysis

The LTEA analysis can now use the harmonic datasets created from the ADCIRC run to create a “size function” called DelX.

9. In order to get LTEA values close to the domain boundary, make sure that the “Use partial molecule” check box is selected.

- The option for the molecule size should be left at 9X9. This means that a small equally spaced grid 9 cells by 9 cells is created around each node, and values from the ADCIRC solutions are interpolated for all 81 cells. These values are then used for the LTEA analysis.

Nodes on the edges of the domain will not have values for cells outside of the domain and normally LTEA will not give a value for these nodes with "Partial Molecules." If Partial molecules are not used, the boundary spacing from the domain boundary will be used to place nodes on the final mesh until a region where LTEA values are present and the process then uses LTEA values.

10. Click on the "*Continue*>" button to move onto the next step.

(Note: If desired, the "Stop and Run" button would cause SMS to close the toolbox and run the LTEA analysis. The analysis generates several datasets used as size functions in the mesh generation process.)

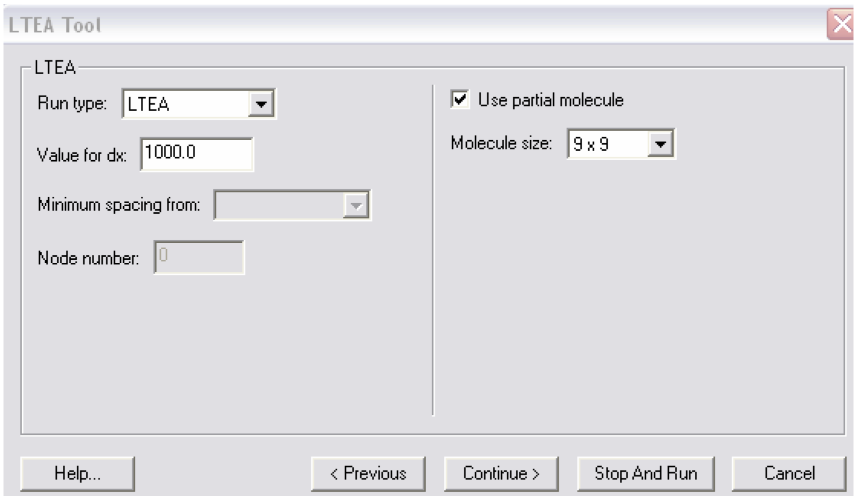

Figure 4 LTEA options page for the toolbox.

Step 4 – Generating the Mesh

Now that we have the size function called DelX we can use it to generate a mesh that is "influenced" by Local Truncation Error Analysis.

11. Input 10,000 for "*Target Number of Nodes*" and 500 for "*plus/minus*"
12. Leave the "*Element area change limit"* at 0.5.
13. In order to get more resolution in the body of the mesh rather than the edges, check the "*Redistribute Boundaries*" check box. This will allow the mesh generation process to redistribute the domain spacing according to LTEA values.
14. Click "*Run*".
15. Wait a few seconds and the model will begin to run (The LTEA dialog might still be showing. That is ok).

Note: If you get a message stating that the program could not find the file M2.legi, go to Aquaveo.com/downloads/SMS and download the LeProvost Tidal Files. Then direct SMS to find the files.

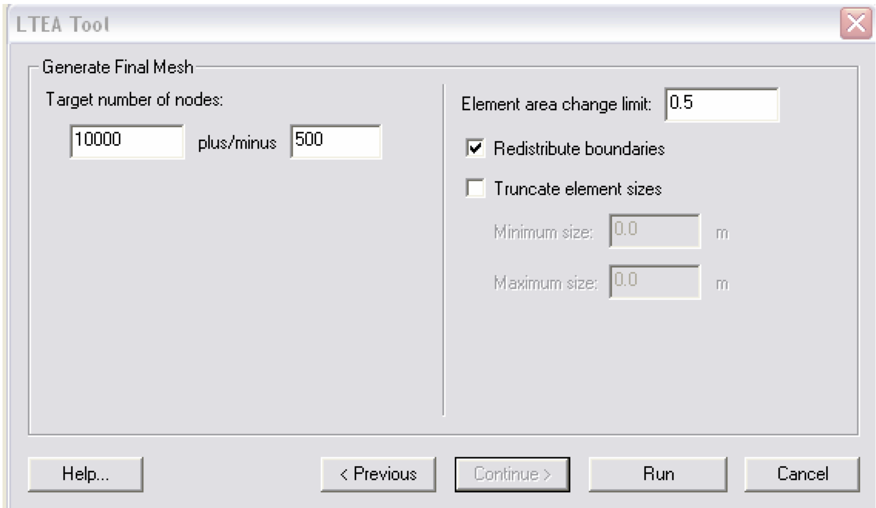

Figure 1-5 Final mesh page of the dialog

1. After the model finishes, a 2D Mesh Options dialog will be displayed. Press *OK (* you might have to press this a few times).
2. Click *Done* on the *Mesh Generation Toolbox*.
3. With the final mesh being loaded, note the different data sets labeled *Scaled*, with varying numbers that follow. In order to reach the correct target number of nodes, the original size function is scaled and a mesh is created. The final number of nodes in that mesh is then checked to the target to see if it is within the tolerance. If the total number of nodes is not within tolerance it is deleted and the size function is rescaled and the process is repeated until the target number of nodes is reached. The number after "Scaled" is the scaling factor used to create the different number of meshes.

1.3 Conclusion

For this tutorial the process was done step by step to explain the different options of each page. You might try and run the toolbox straight through once by just filling in all of the required fields and letting it run. The result will be the same. That is one of the most important parts of the toolbox. Anybody can take your inputs and get the same result every time. The process is repeatable.

Finding the correct number of nodes for a mesh is an ambiguous task and depends on the needs of your research. With the toolbox, many different meshes can be created with varying target number of nodes. These meshes can then be tested to determine how many nodes are the most advantageous. The figure below shows a domain with several meshes created with varying number of nodes.

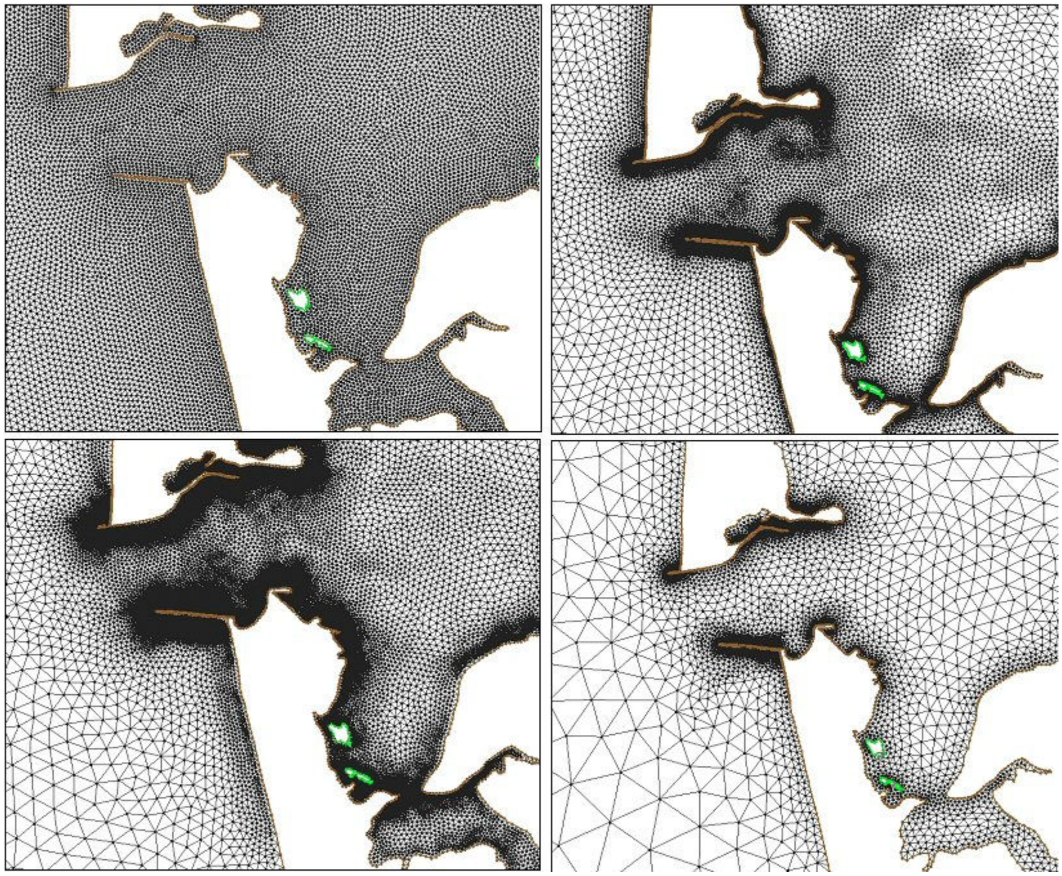

Figure 1-6 Various grid resolutions for a single domain