SMS 13.0 Tutorial
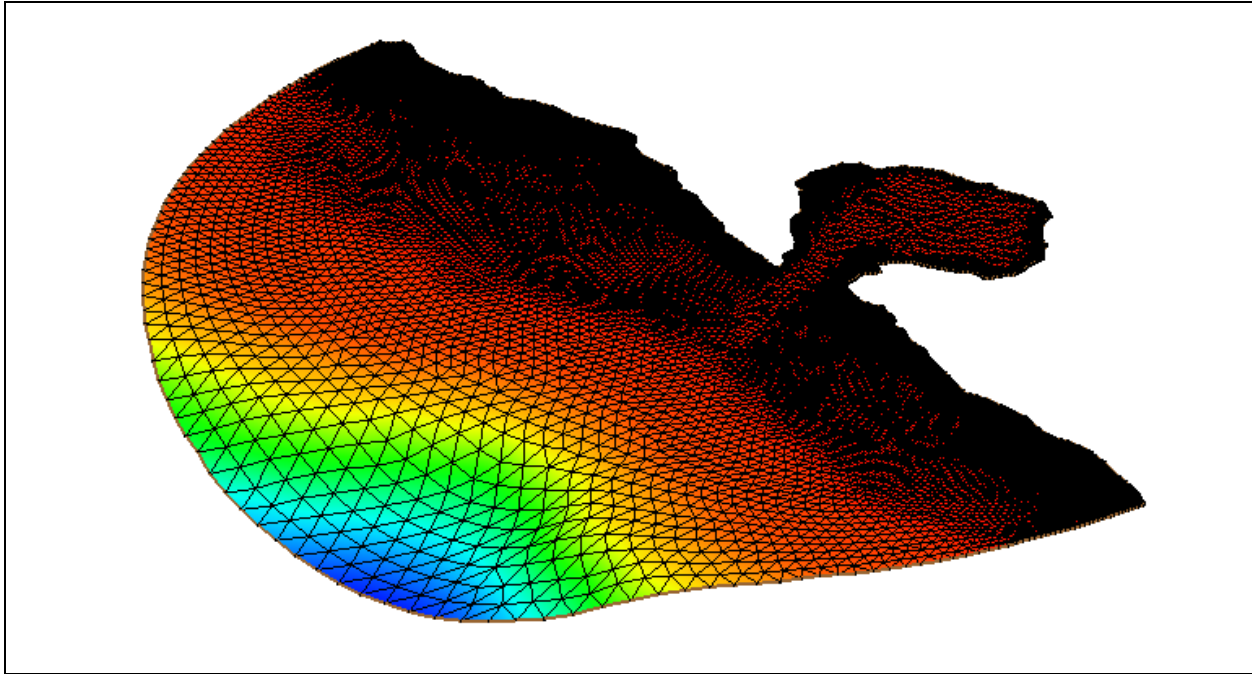
Creating a Wavelength Based Mesh

Objectives

Learn how to generate a mesh using a wavelength function.

Prerequisites

- Overview
- Map Module
- Mesh Generation

Requirements

- CGWAVE
- Scatter Module
- Map Module
- Mesh Module

Time

- 15–25 min

1   Getting Started

Note this tutorial uses the CGWAVE model interface, but the principles could be adapted to other coastal models such as ADCIRC.

This tutorial begins by opening the XYZ file containing a set of points with depth data. This data will be used to create a mesh.

1. Select *File | **Open…*** to bring up the *Open* dialog.

2. Browse to the *data files* folder for this tutorial and select "indiana.sms".

3. Click **Open** to exit the *Open* dialog and open the project.

A scatter set named " indiana" will be created in the Project Explorer, and the project should appear similar to Figure 1. This data is referenced to a UTM coordinate frame and is in meters.
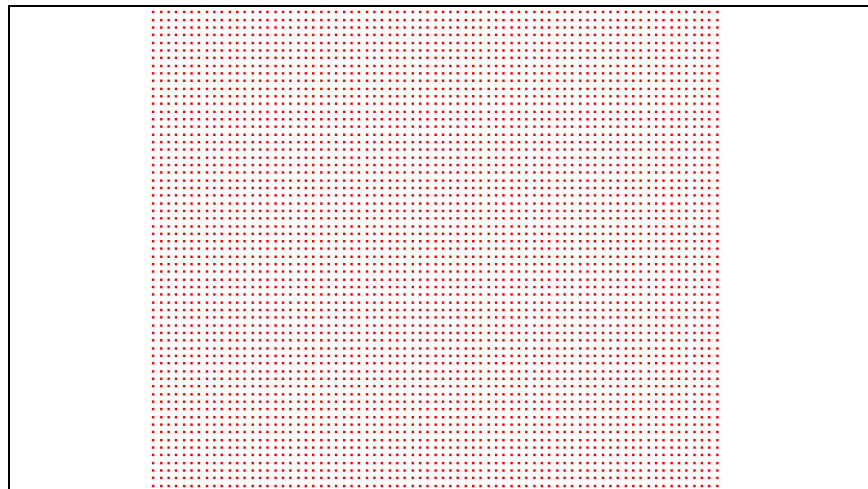

Figure 1   Scatter set from the imported XYZ file

2   Creating a Wavelength Function

The first step is to create a wavelength function. The wavelength function is an intermediate step to creating a size function, which is covered in the next section.

The *z* value of each point in the "indiana.xyz" data is actually a water depth value. The wavelength at each point is calculated from this depth value. A larger wavelength is calculated from a larger water depth value.

To create the wavelength function:

1. Select " indiana" in the Project Explorer to make it active.

2. Select *Data | **Data Set Toolbox…*** to bring up the *Dataset Toolbox* dialog.

3. In the *Tools* section, select "Wave Length and Celerity" from the tree list.

4. In the *Wave Length and Celerity* section, enter "Transition" as the *Output base name*.

5. Enter "20.0" as the *Period*.

6. Click **Compute**.

7. Click **Done** to close the *Dataset Toolbox* dialog.

Two new datasets—" Transition_Wavelength" and " Transition_Celerity"—will be visible in the Project Explorer.

3   Creating a Size Function

The size function is created from the wavelength function. The size function determines the element size that will be created by SMS. Each point is assigned a size value. This size value is the approximate size of the elements to be created in the region where the point is located. The mesh will be denser where the size values are smaller.

The wavelength function created above contains values that are twice as large as the desired size values. The wavelength function will be scaled by one half to create the size function.

To do this:

1. Select *Data | **Data Set Toolbox…*** to bring up the *Dataset Toolbox* dialog.

2. In the *Tools* section, select "Data Calculator" from the tree list.

3. In the *Data Sets* section of the *Data Calculator* section, double-click "d2. Transition_Wavelength" to add it to the expression in the *Calculator* section.

4. Click **/** (division) on the calculator button pad.

5. Press the "5" on the keyboard.

This represents the number of elements generated per wavelength. It is usually more appropriate to use a larger number of elements per wavelength (e.g., 10). The smaller number is used here to allow faster execution of the model.

6. Enter "size" as the *Output dataset name* and click **Compute**.

7. Click **Done** to close the *Dataset Toolbox* dialog.

A new dataset—" size"—based on the " Transition_Wavelength" dataset will appear in the Project Explorer.

3.1   Smooth Size Function

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

1. Select *Data | **Data Set Toolbox…*** to bring up the *Dataset Toolbox* dialog.

2. In the *Tools* section, select *Smooth datasets*.

3. In the *Data Sets* subsection of the *Smooth datasets* section, select the "size" dataset.

4. In the *Smoothing Options* section, select *Element area change*.

5. Enter "0.5" as the *Area change limit*.

This modifies the size function so the elements created by it are—at most—twice as big or half as small as their adjacent elements.

6. Enter "size smoothed 0.5" as the *Output dataset name* and click **Compute**.

7. Click **Done** to close the *Dataset Toolbox* dialog.

If desired, the differences between the dataset "🔳 size" and "🔳 size smoothed 0.5" can be visualized by using the data calculator to subtract "size" from "size smoothed 0.5" and contouring the resulting dataset.

4   Defining the Domain

A domain represents the region that is offshore. In CGWAVE, the domain can be a circular, semi-circular, or rectangular region. In SMS, a feature arc is used to define the coastline. After the coastline is defined, feature arcs and feature polygons are used to define the domain region.

4.1   Creating the Coastline

SMS can automatically create a coastline at a specific elevation or water depth from a scattered dataset. The active function of the active scattered dataset will be used for this operation. The project should currently have only one scattered dataset. To make the elevation function active:

1. Select the "🔳 Z" dataset in the Project Explorer to make it active.

2. Right-click on the "🔵 Area Property" coverage and select *Type | Models |* **CGWAVE**.

3. Right-click on "🔵 Area Property" and select **Rename**.

4. Enter "CGWAVE" and press the *Enter* key to set the new name.

5. Select "🔵 CGWAVE" to make it active.

With the coverage type set and the active scattered dataset defined, create the coastline by doing the following:

1. Select *Feature Objects* / **Create Coastline…** to bring up the *Create Contour Arcs* dialog.

2. In the *Contour options* section, enter "1.0" as the *Elevation*.

3. Enter "10.0" as the *Spacing along contour*.

4. Click **OK** to close the *Create Contour Arcs* dialog.

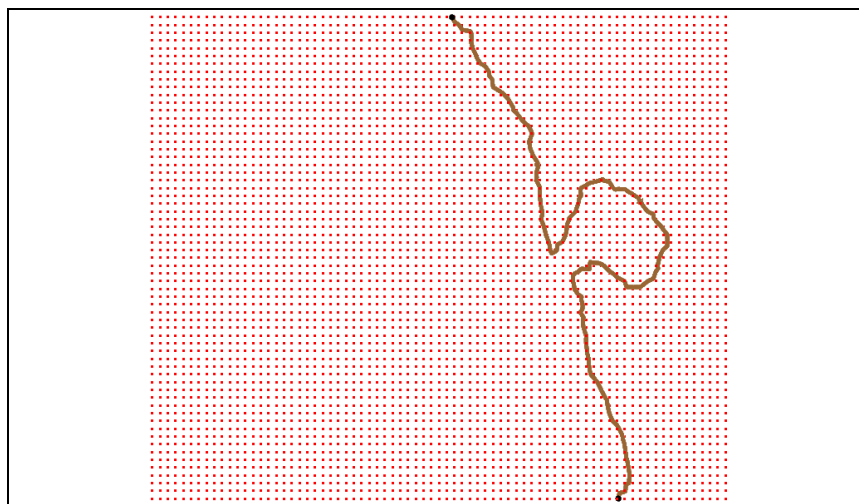The display will refresh with an arc representing the water depth line (see Figure 2).

Figure 2   The arc representing the coastline

4.2   Creating the Domain

SMS can create a domain from the coastline. This model will use a semi-circular domain that intersects with the coastline.

To create the domain:

1. Using the **Select Feature Vertex** tool, hold down the *Shift* key and select one vertex near each end of the coastline arc.

2. Select *Feature Objects* / **Define Domain…** to open the *Domain Options* dialog.

3. Select *Semi-circular* and turn on *Switch side*.

4. Click **OK** to close the *Domain Options* dialog.

5. Select *Feature Objects* / **Build Polygons**.

This creates a semicircular *Ocean* arc and a feature polygon (Figure 3). The feature polygon must be created from the feature arcs. Make sure that the domain does not extend outside the extent of the scatter set. If it does, delete the semi-circular arc and recreate it with points further within the interior of the scatter set.

Because the domain does not extend outside the scatter set, there are now two arcs on the coastline outside of the domain. These need to be removed, otherwise SMS will render these arcs as unwanted nodestrings when the coverage is converted to a mesh.

1. Using the **Select Feature Arc** tool, select the arc at the top of coastline that is outside of the domain and delete it.

2. Repeat the step above for the arc at the bottom of the coastline that is outside of the domain.
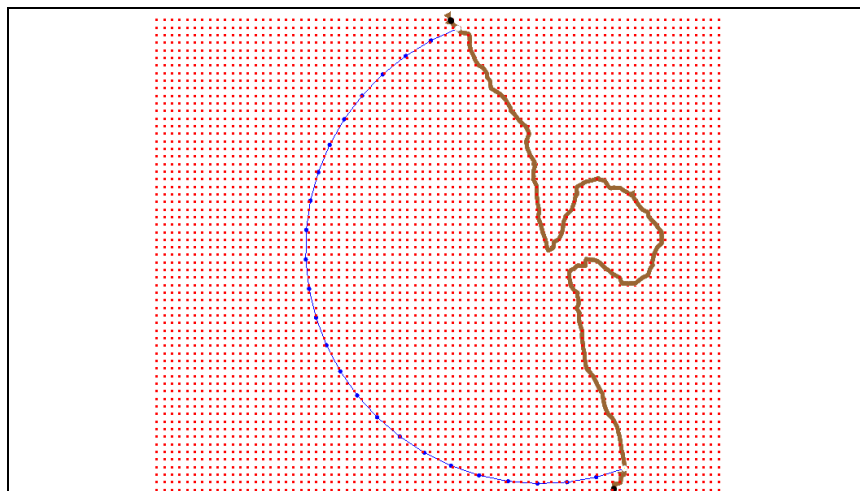

Figure 3   The ocean domain arc

The polygon is formed from the set of arcs that form a closed loop (in this case, from the ocean domain arc and the part of the coastline arc between the ends of the ocean domain arc). The screen will not refresh when polygons are built, so it may appear that nothing happened even though polygons were created.

4.3   Redistribute Vertices

Before generating the mesh, the vertices along the arcs need to be redistributed to match the smoothed size function. To do this:

1. Using the **Select Feature Arc** ⟨icon⟩ tool, select the ocean domain arc.

2. Right-click on the ocean arc and select **Redistribute Vertices** to open the *Redistribute Vertices* dialog.

3. Select "Size Function" for the *Specify* option.

4. Click the **Options** button to open the *Interpolate Source and Options* dialog.

5. For the *Source dataset*, select "Project\Scatter Data\indiana\size smoothed 0.5".

6. Click **OK** to close the *Interpolation Source and Options* dialog.

7. Click **OK** to close the *Redistribute Vertices* dialog.

The vertices along the arcs will be redistributed to match the smoothed size function dataset.

5   Creating the Finite Element Mesh

There are various automatic mesh generation techniques used to create elements inside a specified boundary. One of these is applied to each polygon, after which a finite element mesh can be generated. For this tutorial, there is only one polygon, which will be assigned the density mesh type.

5.1   Setting up the Polygon

When using density meshing, SMS determines element sizes from a size function in a scattered dataset. The size function to be used in this example was created in Section 3.

To set up the feature polygon for density meshing:

1. Using the **Select Feature Polygons** ⬚ tool, double-click inside the ocean domain polygon to bring up the *2D Mesh Polygon Properties* dialog.

2. In the *Mesh Type* section, select *Scalar Paving Density* from the drop-down.

3. Click **Scatter Options…** to bring up the *Interpolation* dialog.

4. In the *Interpolation Options* section, enter "10.0" in the *Single Value* field.

The value in this field and in the *Min* field—in step 6 below—must be the same. If they do not match when closing the *Interpolation* dialog, a warning will advise they have been changed to match.

5. In the *Other Options* section, turn on *Truncate values*.

6. Enter "10.0" as the *Min*.

7. Enter "10000.0" as the *Max*.

This sets up a minimum and maximum size to be used when creating elements.

8. In the *Scatter Set to Interpolate From* section, select the "size smoothed 0.5" dataset.

9. Click **OK** to close the *Interpolation* dialog.

10. In the *Bathymetry Type* section, select "Scatter Set" from the drop-down.

11. Click **Scatter Options…** to bring up the *Interpolation* dialog.

12. In the *Scatter Set to Interpolate From* section, select "Z" from the list of datasets.

13. In the *Other Options* section, turn off *Truncate Values*.

As mesh nodes are created, their elevation values will be assigned from the original water depth values that were imported from the original XYZ file.

14. Click **OK** to close the *Interpolation* dialog.

15. Click **OK** to close the *2D Mesh Polygon Properties* dialog.

The polygon is now set up to generate finite elements inside the boundary. When more than one polygon exists, the meshing attributes need to be set up for each of the polygons. As only one polygon exists in this tutorial, this process is complete.

5.2   Generating the Elements

The next step is to have SMS generate the finite element mesh from the defined domain.

To create the mesh:

1. Select *Feature Objects | **Map→2D Mesh*** to bring up the *Mesh Name* dialog.

2. Accept the default name by clicking **OK** to close the *Mesh Name* dialog.

3. Turn off " Scatter Data" and " Map Data" in the Project Explorer.

The Main Graphics Window should now appear as in Figure 4.
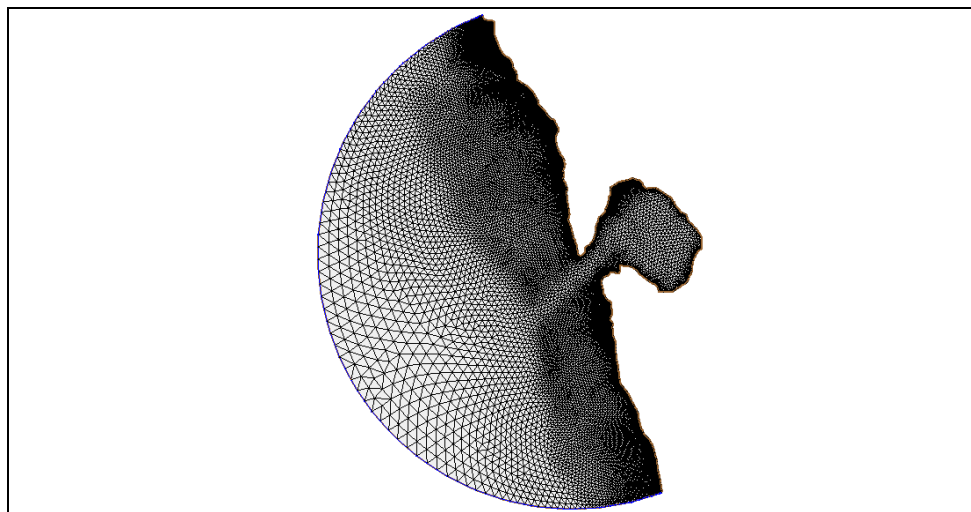

Figure 4   The mesh after turning off scatter and map data

To further declutter the project display, do the following:

1. Click **Display Options** to bring up the *Display Options* dialog.

2. Select "2D Mesh" from the list on the left.

3. On the *2D Mesh* tab, click **All Off**.

4. Turn on *Elements, Contours*, and *Nodestrings*.

5. On the *Contour Options* tab, in the *Contour method* section, select "Color Fill" from the drop-down.

6. In the *Contour interval* section, select "Number" from the drop-down and enter "20" in the field to the right of the drop-down.

7. Click **OK** to close the *Display Options* dialog.

After the display is refreshed, notice the contours of water depth with the elements drawn on top of those (Figure 5). As the water depth decreases, so does the element size. A dredged channel can be seen running into the harbor.
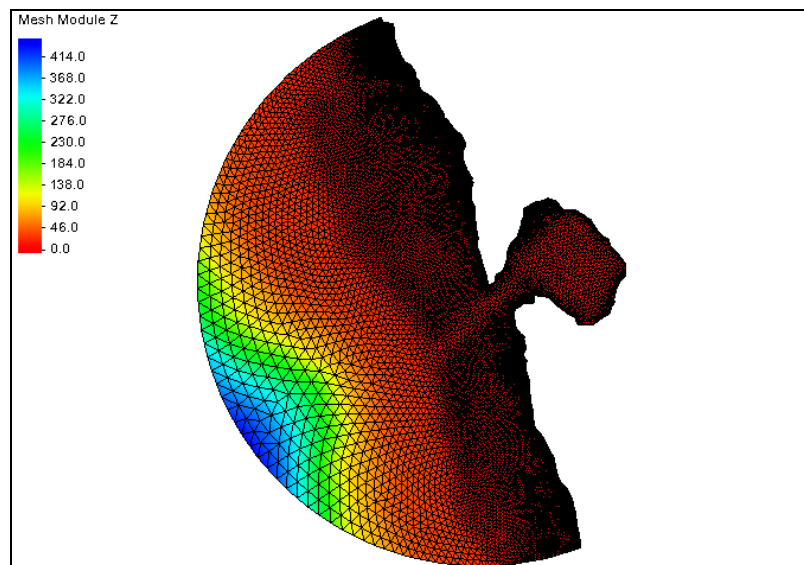
Figure 5   The contour color fill shows the increasing depth

The mesh can now be used to run a coastal model simulation. For more information on how to do this, see the "CGWAVE" tutorial.

6   Conclusion

This concludes the "Creating a Wavelegnth Based Mesh" tutorial. Continue to explore generating wave based meshes, or exit SMS at this point.