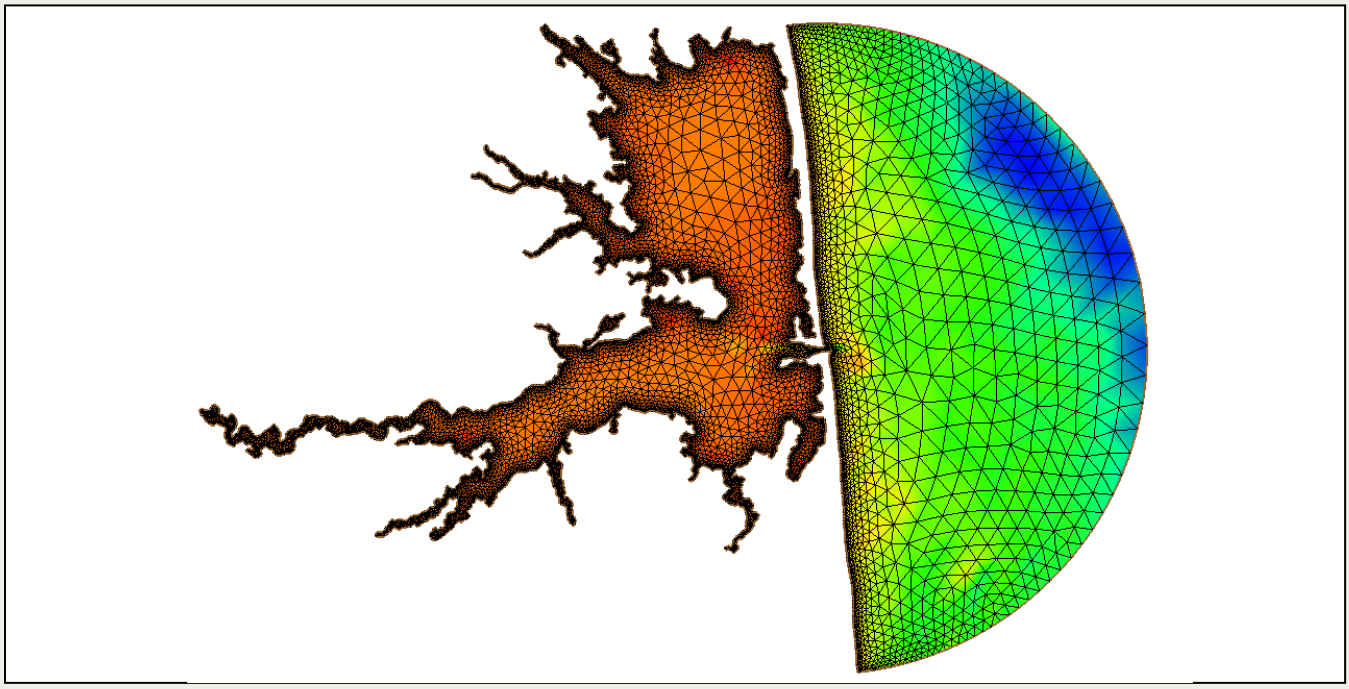
SMS 14.0 Tutorial

Mesh Generation



Objectives

This tutorial demonstrates how to generate a mesh to fill a numeric model domain for a coastal application.

Prerequisite Tutorials

- Overview
- Map Module
- Raster
- Background Data
- Simulation Domain

Required Components

- SMS Core

Time

- 15–20 minutes

AQUAVEO™

1 Introduction

This exercise illustrates how to define a mesh or unstructured grid for use with a numeric model such as ADCIRC or SCHISM. This mesh creation is based on dividing the model domain into one or more sub-areas and filling these areas in with cells (finite difference terminology) or elements (finite element terminology).

This exercise utilizes data from the Delaware coast.

2 Getting Started

Start by importing a project file containing an existing mesh:

1. Launch the SMS application.
2. From the *Menu* bar, select the *File* | **Open…** menu item to bring up the *Open* dialog.
3. In the *Open* dialog, from the *Files of type* drop-down menu, select "Project Files (*.sms)".
4. Navigate to the *Exercises/Indian River* folder.
5. Select the “IR_domain.sms” file.
6. Click the **Open** button to import the project and exit the *Open* dialog.

The initial project should appear similar to Figure 1.

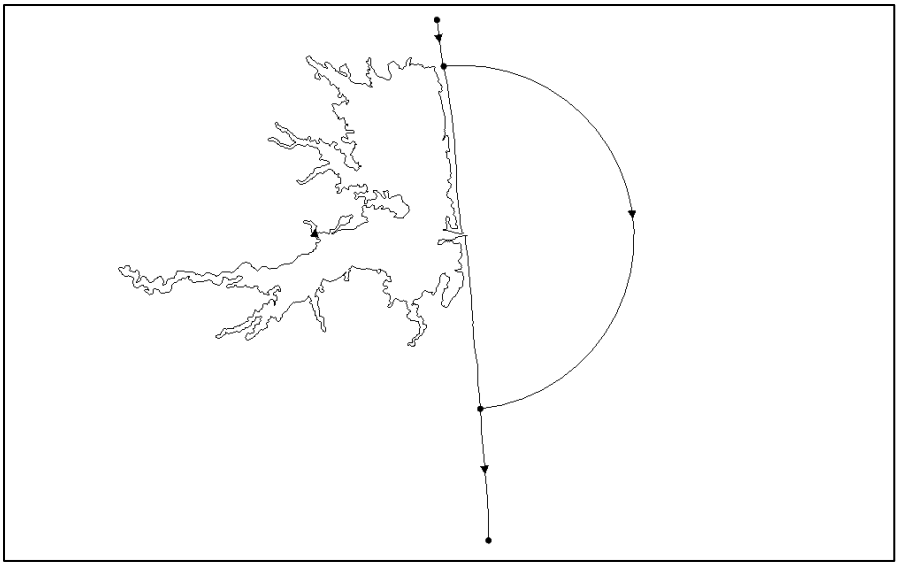


Figure 1: Initial Project

3 Defining Mesh/Grid Density

SMS can create elements/cells for numerical calculations from the arcs that make up the domain boundary. The most intuitive approach is to assign cell/element size based on the distribution of vertices around the boundary. This method can also include interior features (arcs inside the domain) with vertex spacing specified along those arcs.

SMS includes many options for distributing vertices along an arc which are described in the detail at https://www.xmswiki.com/wiki/SMS:Arcs#Redistribute_Vertices as well as illustrated in various SMS tutorials (*Map Module, Mesh Generation, Mesh Generation from Extracted Features, …*) Two variations are illustrated in this section.

3.1 Redistributing the coastline

For this example, we will use a constant 50 m spacing along the coastline. To do this:

1. In the *Project Explorer,* in *Map Data* folder, click on the "IR domain" coverage to make it active.
2. Right-click on the "IR domain" coverage and select the **Duplicate** context menu item to create a copy of this coverage.
3. Right click on the "IR domain (2)" coverage and select the **Rename** context menu item.
4. Enter the name "constant pave".
5. Select the "constant pave" coverage to make it active.
6. From the *Dynamic Tool* bar, select the **Select Feature Arc** tool,
7. From the *Graphics Window,* select the coastline arc.
8. Right-click on the arc and select the **Redistribute Vertices…** context menu item. The *Redistribute Vertices* dialog appears. This dialog displays the range of

segment lengths in the selected arc, see Figure 2. Note the large range resulting from the extraction and pruning process.

9. In the *Arc Redistribution* section of the dialog, set the value in the *Specify* field to "Specified spacing".
10. In the *Average spacing* field, enter "50.0".
11. Click the **OK** button to redistribute the vertices and close the *Redistribute Vertices* dialog.

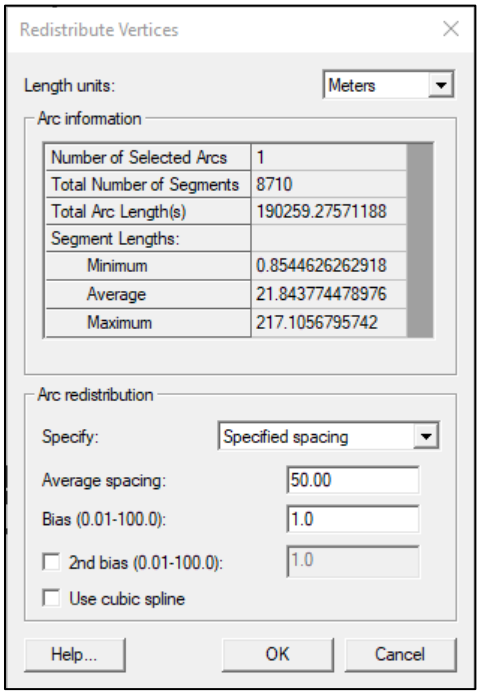


Figure 2: Specified Spacing Redistribution

SMS will redistribute the vertices to attempt to match the specified 50 m goal. It may not be able to do this precisely because of the curvature and overall length of the arc. Repeating the command can slightly improve the distribution as areas of high curvature or where arcs bend back on themselves are smoothed out. You can also reissue the command to review the spacing after each operation.

3.2 Redistributing the Ocean Arc

In the ocean area of the domain, we would like to transition from 50 m spacing near the coastline (consistent with the coastline spacing) to 1000 m at the farthest point offshore. In large domains, the maximum spacing may be 50 km or more. This domain is small, so the spacing used here is not that extreme.

To obtain this type of variable vertex spacing:

1. From the *Menu* bar, select the *Display* | **Display Options…** menu item to open the *Display Options* dialog.
2. From the list on the left, select the **Map** item.
3. From the section on the right, turn on the *Vertex* checkbox to instruct SMS to display vertices.

4. Click the **OK** button to exit the *Display Options* dialog. (Vertices will appear in the display.)
5. From the *Dynamic* toolbar, select the **Select Feature Arc** tool.
6. Select the ocean arc.
7. Right-click on the arc and select the **Redistribute Vertices…** context menu item. The *Redistribute Vertices* dialog appears. Note that the spacing along this ocean is around 1000 m based on the tool that generated this arc for the domain.
8. In the *Arc redistribution* section, in the *Average spacing* field, enter “500.0”. This initial approximation comes from averaging the 50 m and 1000 m target spacing.
9. In the *Bias (0.01-100.0)* field, enter “20.0”. This is the ratio between the smallest and largest spacings (1000/50 = 20).
10. Turn on the *2nd bias (0.01-100.0)* checkbox and enter “20.0” in the data field. This enables a double biasing so that the spacing will increase from both ends of the arc.
11. Click the **OK** button to redistribute the vertices and close the *Redistribute Vertices* dialog. Note that spacing increases as the distance from either coastline increases.
12. Right-click on the arc and select the **Redistribute Vertices…** context menu item again. Note that the spacing along this ocean now varies from around 70 up to 1500 meters. This is not quite the 50-1000 m range desired, but it is close.
13. Change the *Average spacing* field to “325.0”.
14. Click the **OK** button to redistribute the vertices and close the *Redistribute Vertices* dialog again. This value was determined using a trial-and-error approach to the desired 50-1000 m range.

Figure 3 shows the vertices redistributed on the domain boundary. Zoom and pan around the domain to review the results.

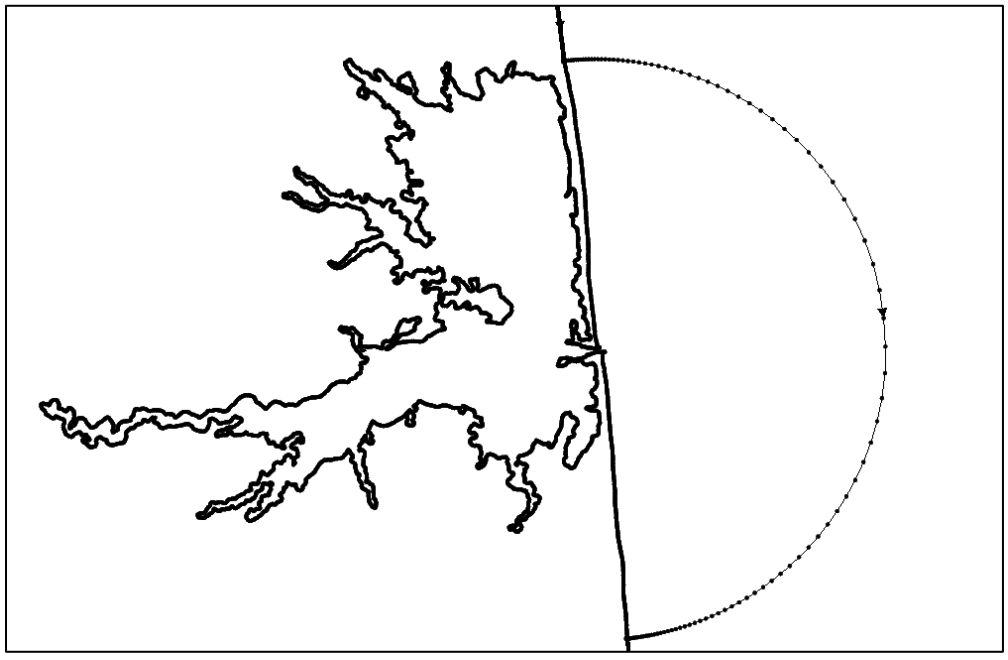


Figure 3: The Redistributed Vertices

You could toggle the “IR domain” coverage ON to compare the spacing now with the initial spacing. You will probably need to zoom in significantly to note the changes.

4 Constant Paving Mesh Generation

We now have the geometric polygon ready to generate a mesh for numerical analysis. We need to set the coverage to be used for mesh generation. To do this:

1. In the *Project Explorer,* right-click on the “ constant pave” coverage to bring up the context menu.
2. Select the *Type* | *Generic* | **Mesh Generator** context menu item. This informs SMS that the data in this coverage is intended to be used to generate a mesh.
3. From the *Macro* bar, click on the **Build Polygon** macro to build the polygon of the domain. This creates the polygons that will be filled with cells/elements.

4.1 Setting Polygon Attributes

The next step in the mesh generation process is to assign attributes to define how SMS should fill each polygon in the coverage with cells/element. To do this:

1. From the *Dynamic* tool bar, select the **Select Feature Polygon** tool, and double-click in the domain polygon. This brings up the *2D Mesh Polygon Properties* dialog.
2. Set the *Mesh Type* drop-down menu to “Constant Paving (triangles)”.

This tells SMS that it should fill the area with triangles, transitioning from the border spacing up to the specified interior spacing.

3. For the *Target Size (m),* enter “1000.0”.
4. Click on the **Preview Mesh** button. SMS will display a preview of how the polygon could be filled with triangles based on these parameters.

The *Bias (0.0-1.0)* defines how fast the elements can change size. The default of 0.3 is generally good starting value, but for some coastal applications a more gradual transition may be desired.

5. Change the *Bias* value to “0.2”
6. Click the **Preview** button again. Note that the elements grow more slowly with this bias.
7. Change the *Elevation (bathy/topo) type* to “Raster”.
8. Click on the **Raster Options…** button to open the *Raster Sets* dialog.
9. Select the “IR_Inlet_DEM_2015.tif”
10. Click the **OK** button to close the *Raster Sets* dialog.
11. Click the **OK** button to close the *2D Mesh Polygon Properties* dialog.

We are now ready to generate the mesh.

This example only has one polygon. If the domain has been subdivided for more explicit control, the steps in this section would be repeated for each polygon.

4.2 Generating the Mesh

To generate the mesh:

1. In the *Graphics Window,* click outside all polygons to clear the selection. This is not needed in this example because there is only one polygon. The mesh generation command processes selected polygons (or all polygons if none are selected).
2. In the *Project Explorer*, right-click on the "  constant pave" coverage and select the *Convert* | **Map** → **2D Mesh** context menu item to open the *Mesh Name* dialog.
3. In the *Mesh Name* dialog, for the *Mesh name,* enter "constant pave 50-1000m".
4. Click the **OK** button to close the *Mesh Name* dialog.

The mesh is now generated and ready for use.

5 Reviewing the Mesh/Grid

With a completed mesh, we will modify some of the display options to better view the mesh.

1. In the *Project Explorer*, right-click in the white space and select the *Uncheck All* context menu item to turn off the display off all layers.
2. In the *Project Explorer*, turn on the checkbox next to the "  constant pave 50-1000m" mesh. The mesh should now be the only item displayed.
3. In the *Project Explorer*, right-click on the  *Mesh Data* folder and select the **Display Options…** context menu item to open the *Display Options* dialog.
4. In the *Display Options* dialog, from the list on the left, select the *2D Mesh* option
5. On the right side of the dialog, in the *2D Mesh* tab, click the **All Off** button at the bottom to turn off all the components of the mesh.
6. Turn on the *Elements* and *Contours* checkboxes.
7. Select the *Contours* tab.
8. Change the *Contour method* drop-down to "Color Fill".
9. Click on the **Color Ramp…** button to bring up the *Choose color ramp* dialog.
10. From the *Aquaveo* folder, select the *Atlas Shader* color ramp.
11. Click the **OK** button to close the *Choose color ramp* dialog.
12. Click the **OK** button to close the *Display Options* dialog.
13. Use the *Rotate*  and *Zoom*  tools to rotate and zoom into different areas of the mesh.

Notes:

- This color ramp assigns dark blue to the deepest water while red represents the shallow areas. You could experiment with other color ramps.
- The resolution of this mesh allows for 3 elements through Indian River Inlet. This would be a minimum amount to convey flow for most numeric engines. Similarly,

the prune operation (in the previous exercise) eliminated all internal branches of the bay that would result in fewer than 3 elements with the specified spacing.

- Refer to the XMSWIKI or other SMS tutorials to learn about other mesh display options such as shading.

Spend a few minutes examining the variation in element size, the elevations that have The final mesh should look similar to the one shown in Figure 4:

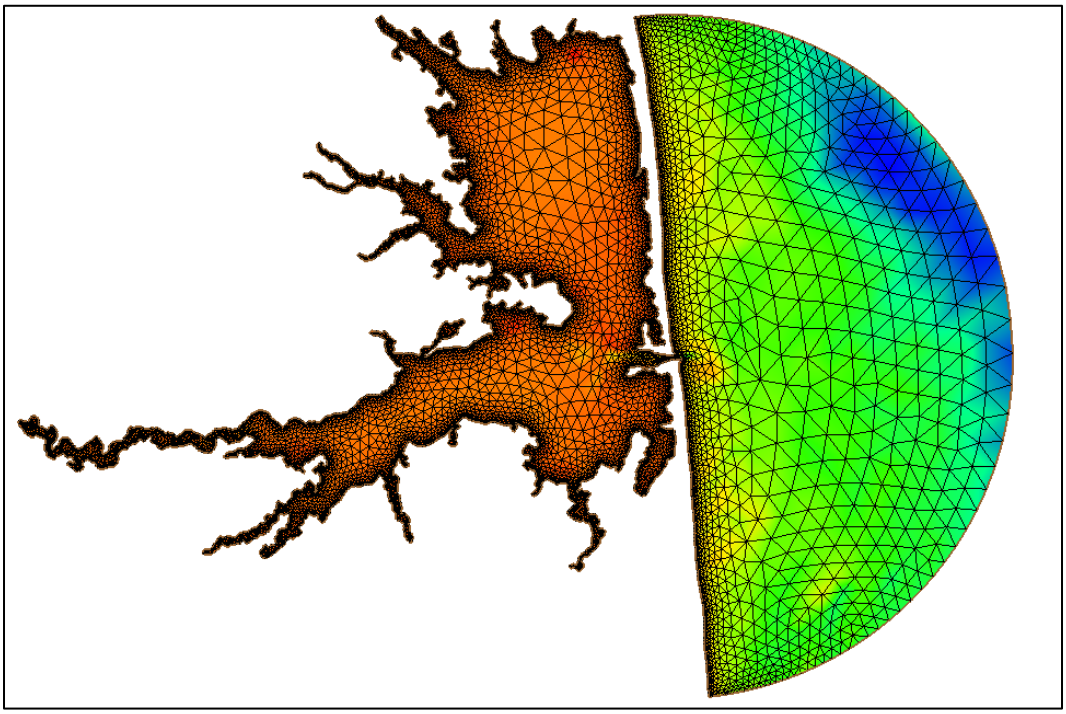


Figure 4: The Constant Paved Mesh

6 Saving the Project

To save the project:

1. From the *Menu* bar, select *File* | ***Save As...*** to open the *Save As* dialog.
2. Navigate to the *Exercises/Indian River/work* folder.
3. For the *File name,* enter "IR_mesh.sms"
4. Click the **Save** button to save the file and close the *Save As* dialog.

7 Conclusion

This concludes the “Mesh Generation” exercise. Several features have been demonstrated including:

- Vertex redistribution using specified size and bias factors.
- Constant paving.
- Mesh display options.
- Changing the display projection.

If time permits, you may wish to try experimenting with:

- Different values of vertex spacing for redistribution.
- Different constant paving options.
- Graphical interaction within SMS to review the mesh.