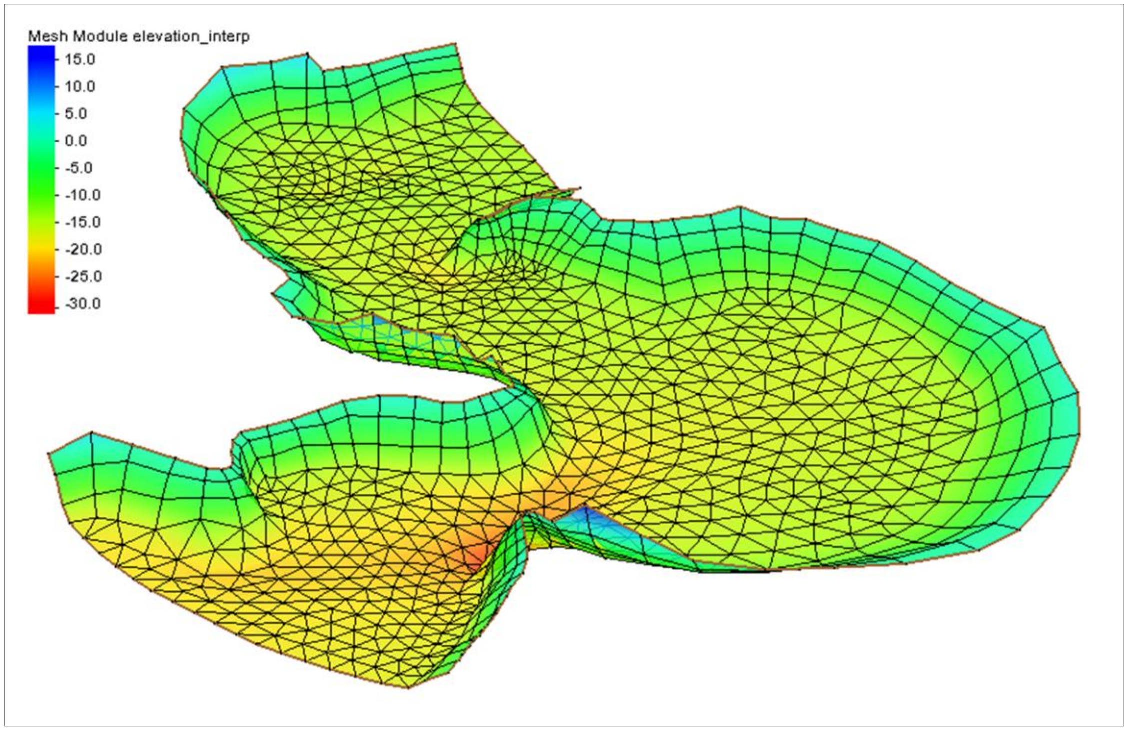
SMS

v. 12.3

SMS 12.3 Tutorial

Mesh Generation



Objectives

This tutorial demostrates the fundamental tools used to generate a mesh in the SMS.

Prerequisites

- SMS Overview
- SMS Map Module

Requirements

- Mesh Module
- Scatter Module
- Map Module

Time

- 15–20 minutes

AQUAVEO™

1 **Getting Started** ... 2
1.1 Opening an Existing Project ... 2
2 **Closing the Domain** ... 3
3 **Generating a Paved Mesh** ... 4
4 **Revising the Size Transition** ... 6
5 **Subdividing the Domain** ... 7
6 **Defining a Patch** ... 9
7 **Sample Mesh Generation** ... 11
8 **Editing the Generated Mesh** ... 12
9 **Interpolating to the Mesh** ... 13
10 **Saving a Project File** ... 15
11 **Conclusion** ... 15

1 Getting Started

This tutorial covers some of the basic methods used to generate a mesh using the SMS. The functionality spans tools in the map module, mesh module and TIN module.

The "SMS Overview" and "Map Module" tutorials should be completed before this tutorial. All files for this tutorial are found in the "data files" folder within the "Mesh Generation" folder.

In this example, a mesh consists of randomly spaced and positioned nodes connected to form elements or cells with three or four sides. The sides can be linear, or quadratic. The sides of a quadratic cell include a midside node to define the shape of that side. Any number of cells can be connected at a single node. However, well formed grids (meshes) restrict the number of cells connected to a node to be between four and eight. SMS also restricts the creation of cells with interior angles greater than 180 degrees. Such a node in a cell/element is called a concave node and is illegal.

A mesh could also be referred to as an unstructured grid.

Before starting this tutorial, do the following:

1. Start SMS
2. If SMS is already open, select *File* | **Delete All**.
3. Click **Yes** to clear all data in SMS.

1.1 Opening an Existing Project

Begin with opening an existing project that contains a defined domain for the mesh.

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Select the file "start.sms" from the "data files" folder in the "Mesh Generation" folder.
3. Click **Open** to import the project file.

This project consists of two arcs on either side of a channel off of Chesapeake Bay (Figure 1). The arcs have been created on a map coverage with the Mesh Generator type. This tutorial describes methods of creating a mesh covering this region.

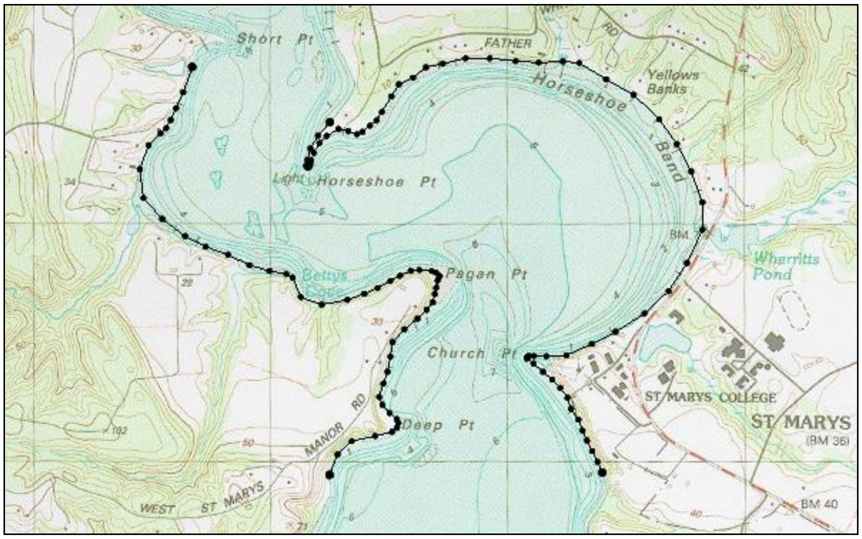


Figure 1 Project area

2 Closing the Domain

As described in the “Map Module” tutorial, a conceptual model requires the domain be defined as polygons. Before doing this, duplicate the original coverage.

1. Right-click on the “ Channel” coverage and select **Duplicate** *to* create a copy of the coverage.
2. Right-click on the “ Channel (2)” coverage and select **Rename**.
3. Enter the name “Domain” for the new coverage then press *Enter*.
4. Select the “ Domain” coverage to make it active.

Ideal meshes consist of equilateral triangles and rectangular quadrilaterals. That means that corners formed between arcs that define polygons in the coverage are ideally close to 90, 120, 150, or 180 degrees. Normally angles less than 90 degrees should be avoided to prevent isolated cells/elements from being formed. Therefore, when creating arcs to close the domain in this case, the guideline would be to attempt to create square 90 degree corners in the corners of the domain. To do this:

1. Using the **Create Feature Arc** tool, click on the node at the top of the right bank arc (point A in the upper left side of Figure 2) to start creating an arc.
2. Click to create a vertex roughly one third of the way across the channel at as close to a right angle as possible (point B in Figure 2).
3. Click to create a second vertex roughly two thirds the way across the channel that is perpendicular to the first segment on the left bank (point C in Figure 2).
4. Click on the node as the top of the left bank arc (point D in Figure 2) to complete the arc and close off the upsteam end of the river.

5. Repeat steps 1–4 to create an arc that closes off the downstream end of the channel as seen in Figure 2.

   Note: the location of the vertices and nodes can be updated at any time by selecting and dragging or editing the values.

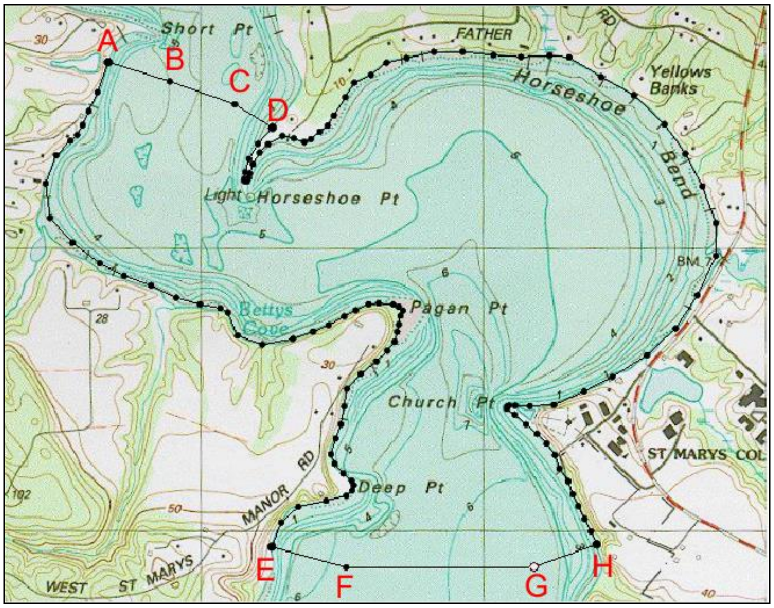


Figure 2 Initial domain

To build the polygon:

1. Choose the **Select Feature Arc** tool from the toolbox to make it active.

This also clears the selection list. When building polygons it is best to have no selections.

2. Select *Feature Objects* | **Build Polygons**.

Notice that **Select Polygon** tool becomes available or un-dimed indicating polygons are now included in the coverage.

3 Generating a Paved Mesh

The first mesh generation method illustrated in this tutorial generates an all triangle cell mesh from a method referred to in SMS as “Paving”. This method principally creates triangles of various sizes based on the vertex distribution of the boundary arcs. To generate a paved mesh:

1. Using the **Select Feature Arc** tool, click on the arc on the bottom of the channel.
2. Hold the *Shift* key down and click on the arc at the inflow (top) of the channel.
3. Select *Feature Objects* | **Redistribute Vertices** to open the *Redistribute Vertices* dialog.

The *Redistribute Vertices* dialog shows information about the feature arc segments and vertex spacing.

4. Turn on the option to *Use Cubic Spline* at the bottom of the dialog.

This will cause the redistribution to follow a curve fit through the initial arc points.

5. Make sure the "Specified spacing" option is selected for *Specify* and enter a value of "200" for *Average*.

This tells SMS to create vertices 200 feet apart from each other (or 200 meters apart if working in metric units).

6. Click **OK** to close the dialog and redistribute the vertices along the arcs.

After redistributing vertices, duplicate the " Domain" coverage so it can be used again later in the tutorial.

7. Right-click on the " Domain" coverage and select **Duplicate**.
8. Right-click on the " Domain (2)" coverage and select **Rename**.
9. Enter the name "Paved1" and press *Enter*.
10. Select the " Paved1" coverage to make it active.

The meshing parameters can now be assigned to the polygon.

11. Using the **Select Polygon** tool, double-click inside the polygon to open the *2D Mesh Polygon Properties* dialog (Figure 3).

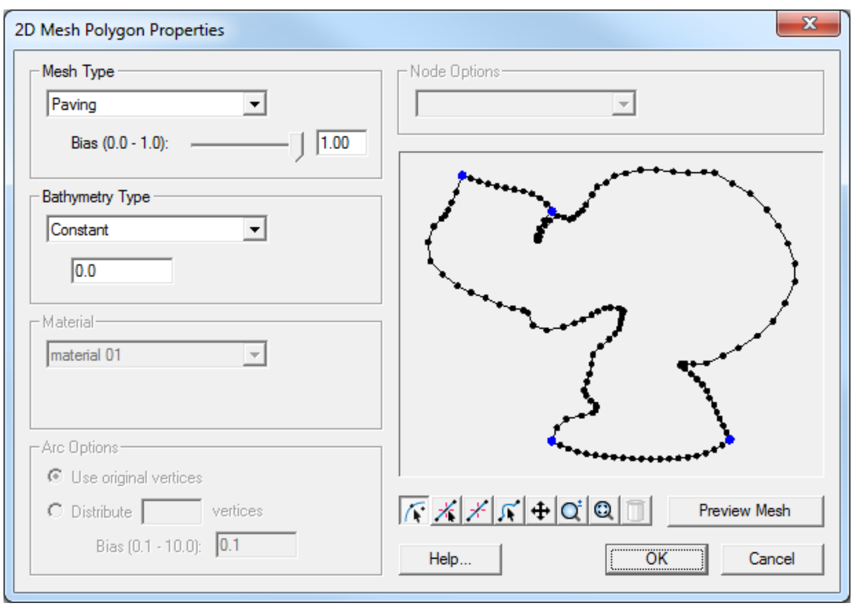


Figure 3 2D Mesh Polygon Properties dialog

12. Ensure that the *Mesh Type* is "Paving".
13. Click **Preview Mesh** to see what the elements will look like.
14. Click **OK** to close the *2D Mesh Polygon Properties* dialog.

15. Right-click on the " Paved1" coverage and select *Convert* | **Map → 2D Mesh** to open the *2D Mesh Options* dialog.

The dialog contains options for the mesh generation including quadratic verses linear elements and an option to merge triangles into quadrilaterals.

16. Click **OK** to accept the default options and close the *2D Mesh Options* dialog.

With the mesh generation options chosen, SMS now generates a mesh. Before completing the mesh generation, the *Mesh Name* dialog will appear.

17. Click **OK** to accept the default name and close the *Mesh Name* dialog.

The mesh should resemble Figure 4. A new item, " Paved1 Mesh" should now appear in the Project Explorer.

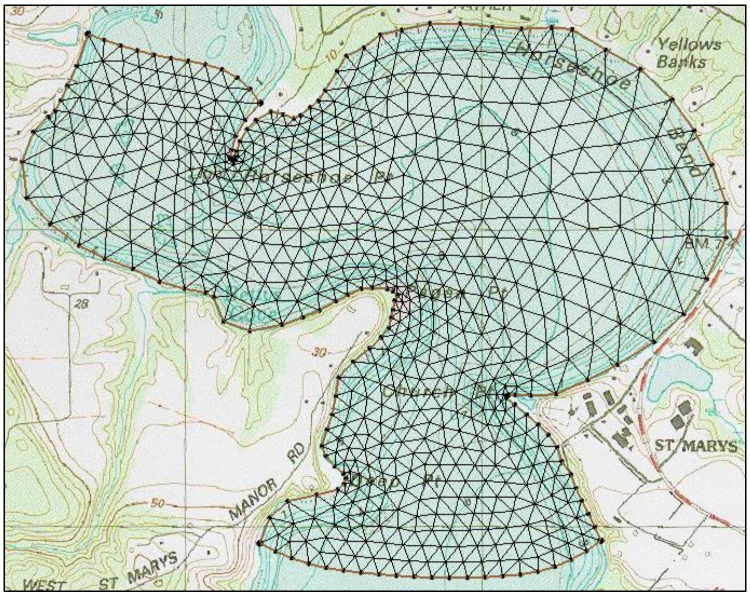


Figure 4 A paved mesh

Use the **Pan** and **Zoom** tools to examine the variation in cell size.

4 Revising the Size Transition

If the variation in element size is large, a more gradual transition from small cells to large cells can result in better cell shape. To illustrate this:

1. Right-click on the " Paved1" coverage and select **Duplicate**.
2. Right-click on the " Paved1 (2)" coverage and select **Rename**.
3. Enter "Paved2" and press *Enter*.
4. Select the " Paved2" coverage to make it active.
5. Using the **Select Polygon** tool, double-click inside the polygon to open the *2D Mesh Polygon Properties* dialog.

6. Set the *Bias (0.0 – 1.0)* variable to “0.0”.

This sets the transition to the most gradual.

7. Click **OK** to close the *2D Mesh Polygon Properties* dialog.
8. Right-click on the “ Paved2” coverage and select *Convert* | **Map → 2D Mesh** to open the *2D Mesh Options* dialog.
9. Click **OK** to close the *2D Mesh Options* dialog and open the *Mesh Name* dialog.
10. Click **OK** to close the *Mesh Name* dialog.
11. **Zoom** into the area around Light Horseshoe Point at the inside of the first bend in the channel.
12. Select back and forth between the two meshes to see the difference between the results.

The “ Paved2 Mesh” has smaller, better elements around the tight bend (Figure 5).

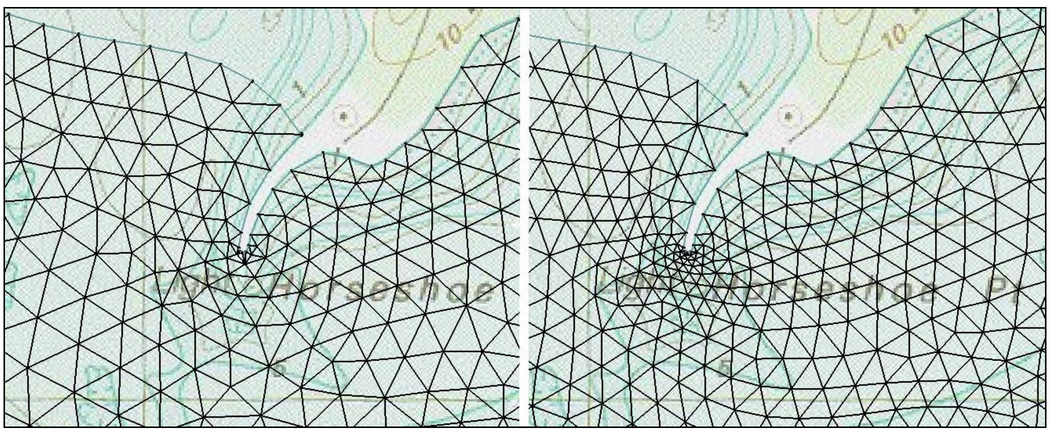


Figure 5 Mesh on the right has been generated with a more gradual transition

13. Uncheck the boxes next to “ Paved1 Mesh” and “ Paved2 Mesh” to turn the display off while further options are demonstrated.

5 Subdividing the Domain

It is often useful to refine a domain along boundaries with steep slopes or where wetting and drying may occur as the water level changes. Increasing the resolution throughout the domain results in an unnecessarily large number of elements. Local refinement creates a more economical solution.

One method of locally refining a mesh includes subdividing the domain into multiple polygons. In this example, the subdivision will isolate the sloped banks at the edges of the channel. To create this subdivision:

1. Right-click on the “ Domain” coverage and select **Duplicate**.
2. Right-click on the “ Domain (2)” coverage and select **Rename**.
3. Enter “Left Toe” and press *Enter*.

4. Select the “Left Toe” coverage to make it active.

Notice in the image that there is a cluster of contours close to the banks. The idea is to create an arc along the “toe” of this slope to better represent the shape. For the left bank, that last clear contour in the cluster is the 4 meter contour.

1. Using the **Create Feature Arc** tool, click out an arc that starts at the intersection of the upstream arc and the 4meter contour and follows the 4 meter contour along the left bank as seen in Figure 6.

Don’t worry about the density of the vertices at this point, just try to make a smooth arc. Two adjacent segments should not vary in direction by more than 20–30 degrees. At the second sharp bend at Church Pt., transition out to the 5meter contour since the channel has become deeper.

> ***Hint***: Digitizing accurately may require zooming in. Using the mouse wheel to zoom and the mouse wheel button to pan digitizing simplifies this process.

2. Terminate the arc by clicking on the downstream arc. (See Figure 6.)

Because a new arc now exists the polygon is out of date.

3. Select the **Select Feature Arc** tool to clear the selection list.
4. Select *Feature Objects* | **Build Polygons**.

There should now be two distinct polygons that can be selected.

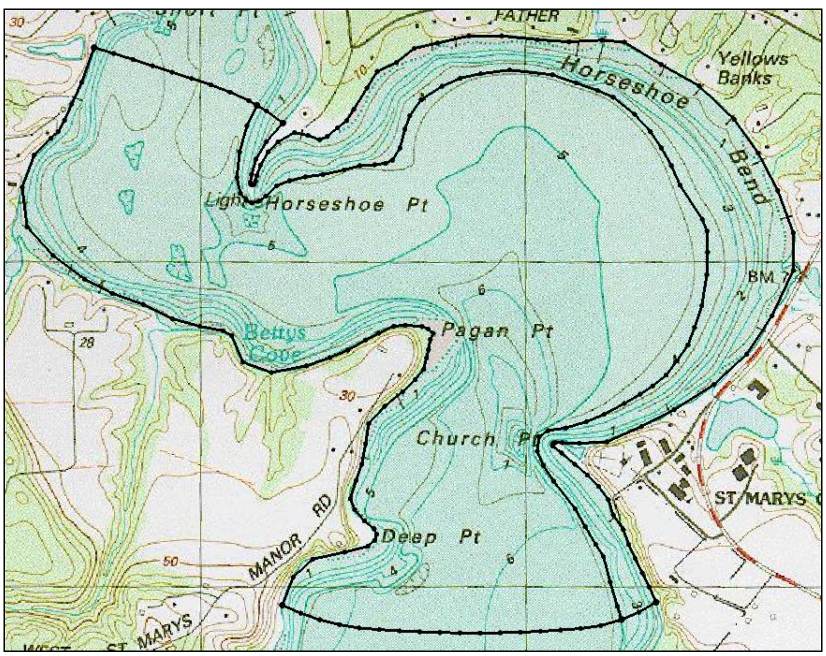


Figure 6 Arc along the left bank

6 Defining a Patch

In addition to the paving approach illustrated above, SMS supports a patching approach for generating cells/elements in a polygon. The patch mesh generation method requires three or four sided regions to be created. However, it is not uncommon to use the patching technique to fill a polygon defined by more than four arcs. Figure 7 shows an example of a rectangular patch made up of four sides. Note that Side 1 and Side 2 are both made from multiple feature arcs.

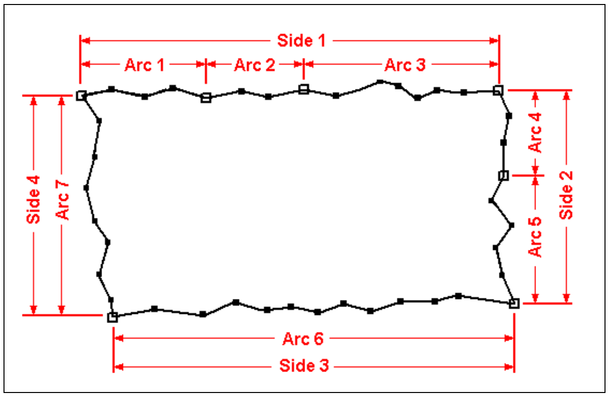


Figure 7 Four sides required for a rectangular patch

The four sided region can be bent, warped or otherwise deformed. However, the generation of quality elements/cells general requires that the number of divisions on the opposing edges be consistent. That means that if there are 10 edges on side 1, better elements will result if there are also 10 edges on side 3. If the polygon is long and skinny, transitions on the short edge can be applied and still result in well-formed elements.

In this case the elevations transitions quickly from the outer edge to the toe of the bank, so at least 3 elements should be used to represent that variation. To specify this:

1. Right-click on the " Left Toe" coverage and select **Duplicate**.
2. Right-click on the "Left Toe (2)" coverage and select **Rename**.
3. Enter "Patch1" and press *Enter*.
4. Select the " Patch1" coverage to make it active.
5. Using the **Select Feature Arc** tool, select the left bank arc on the far right side of the Graphic Window.

Note at the bottom of the screen that the "id" of the arc is 25.

6. Hold the *Shift* key down and click on the arc at the toe of the bank (the arc created in Section 5).
7. Select *Feature Objects* | **Redistribute Vertices** to open the *Redistribute Vertices* dialog.
8. Turn on the option to *Use Cubic Spline* at the bottom of the dialog.
9. Select the "Source Arc" option for the *Specify* option.

This redistribution method distributes vertices on one arc to match the spacing of another arc.

10. Set the *Source Arc* to “25”.

This changes the *Target Arc* to be “28” (the toe arc) and will use the spacing on arc 25 to distribute vertices on arc 28.

11. Click **OK** to close the *Redistribute Vertices* dialog and to redistribute the vertices along the arc 28.
12. Select the two arcs that connect the left bank and toe arcs at the top and bottom.
13. Right-click on either of the two selected arcs and select **Redistribute Vertices** to open the *Redestribute Vertices* dialog again.
14. Select the “Number of segments” option for *Specify*.
15. Enter “4” in the *Number of segment* field.

This instructs SMS to position three vertices along the arc (to create four segments).

16. Click **OK** to close the *Redistribute Vertices* dialog and to redistribute the vertices along the arcs to request four elements across the bank.

To define a patch, follow these steps:

1. Using the **Select Feature Polygon** tool, double-click on the bank polygon to open the *2D Mesh Polygon Properties* dialog.
2. Select the “Patch” option from the *Mesh Type* drop down list.
3. Click the **Preview Mesh** button to see a preview of how the elements in the patch will be created (Figure 8).

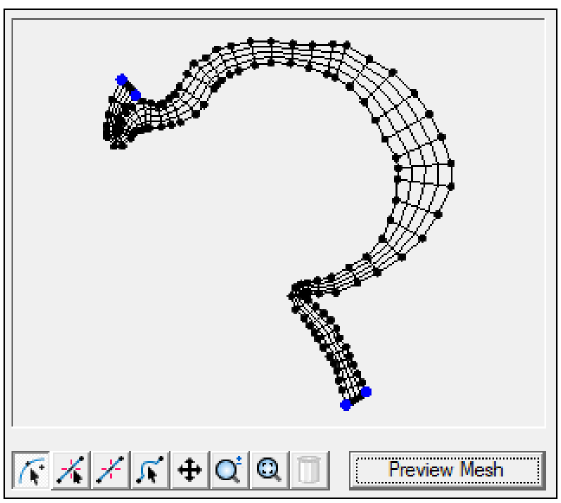


Figure 8 Mesh preview

4. Click the **OK** button to close the *2D Mesh Polygon Attributes* dialog.
5. Right-click on the “ Patch1” coverage and select *Convert* | **Map → 2D Mesh** to open the *2D Mesh Options* dialog.

6. Accept the default options and click **OK** to close the *2D Mesh Options* dialog, generate the mesh, and open the *Mesh Name* dialog.
7. Accept the default name and click **OK** to close the *Mesh Name* dialog.

The new mesh shows patch elements along the bank polygon (Figure 9).

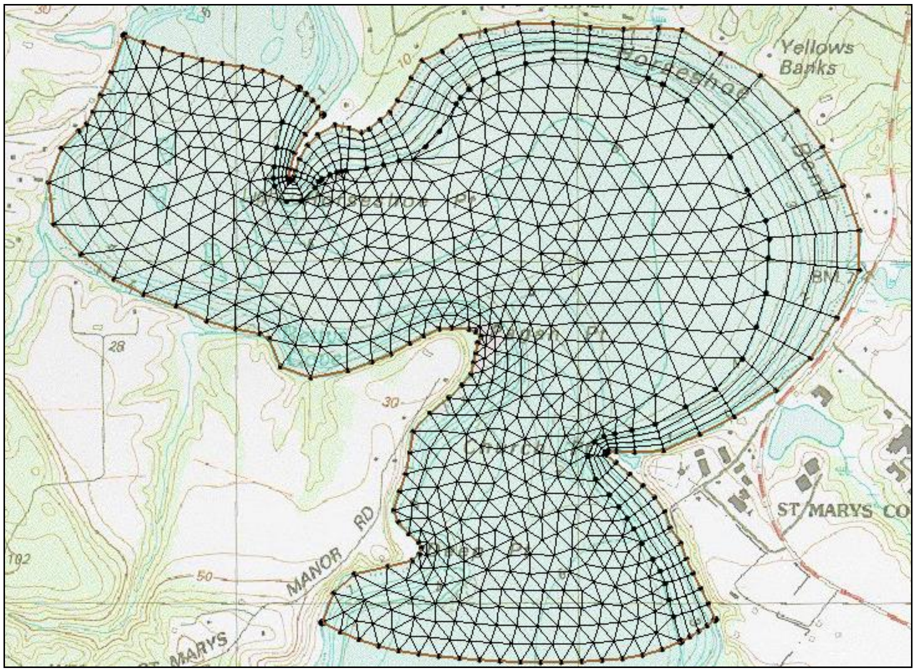


Figure 9 Mesh with both paved and patch methods

8. Use the **Pan** and **Zoom** tools to examine the resulting mesh.
9. Uncheck the box next to " Patch1 Mesh" to hide the display before continuing.

7 Sample Mesh Generation

This tutorial has demonstrated several options for mesh generation, but many more are available. These are discussed in detail in other tutorials or help documents. To illustrate a completed conceptual model:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Select the file "StMary.map" from the "data files" folder.
3. Click **Open** to import the file.

SMS opens the file and a new coverage named " StMary" appears.

4. Right-click on the " StMary" coverage and select *Convert* | **Map → 2D Mesh** to open the *2D Mesh Options* dialog.
5. Accept the defaults and click **OK** to close the *2D Mesh Options* dialog.

6. When the *Mesh Name* dialog appears, accept the default name of “StMary Mesh” and click **OK** to finish generating the mesh.

The mesh should appear similar to Figure 10. Review the objects in “ StMary” coverage and the impact each has on the “ StMary Mesh” mesh. These objects include:

- A refine point in a paved polygon that specifies the resolution around the point.
- An interior arc in a paved polygon that defines local resolution.
- Multiple patches with transition in one direction.

Select the arcs, feature point and polygons to examine the specified attributes.

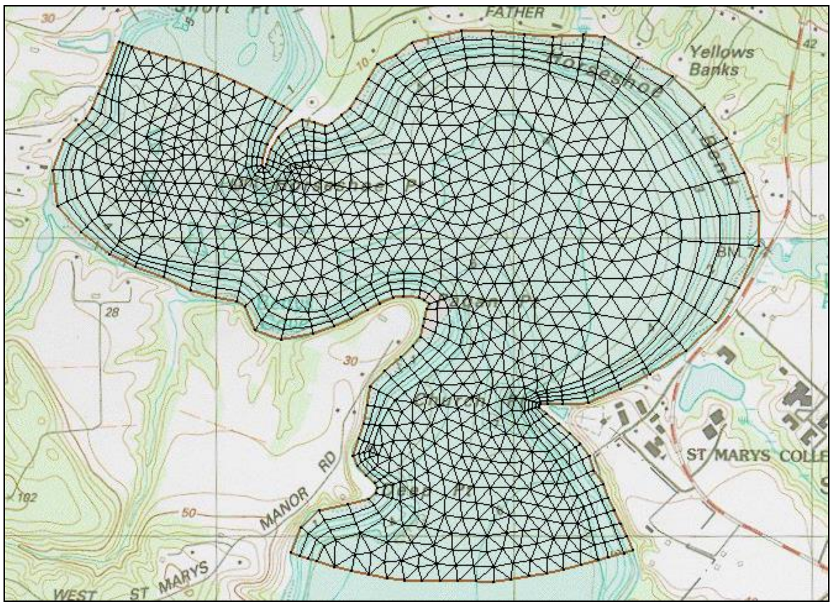


Figure 10 Sample mesh

8 Editing the Generated Mesh

When a finite element mesh is generated from feature objects, it sometimes has aspects that need adjustment. An easy way to edit the mesh is to change the meshing parameters in the conceptual model, such as the distribution of vertices on feature arcs or the mesh generation parameters. Then the mesh can be regenerated according to the new parameters.

If there are only a few changes desired, they can be edited manually using tools in the mesh module. These tools are described in *SMS Help* in the section on the Mesh module as well as in the “Mesh Editing” and additional tutorials.

9 Interpolating to the Mesh

The finite element mesh generated from the feature objects in this case only defined the (x, y) coordinates for the nodes. This is because the bathymetric data had not been read in before generating the mesh. Normally, read in the survey data, and associate it with the polygons to assign bathymetry to the model. However, to illustrate how to update bathymetry for an existing mesh, this section is included.

Bathymetric survey data, saved as scatter points, can be interpolated onto the finite element mesh.

To open the scattered data, do the following:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Locate the file “stmary_bathy.h5” in the “data file” folder.
3. Click **Open** to import the file.

The screen will refresh, showing a set of scattered data points (Figure 11). Each point represents a survey measurement. Scatter points are used to interpolate bathymetric (or other) data onto a finite element mesh. Although this next step requires manually interpolating the scattered data, this interpolation can be set up to automatically take place during the meshing process.

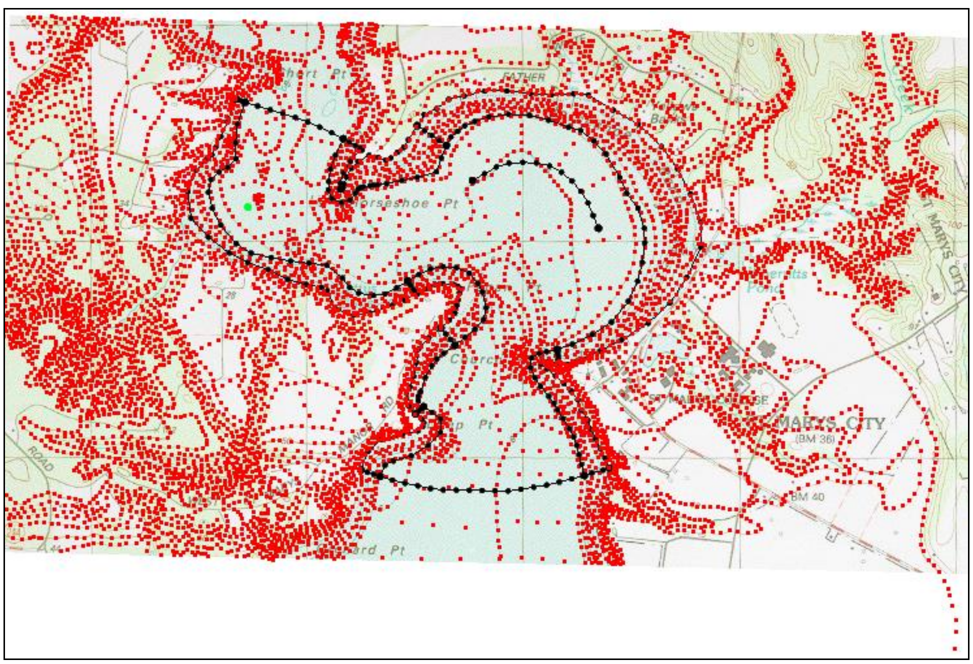


Figure 11 Imported scatter set

To interpolate the scattered data onto the mesh:

4. Make sure the “ stmary_bathy” scatter module is active.
5. Select *Scatter* | **Interpolate to Mesh** to open the *Interpolation Options* dialog.

6. There is only one dataset to interpolate from, so "elevation" will be the source.
7. Select *Map Z* on the left side of the dialog to instruct SMS to make this the elevation (Z) dataset.
8. Click the **OK** button to close the *Interpolation Options* dialog and perform the interpolation.

After this is completed, a new dataset, "elevation_interp" will appear in the Project Explorer under the "StMary Mesh" item in the "Mesh Data" folder.

The scattered data is triangulated when it is read into SMS and an interpolated value is assigned to each node in the mesh. The *Map Z* option causes the newly interpolated value to be used as the nodal *Z*-coordinate.

As with the feature objects, the scattered data will no longer be needed and may be hidden or deleted.

9. Uncheck the box next to the scatter set named "stmary_bathy" in the Project Explorer.

To see if the scatter data was correctly interpolated to the mesh, do the following:

10. Select "StMary Mesh" to make it active.
11. Select the **Rotate** tool then click and drag in the Graphics Window to view how the elevation data was interpolated to the mesh.
12. Click on the **Plan View** macro when done rotating the mesh.

Bathymetry interpolation to a mesh can also be specified when generating the mesh as a polygon attribute if the bathymetry data is already loaded.

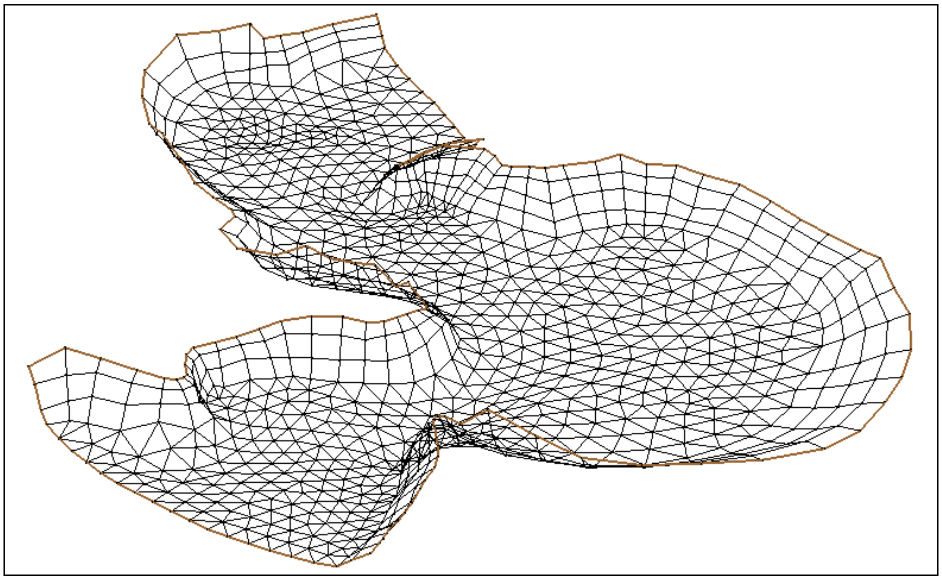


Figure 12 Mesh with interpolated elevation

10 Saving a Project File

Much data has been opened and changed, but nothing has been saved yet. The data can all be saved in a project file. When a project file is saved, separate files are created for the map, scatter data, and mesh data. The project file is a text file that references the individual data files.

To save all this data for use in a later session:

1. Select *File* | **Save as..**. This will bring up a *Save* dialog.
2. Enter a *File name* of “StMaryOut.sms” then click the **Save** button to save the files.

11 Conclusion

This concludes the “Mesh Generation” tutorial. Topics covered in this tutorial included:

- Generating a paved mesh.
- Using the bias option in paving.
- Subdividing a domain.
- Distributing vertices on an arc from a source arc.
- Distributing vertices on an arc to a specified number of segments
- Assigning patch parameters
- Interpolating bathymetry onto a mesh
- Saving a project file

Continue to experiment with the SMS interface or quit the program.