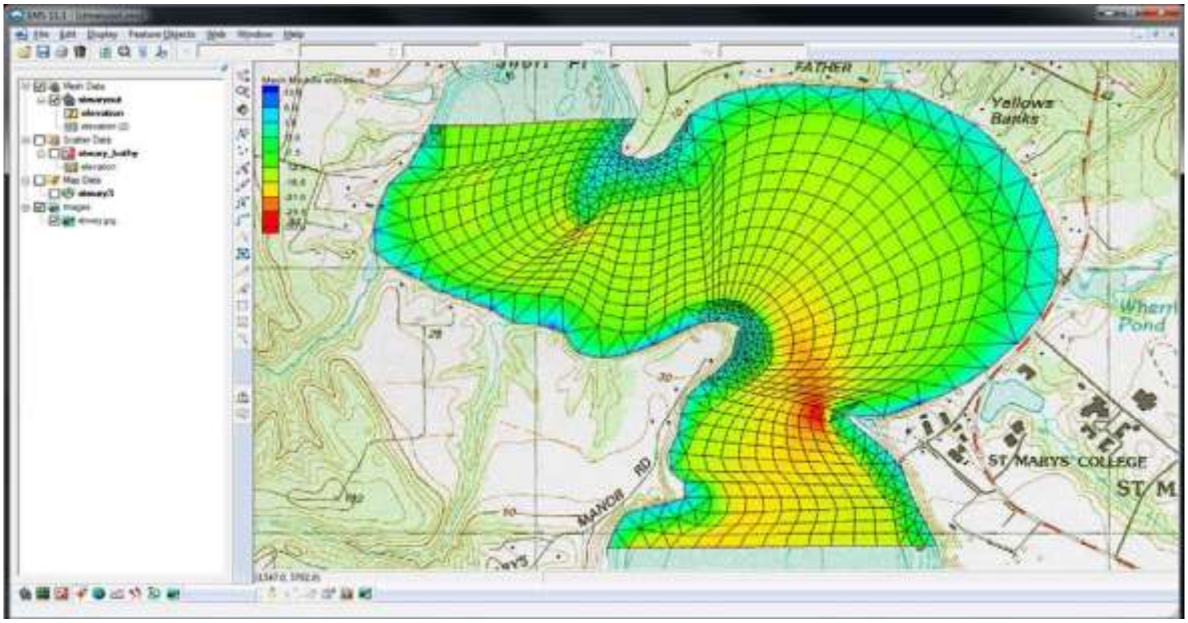
SMS

v. 12.0

SMS 12.0 Tutorial

Overview



Objectives

This tutorial describes the major components of the SMS interface and gives a brief introduction to the different SMS modules. Ideally, this tutorial should be completed before any other tutorial. All files for this tutorial are found in the "data files" folder within the "SMS_Overview" folder.

Prerequisites

- None

Requirements

- Generic 2D Mesh
- TABS or FESWMS
- Mesh Module
- Scatter Module
- Map Module

Time

- 45–60 minutes

AQUAVEO™

1 Getting Started

Before beginning this tutorial, SMS should have been installed. If SMS has not yet been installed, please do so before continuing.

Each chapter of this tutorial document demonstrates the use of a specific component of SMS. If not all modules of SMS have been purchased, or if evaluating the software, run SMS in Demo Mode to complete this tutorial (see section 3 of the “SMS Intro” tutorial).

When using *Demo Mode*, it will not be possible to save files. For this reason, all files that are to be saved have been included in the output subdirectory under the *tutorial\SMS_Overview\data files* directory. When asked to save a file, instead open the file from this output directory.

To start SMS, do the following:

1. Open the *Start* menu.
2. Go to *All Programs*.
3. Click on the “SMS 12.0” folder.

4. Click on **SMS 12.0**.
5. Alternatively, a shortcut icon may already be on the desktop if that option was selected during installation. If so, simply click on that icon.

2 Requirements

In order to complete this tutorial, the Generic Model and either TABS (RMA2 or RMA4) or FESWMS interface must be available under the current SMS license. To check if these models are enabled, click on the *Help* menu in SMS and select **Register**. A list of components and the status of each are displayed in the dialog that appears. Toggle off *Show only enabled modules* to show both enabled and disabled components.

If desired, change the registration according to the project needs by clicking on the **Change Registration** to bring up the *Registration Wizard.* Complete the steps in the *Registration Wizard* dialog to change the registered components.

3 The SMS Screen

The SMS screen is divided into six main sections: the Main Graphics Window, the Project Explorer (this may also be referred to as the Tree Window), the Toolbars, the Edit Window, the Menu Bar and the Status Bars, as shown in Figure 1. Normally the Main Graphics Window fills the majority of the screen. However, plot windows can also be opened in this space to display 2D plots of various data.

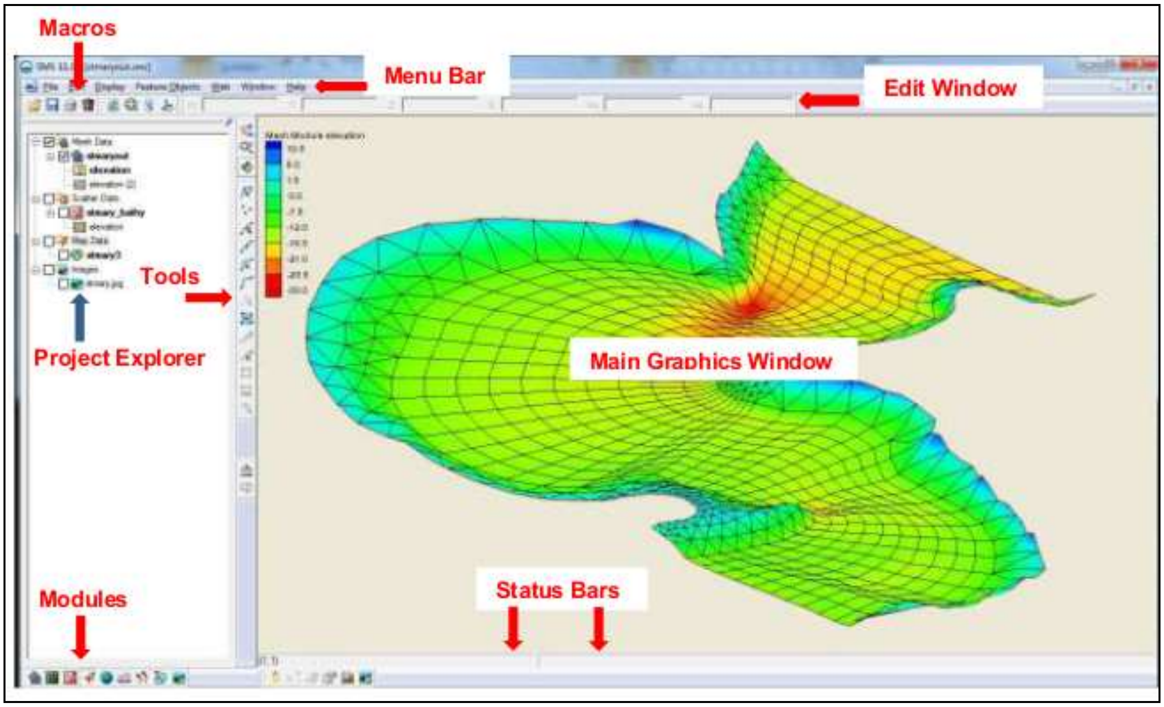


Figure 1 The SMS screen

3.1 The Main Graphics Window

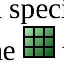
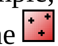
The Main Graphics Window, or just Graphics Window, is the biggest part of the SMS screen. Most of the data manipulation is done in this window. This will be used with each tutorial chapter.

3.2 The Toolbars

Toolbars are dockable. By default, they are positioned at various locations on the left side of the application, but can be positioned around the interface as desired. The macro toolbars that appear at startup are set in the *Preferences* dialog under the *Toolbars* tab (Edit | **Preferences** command or right-click in the Project Explorer and select **Preferences**).

The toolbars include the following:

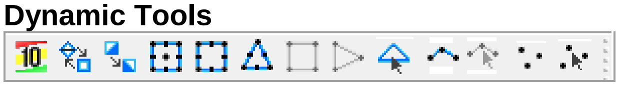
Modules

This image shows the current SMS Modules. As described in the *SMS Online Help*, these icons control what menu commands and tools are available at any given time while operating in SMS. Each module corresponds to a specific type of data. For example, one  icon corresponds to finite element meshes, one  to Cartesian grids, and one  to scattered data. If the scattered data module is active, the commands that operate on scattered data are available. The user can change modules by selecting the icon for the module, by selecting an entity in the Project Explorer, or by right-clicking in the Project Explorer and selecting **Switch Module** from the pop-up menu. The module toolbar is displayed by default at the bottom left of the application.

Static Tools

This toolbar contains a set of tools that do not change for different modules. These tools are used for manipulating the display. By default they appear vertically at the top left of the display, between the Project Explorer and the Graphics Window.

Dynamic Tools



These tools change according to the selected module and the active model. They are used for creating and editing entities specific to the module. By default, the toolbar appear between the Project Explorer and the Graphics Window below the Static Tools.

Macros

There are three separate Macro Toolbars. These are shortcuts for menu commands. By default, the standard macros and the file toolbar appear above the Project Explorer when displayed, and the Optional Macros appear between the Project Explorer and the Graphics Window below the Dynamic Tools.

Macros **File Toolbar** **Optional Macros**



3.3 The Project Explorer

The Project Explorer allows the user to view all the data that makes up a part of a project. It appears by default on the left side of the screen, but can be docked on either side, or viewed as a separate window.

It is used to switch modules, select a coverage to work with, select a dataset to be active, and set display settings of the various entities in the active coverage. By right-clicking on various entities in the Project Explorer, the user may also transform, copy, or manipulate the entity.

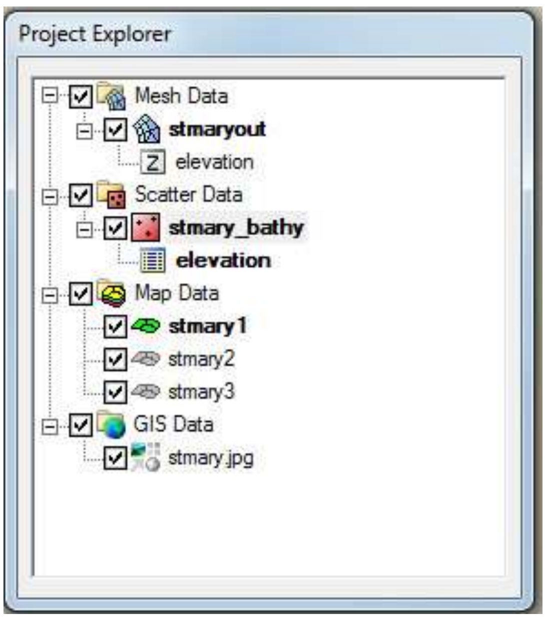


Figure 2 The Project Explorer window

3.4 Time Steps Window

The Time Steps window is used to select a time step to be active and is only visible if a transient dataset has been loaded into the project. By default, it appears below the Project Explorer.

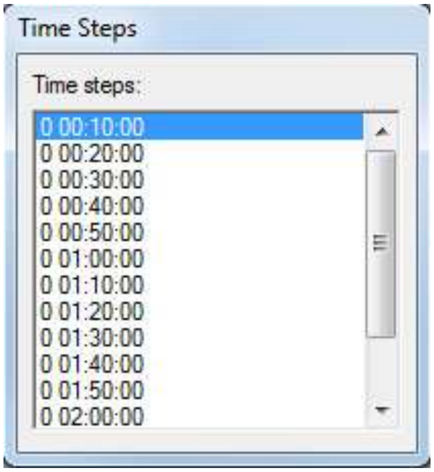


Figure 3 The Time Steps window

3.5 The Edit Window

The Edit Window appears below the menus at the top of the application. It is used to show and/or change the coordinates of selected entities. It also displays the functional data for those selected entities.

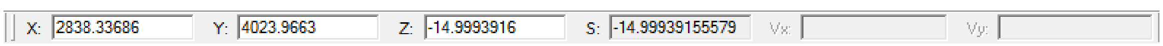


Figure 4 The Edit Window

3.6 The Menu Bar

The Menu Bar contains commands that are available for data manipulation. The menus shown in the Menu Bar depend on the active module and numerical model.

3.7 The Status Bars

There are two status bars: one at the bottom of the SMS application window and a second attached to the Main Graphics Window. The status bar attached to the bottom of the main application window shows help messages when the mouse hovers over a tool or an item in a dialog box. At times, it also may display a message in red text to prompt for specific actions, such as that shown in the figure below.

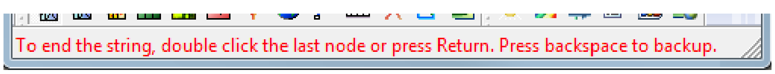


Figure 5 Status bar showing prompt

The second status bar, attached to the Main Graphics Window, is split into two separate panes. The left shows the mouse coordinates when the model is in plan view. The right pane shows information for selected entities.

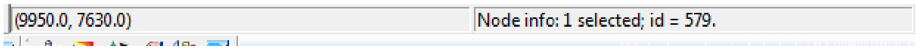


Figure 6 Status bar showing information for a selected entity

4 Using a Background Image

A good way to visualize the model is to import a digital image of the site. For this tutorial, an image was created by scanning a portion of a USGS quadrangle map and saving the scanned image as a JPEG file. SMS can open most common image formats including TIFF, JPEG, and Mr.Sid images. Once the image is inside SMS, it is displayed in plan view behind all other data, or it can be mapped as a texture onto a finite element mesh or triangulated scatter point surface.

4.1 Opening the Image

Do the following to open the JPEG image in this example:

1. Select *File* | **Open**. This will bring up the *Open* dialog.
2. Select the file “stmary.jpg” from the “data files” folder in the “SMS_Overview” folder.
3. Click **Open**. SMS opens the file and searches for image georeferencing data. Georeferencing data define the world locations (x, y) that correspond to each point in an image. It is usually contained inside a world file or sometimes the image itself. A world file could have the extension “.wld,” “.tfw,” “.jpgw,” and so on. If SMS finds georeferencing data, the image will be opened and displayed. If not, the user must define this mapping using the *Register Image* dialog. This is not required in this tutorial.
4. Depending on the preference settings, SMS may ask whether to build image pyramids or not. This improves image quality at various resolutions, but uses more memory. If asked, click **Yes** to generate the pyramids. Note that an entry is added to the Project Explorer as the image is read in under “GIS Data.”

5 Using Feature Objects

A conceptual model consists of a simplistic representation of the situation being modeled. This includes the geometric attributes of the situation (such as domain extents), the forces acting on the domain (such as inflow or water level boundary conditions), and the physical characteristics (such as roughness or friction). It does not include numerical details like elements. This model is constructed over a background image using feature objects in the Map module.

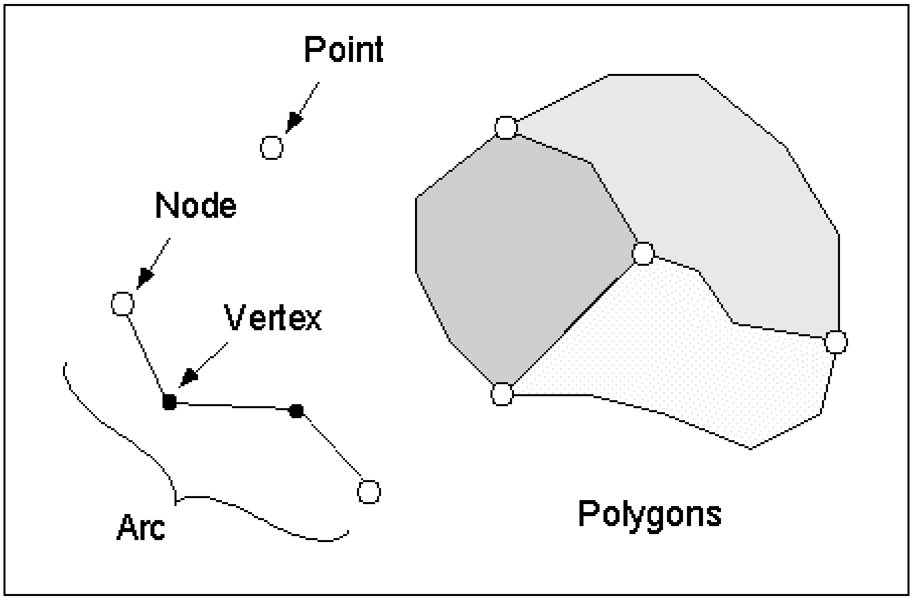


Figure 7 Feature Objects

Feature objects in SMS include points, nodes, arcs, and polygons, as shown in Figure 7. Feature objects are grouped into sets called "coverages." Only one coverage can be active at a time.

A "feature point" defines an (x, y) location that is not attached to an arc. Points are used to define the location of a measured field value or a specific location of interest such as a velocity gauge. SMS can extract data from a numerical model at such a location, or force the creation of a mesh node at the specific location.

A "feature node" is the same as a feature point, except that it is attached to at least one arc.

A "feature arc" is a sequence of line segments grouped together as a polyline entity. Arcs can form polygons or represent linear features such as channel edges. The two end points of an arc are called "feature nodes," and the intermediate points are called "feature vertices."

A "feature polygon" is defined by a closed loop of feature arcs. A feature polygon can consist of a single feature arc or multiple feature arcs, as long as a closed loop is formed. It may also include holes.

The conceptual model in this tutorial will consist of a single coverage in which the river regions and the flood bank will be defined. While going through this tutorial, the user will load new coverages over the existing coverage. The new coverage will become active and the old coverage will become inactive.

6 Creating Feature Arcs

A set of feature objects can be created to show topographically important features such as river channels and material region boundaries. Feature objects can be digitized directly inside SMS, converted from an existing CAD file (such as DXF or DWG), or they can be extracted from survey data. For this example, the feature objects will be digitized inside SMS using the registered JPEG image as a reference. To create the feature arcs by digitizing:

1. Click on the "Area Property" coverage to make it active.
2. Choose the **Create Feature Arc** tool from the Toolbox.
3. Click out the left riverbank, as shown in Figure 8 (some may want to **Zoom** closer). While creating the arc, if a mistake happens and there is a need to back up, press the *Backspace* key. If there is a need to abort the arc and start over, press the *Esc* key.
4. Double-click the last point to end the arc.

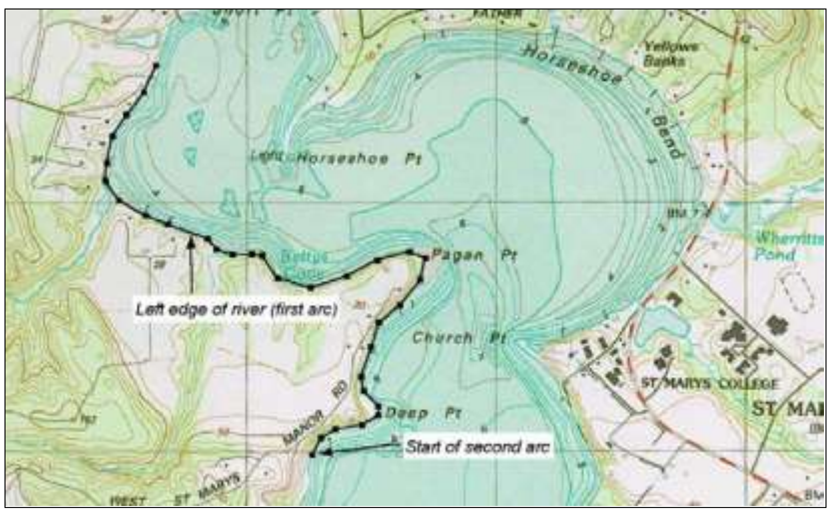


Figure 8 Creation of the first feature arc

A feature arc has defined the general shape of the left riverbank. Three more arcs are required to define the right riverbank and the upstream and downstream river cross sections. Together, these arcs will be used to create a polygon that defines the study area.

Do the following to create the remaining arcs:

1. In the same manner just described, create the remaining three arcs, as shown in Figure 9.
2. Remember to double-click to terminate an arc unless terminating at an existing node.

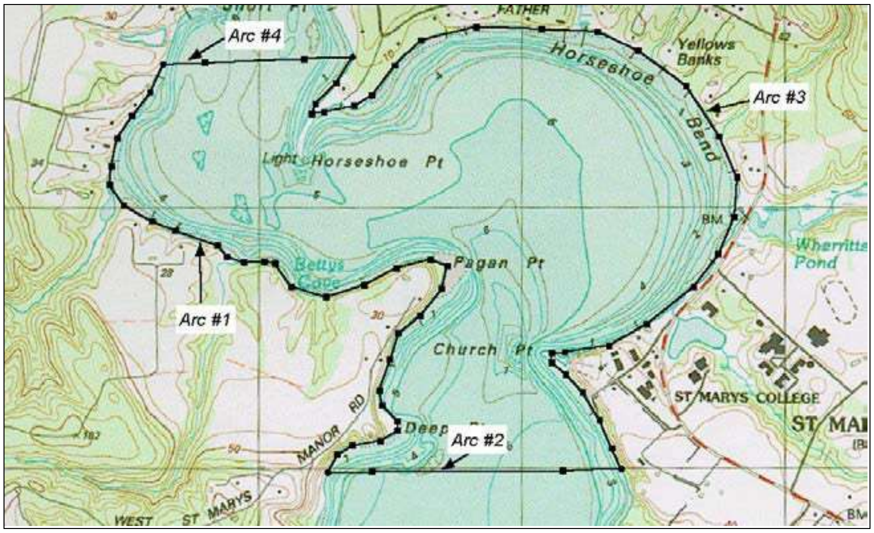


Figure 9 All feature arcs have been created

There is now a defined main river channel. When creating other models, the user would proceed to create other arcs, split the existing arcs to define material zones, and locate specific model features such as hard points on the river. To save time, a conceptual model with this all done has been saved in a file.

Do as follows to open the file:

1. Select *File* | **Open**.
2. Select the file "stmary1.map" from the *tutorial\SMS_Overview\data files* directory.
3. Click **Open**. A new coverage is created from the data in the file. Notice that the "Area Property" coverage is now called "stmary1" this is because no data had been entered into the "Area Property", SMS replaced it with the "stmary1" coverage.
4. To hide the coverage, uncheck the box next to its name ("stmary1") in the Project Explorer. Notice how the data in the Graphics Window disappears. Check the box next to "stmary1" to the data can be seen in the Graphics Window.

The display should look something like Figure 10.

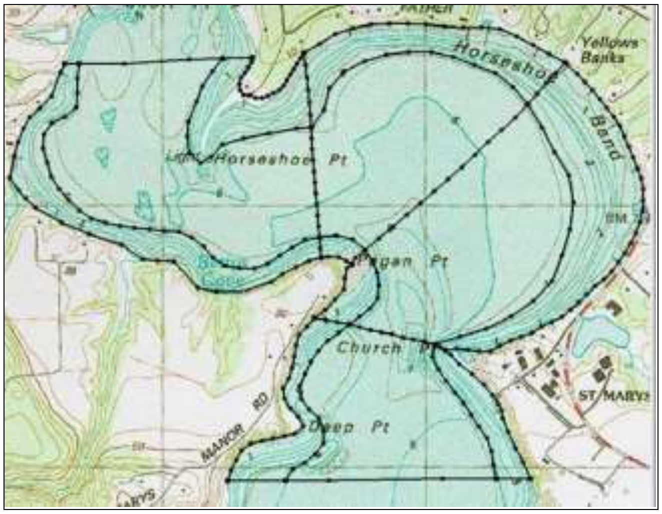


Figure 10 The stmary1.map feature object data

7 Manipulating Coverages

As stated at the beginning of this tutorial, feature objects are grouped into coverages. When a set of feature objects is opened from a file, one or more new coverages are created. The last coverage in the file becomes active. Any creation or editing of feature objects occurs in the active coverage. Inactive coverages are drawn in a blue-gray color by default or not displayed at all depending on the display attribute settings.

Each coverage is also represented by an entry on the Project Explorer. A project commonly includes many coverages defining various options in a design or various historical conditions.

When there are many coverages being drawn, the display can become cluttered. Individual coverages may be turned off by unchecking the box next to the coverage name in the Project Explorer. If a coverage is no longer desired, delete it by right-clicking on the coverage in the Project Explorer and selecting the **Delete** option.

8 Redistributing Vertices

To create the feature arcs, the user simply clicked out a line of points on the image. The user may or may not have paid much attention to the spacing of the vertices along the arc. The final element density in a mesh created from feature objects matches the density of vertices along the feature arcs, so it is desirable to have a more uniform node distribution. The vertices in a feature arc can be redistributed at a desired spacing.

To redistribute vertices, follow these steps:

1. Choose the **Select Feature Arc** tool from the Toolbox.

2. Click on the arc that follows the left of the river bank, labeled *Arc #1* in Figure 9 above. (Note: The addition of the conceptual model split Arc #1 into four segments; hold the *Shift* key down to select all four segments.)
3. Select *Feature Objects* | **Redistribute Vertices**. The *Redistribute Vertices* dialog shows information about the feature arc segments and vertex spacing.
4. Make sure the *Specified Spacing* option is selected and enter a value of "200" for *Average*. This tells SMS to create vertices 200 ft apart from each other (or 200 m apart if working in metric units).
5. Click **OK** to redistribute the vertices along the arc.

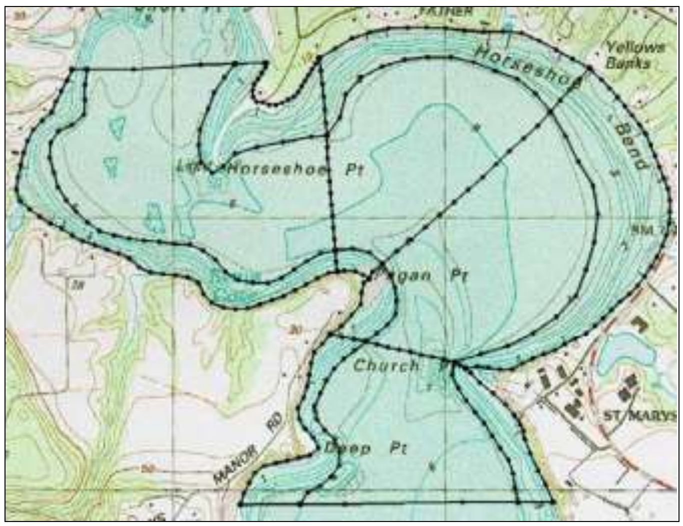


Figure 11 Redistribution of vertices along arcs

After clicking the **OK** button, the display will refresh, showing the specified vertex distribution. The arc will still be highlighted, because it is still selected.

6. Click somewhere else on the display. The selection is cleared and the effect of the command can be more clearly seen.

When creating conceptual models, this redistribution would be done for each arc until there is the vertex spacing that is wanted in all areas. If the spacing is the same for multiple arcs, multiple arcs can be selected and redistributed at the same time. When planning to use arcs in a patch, a better patch is created if opposite arcs have an equal number of vertices. In this case, it's best to use the *Number of Segments* option rather than the *Specified Spacing* option so that the exact number of vertices can be specified along each arc.

9 Defining Polygons

For this tutorial, open another map file, which has the vertices redistributed on all the arcs.

1. Select *File* | **Open**.
2. Select the file "stmary2.map" from the *tutorial\SMS_Overview\data files* directory.
3. Click **Open**.
4. Turn off the display of the "stmary1" coverage.

Before proceeding with defining polygons, the coverage type must be changed to a 2D Mesh:

5. Right-click on the coverage "stmary2" in the Project Explorer.
6. From the menu, choose *Type* | *Models* | **Generic Model**.

Polygons are created from a group of arcs that form a closed loop. Each polygon is used to define a specific material zone. Polygons can be created one by one, but it is more reliable to have SMS create them automatically. To have SMS build polygons out of the arcs, do the following:

7. Make sure no arcs are selected by clicking in the Graphics Window away from any arcs.
8. Select *Feature Objects* | **Clean** to bring up the *Clean Options* dialog.
9. Click **OK** in the *Clean Options* dialog. This will make sure there are no problems with the feature objects that were created.
10. Select *Feature Objects* | **Build Polygons**.

Although nothing appears to have changed in the display, polygons have been built from the arcs. The one evidence of this is that the **Select Polygon** tool becomes available (un-dimed). The polygons in this example are for defining the material zones as well as aiding in creating a better quality mesh.

10 Assigning Meshing Parameters

With polygons, arcs, and points created, meshing parameters can be assigned. These meshing parameters define which automatic mesh generation method will be used to create finite elements inside the polygon. For each method, a corner node of a finite element mesh will be created at each vertex on the feature arc. The difference comes in how internal nodes are created and how those nodes are connected to form elements. SMS has various mesh generation methods. The most commonly applied include patch, paving, and scalar paving density. These methods are described in the *SMS Online Help* (http://www.xmswiki.com/wiki/SMS:SMS), so they will not be described in detail here. As an overview, paving is the default technique because it works for all polygon shapes. Patches require either 3 or 4 polygonal sides. Density meshing options require scattered datasets to define the mesh density.

10.1 Creating a Refine Point for Paving

When using the default paving method, some control can be maintained over how elements are created. A "refine point" is a feature point that is created inside the boundary of a polygon and assigned a size value. When the finite element mesh is created, a corner node will be created at the location of the refine point and all element edges that touch the node will be the exact length specified by the refine point size value. Do the following to create a refine point:

1. Choose the **Select Feature Point** tool from the Toolbox.
2. Double-click on the point inside the left polygon, labeled in Figure 7. This will bring up the *Feature Point/Node Attributes* dialog.
3. In the dialog, make sure the *Refine Point* option is checked.
4. Enter a value of "75.0" (ft).
5. Accept the default for other options and click the **OK** button to accept the refine point. Depending on the display setting, the refine point will be distinguished with a different color than nodes, as shown in Figure 12.

When the finite element mesh is generated, a mesh corner node will be created at the refine point's location, and all attached element edges will be 75.0 feet in length. A refine point is useful when a node needs to be placed at a specific feature, such as at a high or low elevation point.

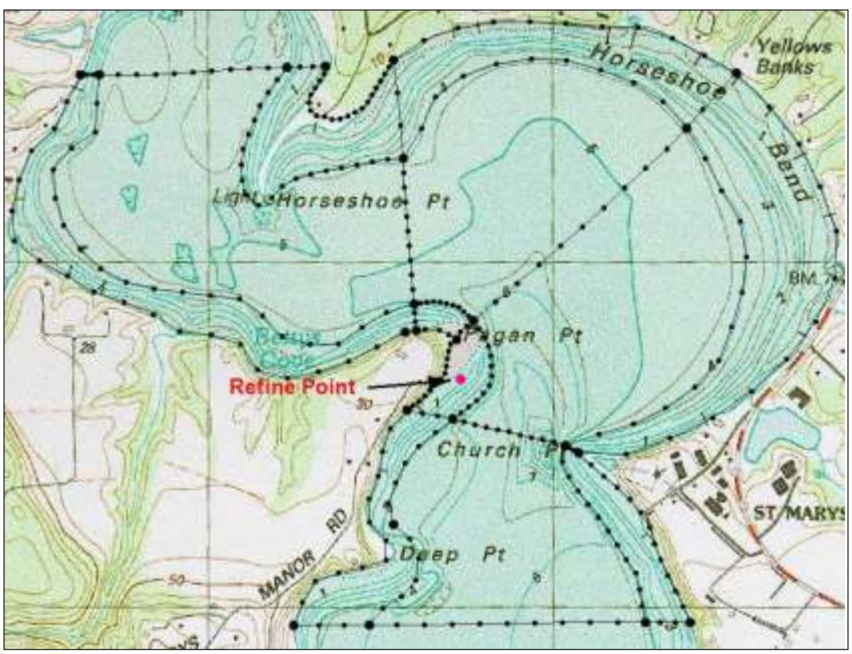


Figure 12 The location of the refine point

10.2 Defining a Coons Patch

The Coons Patch mesh generation method requires three or four sides to be created. However, it is not uncommon to use the patching technique to fill a polygon defined by more than four arcs. Figure 13 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.

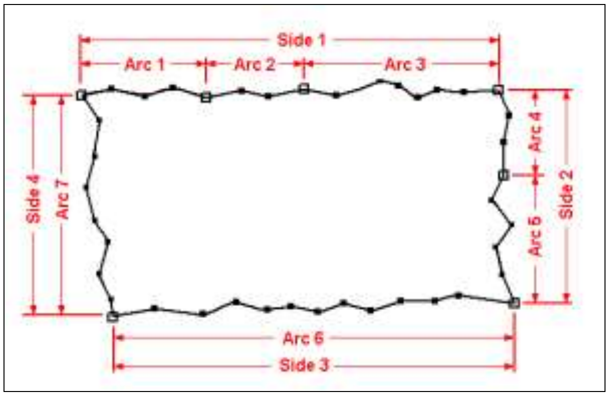


Figure 13 Four sides required for a rectangular patch

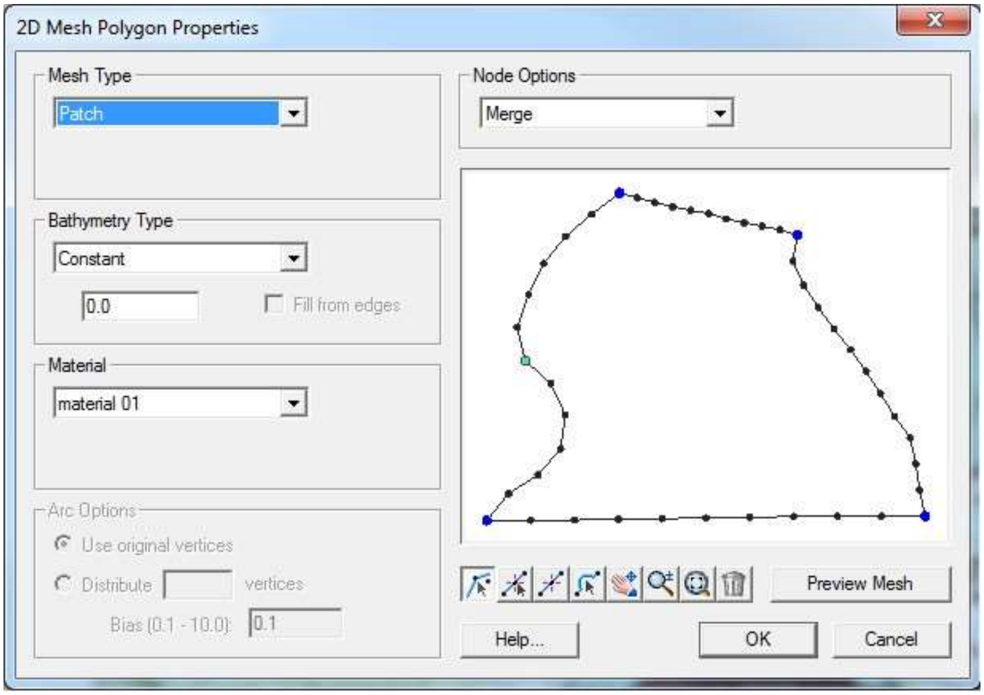


Figure 14 The Feature Polygon Attributes dialog

SMS provides a way to define a patch from such a polygon by allowing multiple arcs to act as one. For example, the bottom middle polygon in the current example contains five arcs, but it should be used to create a patch.

To define a patch, follow these steps:

1. Choose the **Select Feature Polygon** tool and double-click on the bottom middle polygon (the shape is seen in Figure 9 above). The *2D Mesh Polygon Properties* should appear.
2. In the dialog, choose the **Select Feature Point** tool.
3. Click on the node at the center of the left side, as seen in Figure 9.
4. Select the "Merge" option from the *Node Options* drop down list. This makes the two arcs on the left side be treated as a single arc.
5. Select the "Patch" option from the *Mesh Type* drop down list. (If the user tries to assign the meshing type to be "Patch" before merging the node, SMS pops up a message box indicating that 3 or 4 sides need to exist for a patch.) If the user wants to preview the patch, click the **Preview Mesh** button.
6. Click the **OK** button to close the *Polygon Attributes* dialog.

When creating models, it's necessary to set up the desired polygon attributes for each feature polygon in the model. For this tutorial, the rest of the polygons have been set up and saved to a map file.

Do the following to import this data:

1. Select the *File* | **Open** command.
2. Select the file "stmary3.map" from the *tutorial\SMS_Overview\data files* directory.
3. Click **Open**.

In the coverage that opens, all polygon attributes have been assigned. The four main channel polygons are assigned as patches, while the other polygons are assigned as paving.

11 Applying Boundary Conditions

> ***Note***: Boundary conditions can be added only when boundary condition generating models are enabled. If the TABS or FESWMS models are not enabled in SMS, then boundary conditions cannot be added. If these models are not enabled, proceed to section 13 of this tutorial.

The coverage type controls which model will be used when a numeric model is generated from a conceptual model. This also controls the types of boundary conditions that can be assigned to the conceptual model. To view the type of the coverage, do the following:

1. Right-click on the coverage "stmary3" in the Project Explorer.

2. Select *Type* | *Models*, and make sure the type is either **TABS** (RMA2) or **FESWMS**.

Boundary conditions can be assigned to arcs, points, and, for FESWMS, polygons. Feature arcs may be assigned a flow, head, or flux status. Feature points may be assigned velocity or head values. In an FESWMS coverage, feature polygons may be assigned ceiling elevation functions.

The inflow for this example is across the top of the model and the outflow is across the bottom. Notice that there are three feature arcs across each of these sections. A flow rate value could be assigned to each of the arcs at the inflow. However, this would create three separate inflow nodestrings, connected end to end. The same situation exists at the outflow cross section.

Both RMA2 and FESWMS can have numerical problems if two boundary conditions are adjacent to each other with no corner between them. To avoid creating three separate boundary conditions at a single cross section, an arc group should be defined. An "arc group" consists of multiple arcs that are linked together. The arc group can be assigned the boundary condition instead of assigning it at the individual arcs so that, when the model is generated, it creates only a single nodestring that spans the entire cross section.

11.1 Defining Arc Groups

For this example, two arc groups will be defined. One will be positioned at the inflow boundary and one at the outflow boundary. To create the arc groups:

1. Choose the **Select Feature Arc**  tool from the Toolbox.
2. Holding the *Shift* key, select the top three arcs that make up the flow cross section, labeled as "Flow Arcs" in Figure 15 below. (Alternatively, select all three arcs by dragging a box around them. This box must include the entire arc.)
3. Select *Feature Objects* | **Create Arc Group** to create an arc group from the three selected arcs.
4. Now, use the *Shift* key to select the three bottom arcs that make up the head cross-section, labeled as "Head Arcs" in Figure 15. (Make sure only these three arcs are selected by checking the Status Bar at the bottom right of the Main Graphics Window.)
5. Select *Feature Objects* | **Create Arc Group**.

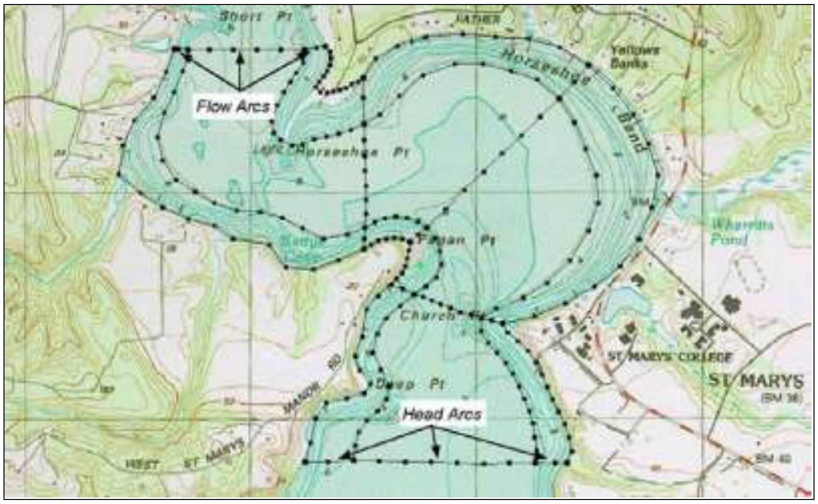


Figure 15 The arc groups to create

11.2 Assigning the Boundary Conditions

With the arc groups created, boundary conditions can now be assigned. To assign the inflow boundary condition, do as follows:

1. Choose the **Select Feature Arc Group** tool from the Toolbox.
2. Double-click the arc group at the inflow (top) cross section.
3. In the *Feature Arc Attributes* dialog, select the *Boundary Conditions* option.
4. Click the **Options** button to open a new dialog.
    a. If using FESWMS, the *FESWMS Nodestring Boundary Conditions* dialog will appear. Make sure the *Boundary type* is set to "Specified Flow / WSE", then toggle on *Flow*.
    b. Enter a of flow rate of “40,000” cfs.
    c. If using RMA2, the *RMA2 Assign Boundary Conditions* dialog will appear. Select *Specified flow rate* as the *Boundary Condition Type*.
    d. Make certain *Constant* is toggled on under *Flow rate* and enter “40,000” cfs.
    e. In the *Flow Direction* section, toggle on *Perpendicular to boundary* to force the flow to enter the mesh perpendicular to the inflow boundary.
5. Click the **OK** button to close each dialogs.

To assign the water surface boundary condition:

1. Double-click the bottom arc group at the outflow cross section. The *Feature Arc Attributes* dialog will appear.

2. Select the *Boundary Conditions* option.
3. Click the **Options** button.
   a. If using FESWMS, the *FESWMS Nodestring Boundary Conditions* dialog will appear. Make sure the *Boundary type* is set to "Specified Flow / WSE", then toggle on *Water surface elevation*.
   b. Enter in a *WSE* (water surface elevation) of “20” ft.
   c. If using RMA2, the *RMA2 Assign Boundary Conditions* dialog will appear. Select *Water surface elevation* as the *Boundary Condition Type*.
   d. Select *Constant* and enter “20” ft.
4. Click the **OK** button to close both dialogs.

The inflow and outflow boundary conditions are now defined in the conceptual model. When the conceptual model is converted to a finite element mesh, SMS will create the nodestrings and assign the proper boundary conditions.

12 Assigning Materials to Polygons

Each polygon is assigned a material type. All elements generated inside the polygon are assigned the material type defined in the polygon. In order to assign the materials, new materials must be specified:

1. Click on *Edit* | **Materials Data.** This will bring up the *Materials Data* dialog.
2. Select the **New** button. This creates a new material named “material 02”.
3. Select **New** again.
4. Double-click on “material 01” and rename it “Left Bank.”
5. Double-click on “material 02” and rename it “Main Channel.”
6. Double-click on “material 03” and rename it “Right Bank.” The user may change the colors/patterns if desired for all three materials by using the *Pattern* drop-down menu to the right of each material.
7. Click **OK**.

Once the materials have been specified, assign them using the following steps:

1. Choose the **Select Feature Polygon**  tool from the Toolbox.
2. Hold down the *Shift* key and select all the polygons labeled “Left Bank” in Figure 16 below.
3. Right-click on any of the selected polygons and select **Attributes…**.

4. In the *2D Mesh Polygon Properties* dialog, change the *Material* to “Left Bank” from the drop-down menu.
5. Click the **OK** button.
6. Deselect the polygons by clicking anywhere on the image outside the polygons.
7. Repeat the previous steps 119-5 for the Main Channel and Right Bank, being sure to select the appropriate material in the *2D Mesh Multiple Polygon Properties* dialog ("Main Channel" for the Main Channel polygons, and "Right Bank" for the Right Bank polygons).

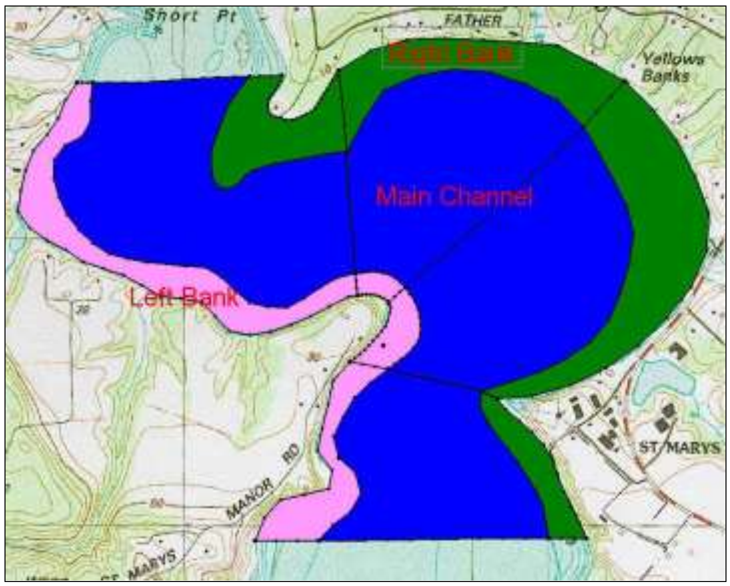


Figure 16 Polygons with defined material types

12.1 Displaying Material Types

With the materials assigned to the polygons, now fill the polygons with the material colors and patterns by following these steps:

1. Click the **Display Options** macro from the Toolbox to bring up a dialog.
2. In the *Display Options* dialog, if not active, select *Map* from the list on the left side of the dialog.
3. Turn on the *Fill* option under *Polygon.*
4. Make sure the *Fill with materials* option is selected.
5. Click the **OK** button to close the *Display Options* dialog.

The display will refresh, filling each polygon with the material color and pattern.

13 Converting Feature Objects to a Mesh

With the meshing techniques chosen, boundary conditions assigned, and materials assigned, the next process it to generate the finite element mesh.

To do this, follow these steps:

1. Make sure no objects are selected by clicking in the Graphics Window away from the river channel.
2. Select *Feature Objects* | **Map →2D Mesh** to open the *2D Mesh Options* dialog.
3. In the dialog that appears, click the **OK** button to start the meshing process.
4. In the *Mesh Name* dialog, accept the default name by clicking **OK**.

After a few moments, the display will refresh to show the finite element mesh that was generated according to the preset conditions. With the mesh created, it is often desirable to delete or hide the feature arcs and the image. To hide the feature arcs and image, do as follows:

1. Click the **Display Options** macro from the Toolbox.
2. In the *Display Options* dialog, if it is not active, select the *Map* tab.
3. Turn off the following options:
    - *Arc*
    - *Node*
    - *Fill* in the *Polygon* section
4. Click the **OK** button to close the *Display Options* dialog.
5. To hide the image, uncheck the toggle box next to the “stmary” image icon under the “GIS Data” folder in the Project Explorer.
6. Frame the image by selecting *Display* | **Frame Image** or clicking on the **Frame Image** macro in the Toolbar.

The display will refresh to show the finite element mesh, as shown in Figure 17. With the feature objects and image hidden, the mesh can be manipulated without interference, but they are still available if mesh reconstruction is desired.

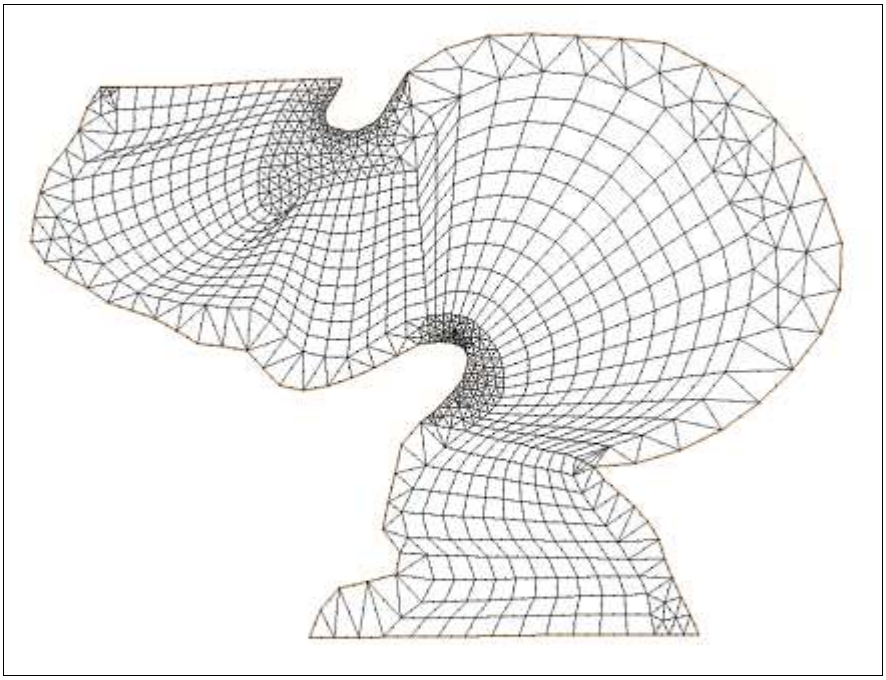


Figure 17 The generated finite element mesh

14 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.

15 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, the user would 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** command.

2. Locate the file "stmary_bathy.h5" from the *tutorial\SMS_Overview\data files* directory.

3. Select **Open**.

The screen will refresh, showing a set of scattered data points. 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.

To interpolate the scattered data onto the mesh:

1. Make sure the "stmary_bathy" scatter module is active.

2. Select *Scatter* | **Interpolate to Mesh**.

3. In the *Interpolation* dialog, make sure "Linear" is selected from the *Interpolation* drop down list. (For more information on SMS interpolation options, see *SMS Online Help*.)

4. Turn on the *Map Z* option at the lower left area of the dialog under *Other Options*.

5. Click the **OK** button to perform the interpolation.

6. After this is completed, a new dataset, "elevation_interp", will appear under the "stmary3 Mesh" item in the "Mesh Data" folder in the Project Explorer.

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. To hide the scatter point data uncheck the box next to the scatter set named "stmary_bathy" in the Project Explorer.

To delete the scatter set:

1. Right-click on this object and select **Delete**.

2. Select OK in the dialog that appears

16 Renumbering the Mesh

The following section can only be completed if the section about applying boundary conditions has been completed. If not, please proceed to the next section.

The process of creating and editing a finite element mesh can cause the node and element ordering to become disorganized. Renumbering the mesh can restore a good mesh ordering. (The mesh is renumbered after the mesh generation, but the mesh is renumbered from an arbitrary nodestring, which does not always give the best renumbering.)

To renumber, follow these steps:

1. Select the “stmary3 Mesh” dataset in the Project Explorer.
2. Choose the **Select Nodestring** tool from the Toolbox.
3. Select the flow nodestring at the top of the mesh by clicking inside the box icon that is at the middle of the nodestring.
4. Select *Nodestrings* | **Renumber Nodestrings**.

Although nothing appears to have changed in the display, the mesh has been renumbered starting from the top flow nodestring.

17 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 New Project**. This will bring up a new dialog.
2. Name the file “stmaryout.sms.”
3. Click the **Save** button to save the files.

18 Conclusion

This concludes the “SMS Overview” tutorial. Topics covered in this tutorial included:

- An overview of the SMS layout and interface.
- Using a background image
- Using feature objects
- Manipulating coverages
- Assigning mesh parameters
- Applying boundary conditions
- Assigning materials
- Basic mesh generation
- Saving a project file

The user may continue to experiment with the SMS interface or may quit the program.