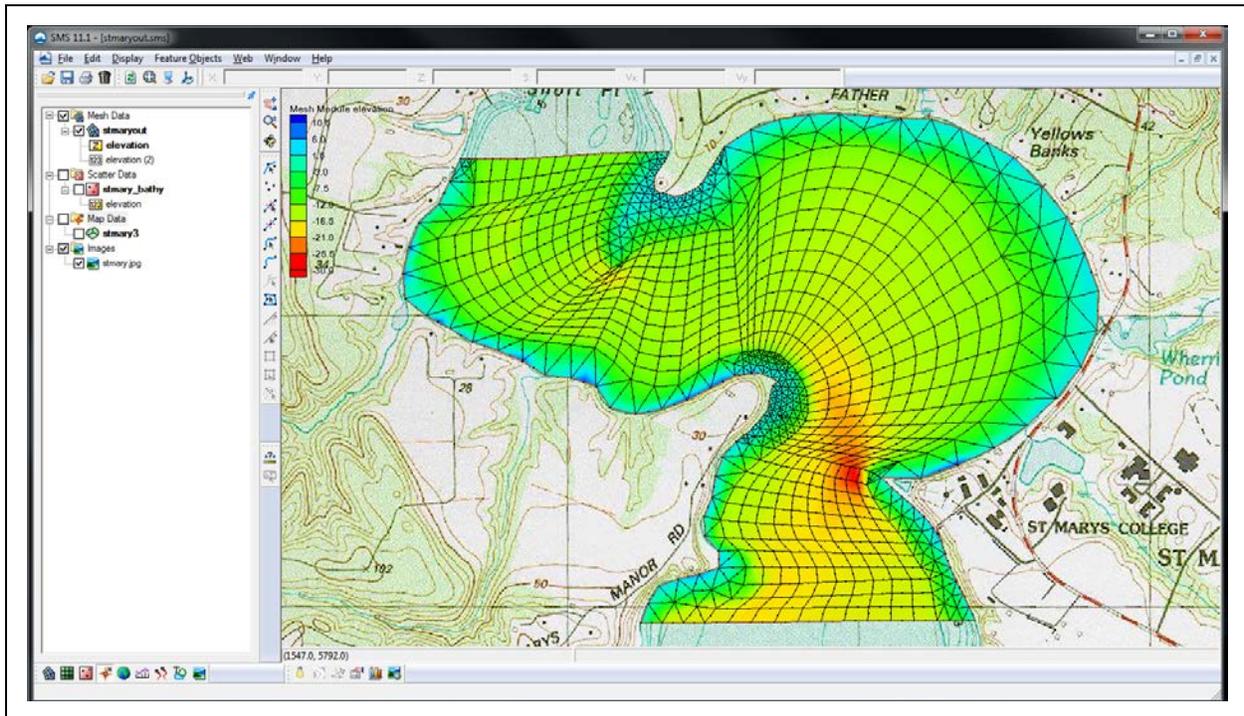


SMS 11.1 Tutorial Overview



Objectives

This tutorial describes the major components of the *SMS* interface and gives a brief introduction to the different *SMS* modules. It is suggested that this tutorial be completed before any other tutorial. All files for this tutorial are found in the Data Files Folder.

Prerequisites

Requirements

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

Time

- 45-60 minutes

AQUAVEO™



1 Requirements

In order to complete this tutorial, you must have the Generic 2d Mesh and either TABS or FESWMS models enabled under your 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 their status are displayed. Toggle off *Show only enabled modules* to show both enabled and disabled components. If desired, you may change your registration according to your needs by clicking on *Change Registration* and completing the steps in the *Registration Wizard* dialog.

2 Getting Started

Before beginning this tutorial you should have installed *SMS* on your computer. If you have not yet installed *SMS*, please do so before continuing. Each chapter of this tutorial document demonstrates the use of a specific component of *SMS*. If you have not purchased all modules of *SMS*, or if you are evaluating the software, you should run *SMS* in *Demo Mode* to complete this tutorial (see section 3 of the SMS Intro Tutorial). When using *Demo Mode*, you will not be able to save files. For this reason, all files that you are asked to save have been included in the *output* subdirectory under the *tutorial\SMS_Overview* directory. When you are asked to save a file, you should instead open the file from this *output* directory. To start *SMS*, do the following:

- Open the *Start* menu, go to *All Programs*, select *SMS 11.1* and click on *SMS 11.1*.

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 to display 2D plots of various data.

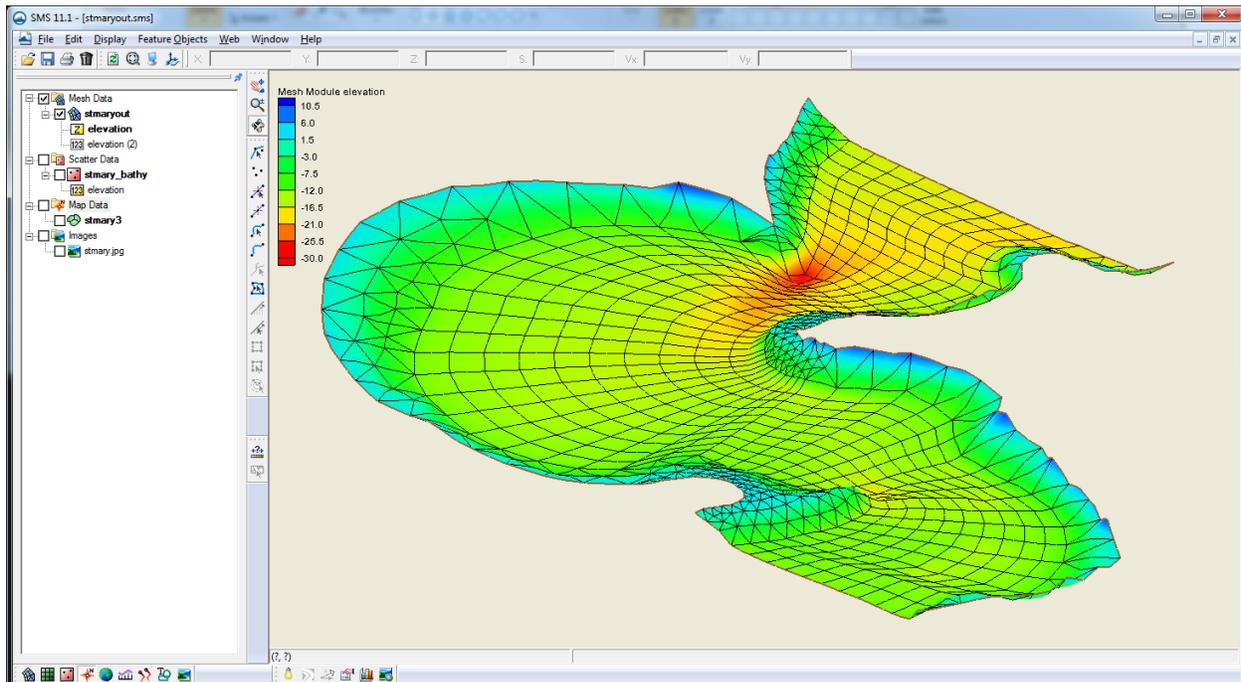


Figure 1 The SMS screen.

3.1 The Main Graphics Window

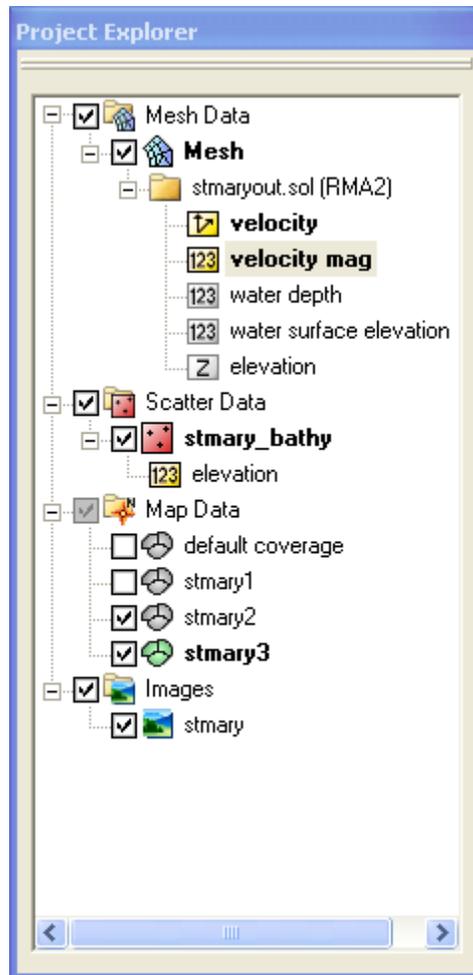
The *Main Graphics Window* is the biggest part of the *SMS* screen. Most of the data manipulation is done in this window. You will use it with each tutorial chapter.

3.2 The Toolbox

The *Toolbar* actually consists of multiple dockable toolbars. By default they are positioned at various locations on the left side application, but can be positioned around the interface as desired. The toolbars include:

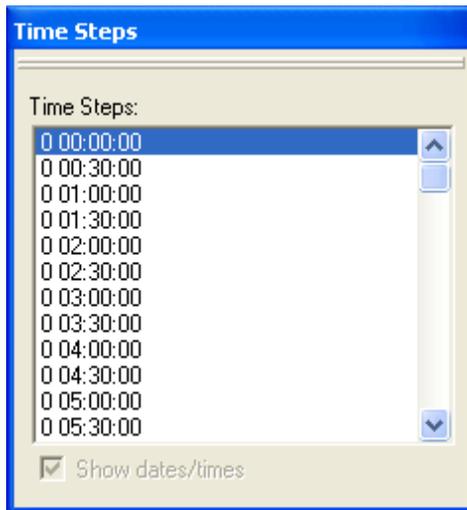
Modules. This image shows the current *SMS Modules*. As described in the *SMS online Help* these icons control what menu commands and tools are available at any given time while operating in *SMS*. Each module corresponds to a specific type of data. For example, one icon corresponds to finite element 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, or selecting an entity in the *Project Explorer* or by

various entities in the project explorer, the user may also transform, copy, or manipulate the entity.



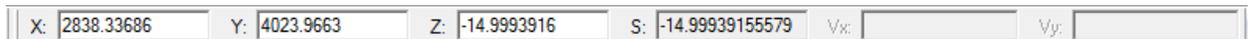
3.4 Time Steps Window

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*.



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.



3.6 The Menu Bar

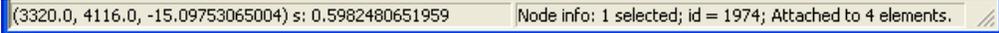
The *Menu Bar* contains commands that are available for data manipulation. The menus shown in the *Menu Bar* depend on the active module and numerical model.

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.

Enter the polygon by clicking in the window. Double-click to end.

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.



((3320.0, 4116.0, -15.09753065004) s: 0.5982480651959 Node info: 1 selected; id = 1974; Attached to 4 elements.

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

To open the *JPEG* image in this example:

1. Select *File /Open*.
2. Select the file *stmary.jpg* from the Data File Folder, in the *File Open* dialog that appears.
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” ... 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 your preference settings, *SMS* may ask whether you want to build image pyramids. 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 “Images”.

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.

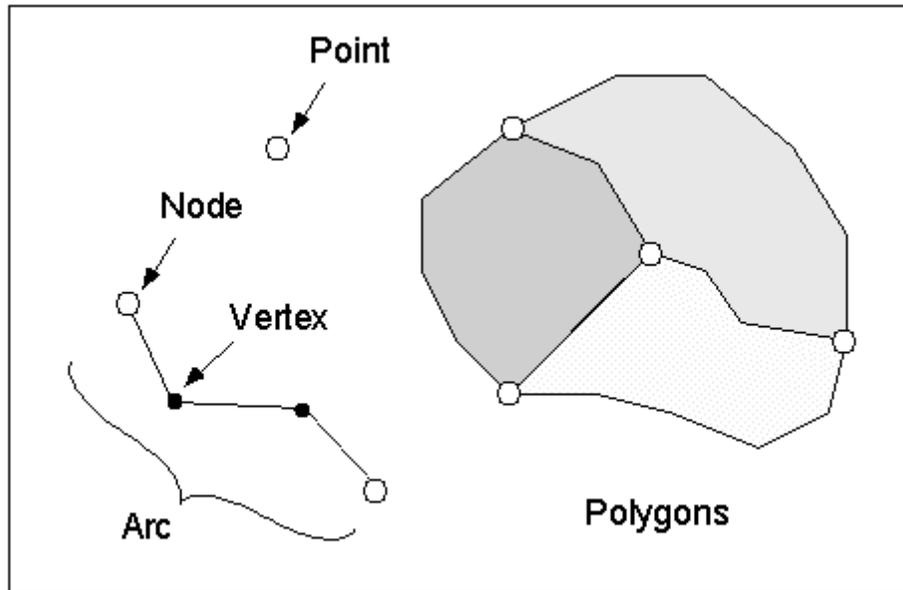


Figure 2 Feature Objects

Feature objects in *SMS* include points, nodes, arcs and polygons, as shown in Figure 2. Feature objects are grouped into sets called *coverages*. Only one coverage is 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. As you go along in this tutorial you 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. Choose the *Create Feature Arc*  tool from the *Toolbox*.
2. Click out the left riverbank, as shown in Figure 3 (you may want to Zoom  closer). As you create the arc, if you make a mistake and wish to back up, press the *BACKSPACE* key. If you wish to abort the arc and start over, press the *ESC* key.
3. Double-click the last point to end the arc.

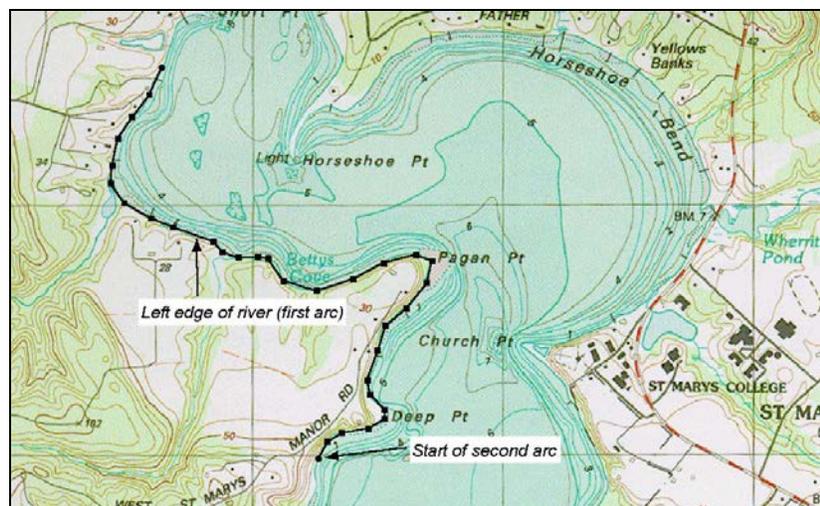


Figure 3 Creation of the first feature arc.

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. To create the remaining arcs:

1. In the same manner just described, create the remaining three arcs, as shown in Figure 4.
2. Remember to double-click to terminate an arc unless you are terminating at an existing node.

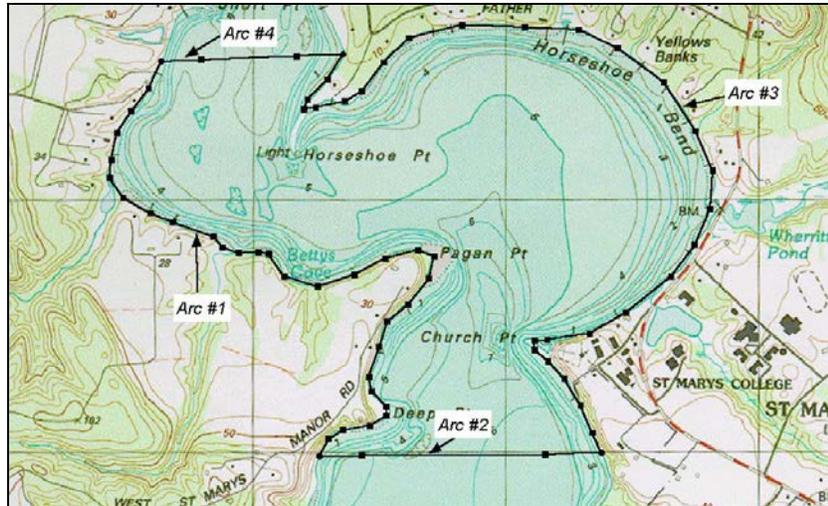


Figure 4 All feature arcs have been created.

You have now defined the main river channel. When creating your own models, you will proceed to create other arcs, and 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. To open the file:

1. Select *File | Open*.
2. Open the file *stmary1.map* from the Data Files Folder.

A new coverage is created from the data in the file, and the coverage you were editing becomes inactive. To hide the inactive coverage:

3. Uncheck the box next to its name (default coverage) in the *Project Explorer*.

The new coverage is added to the *Project Explorer* with the name "stmary1". The display should look something like Figure 5.

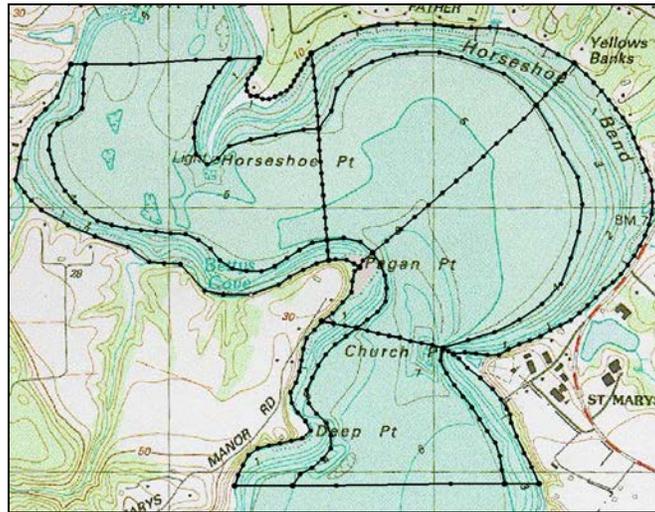


Figure 5 The stmary1.map feature object data.

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, you may also 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, you simply clicked out a line of points on the image. You 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:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Click on the arc to the far left, labeled *Arc #1* in Figure 4.
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. This tells SMS to create vertices 200 ft apart from each other. If you are working in metric units, this would tell SMS to create vertices spaced 200 meters apart.
5. Click *OK* to redistribute the vertices along the arc.

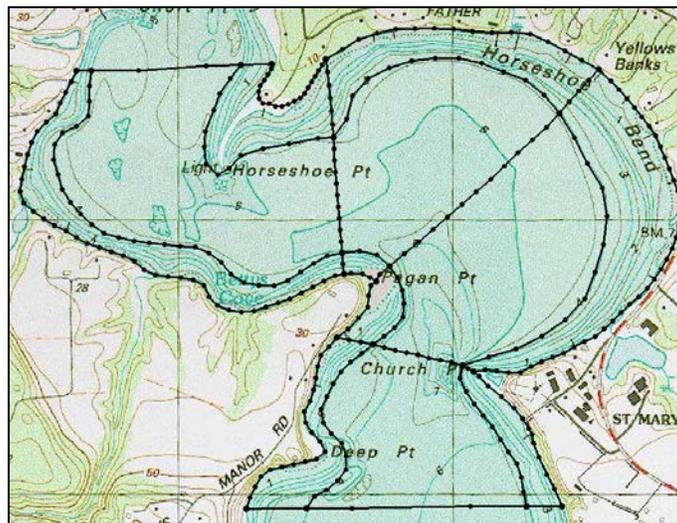


Figure 6 Redistribution of Vertices along arcs.

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

9 Defining Polygons

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

1. Open the map file `stmary2.map` as you did with the previous file.
2. Turn off the display of the “stmary1” coverage.

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

3. Right-click on the coverage “stmary2” in the Project Explorer.
4. From the menu, choose *Type | Models | Generic 2D Mesh*.

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:

5. Make sure no arcs are selected by clicking in the Graphics Window away from any arcs.
6. Select *Feature Objects | Clean* to be sure that there are no problems with the feature objects that were created.
7. Click *OK* in the Clean Options dialog.
8. 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 to aid 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*, 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 data sets 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. 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.
3. In the *Feature Point/Node Options* dialog, make sure the *Refine Point* option is checked on.
4. Enter a value of 75.0 (ft).
5. 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.

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.

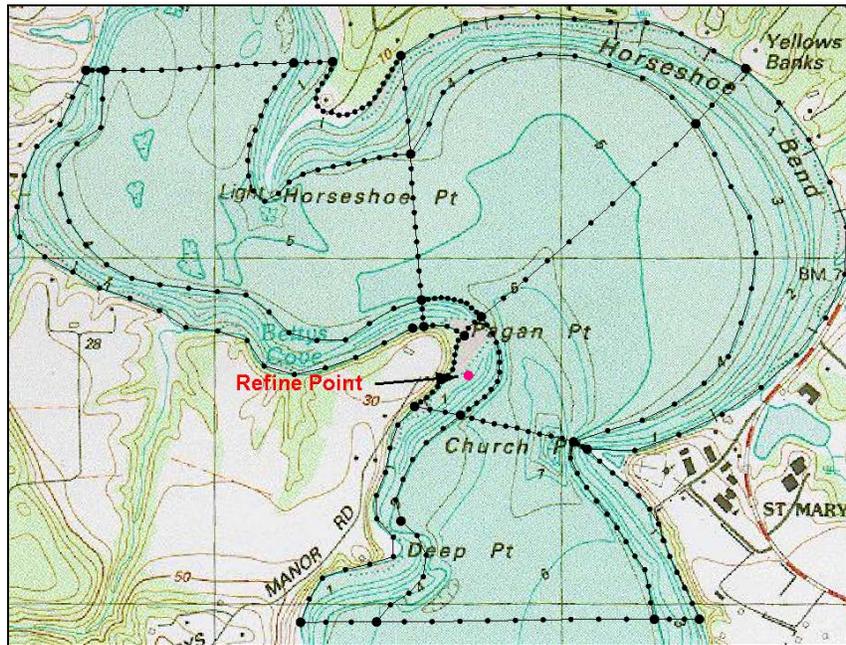


Figure 7 The location of the refine point.

10.2 Defining a Coons Patch

As was previously stated, the *Coons Patch* mesh generation method requires three or four sides to be created. However, it is not uncommon, that we wish to use the patching technique to fill a polygon defined by more than four arcs. Figure 8 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.

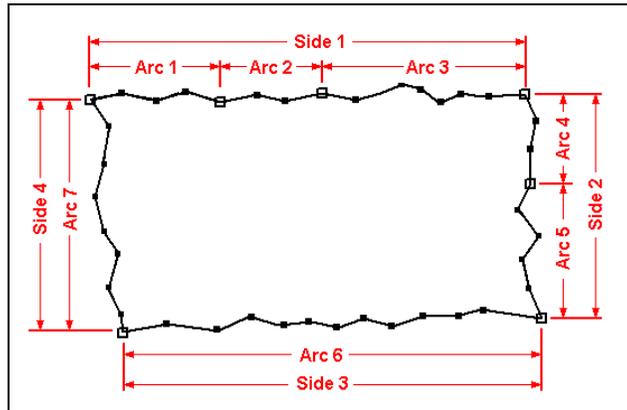


Figure 8 Four sides required for a rectangular patch.

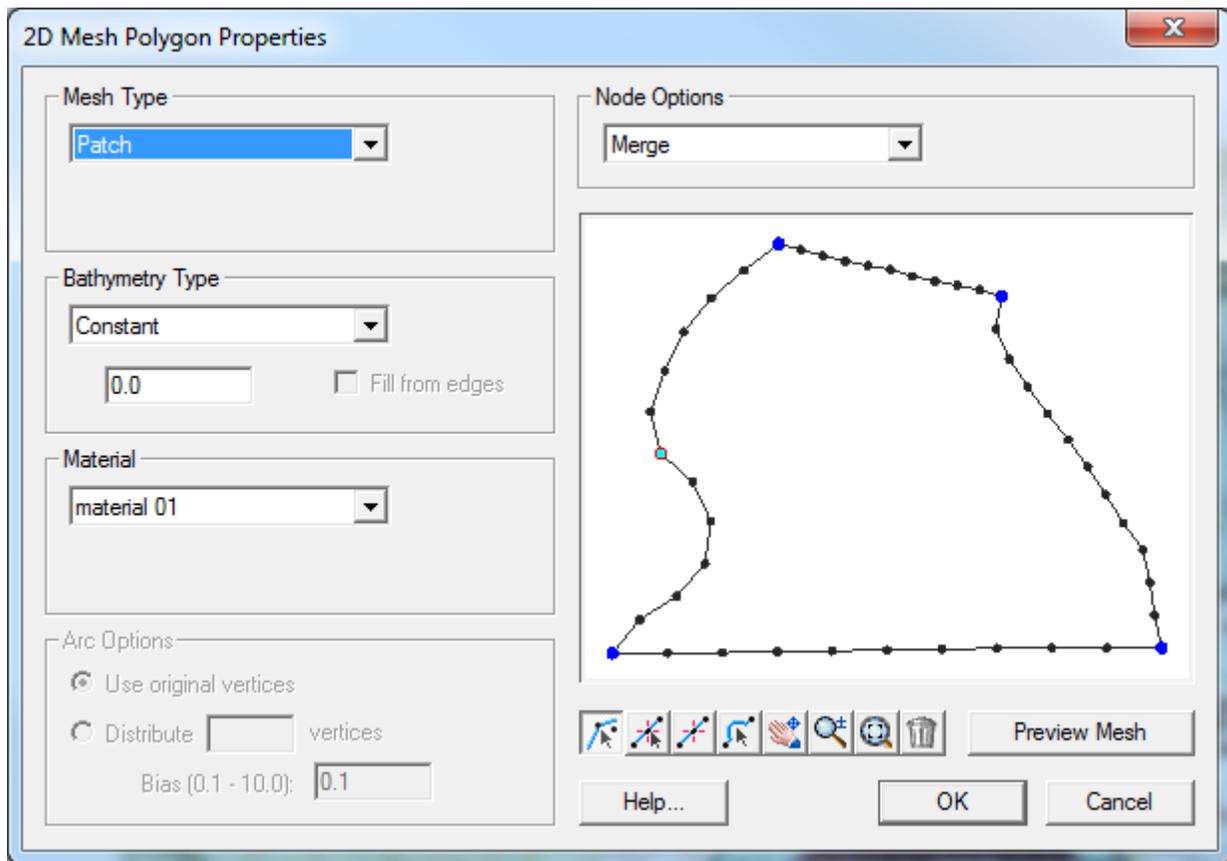


Figure 9 The Feature Polygon Attributes dialog.

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 our example contains five arcs, but it should be used to create a patch. To do this:

1. Choose the *Select Feature Polygon*  tool and double click on the bottom middle polygon.
2. In the *2D Mesh Polygon Properties* 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 you tried to assign the meshing type to be *Patch* before merging the node, SMS would have popped up a message box informing you that you need 3 or 4 sides for a patch.) If you wish to preview the patch, click the *Preview Mesh* button.
6. Click the *OK* button to close the *Polygon Attributes* dialog.

When you are creating your models, you will need to set up the desired polygon attributes for each feature polygon in your model. For this tutorial, the rest of the polygons have been set up for you and saved to a map file. To import this data:

7. Open the file *stmary3.map*.

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 you do not have the following models enabled in SMS then boundary conditions cannot be added. 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:

1. Right click on the coverage *stmary3* in the *Project Explorer* and select *type / Models*, and make sure the type is either “Tabs” 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. Feature polygons may be assigned ceiling elevation functions, but only in a *FESWMS* coverage.

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, only a single nodestring is created, which 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 three arcs that make up the flow cross-section, labeled as *Flow Arcs* in Figure 10. (Alternatively, you can 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, select the three arcs that make up the head cross-section, labeled as *Head Arcs* in Figure 10. (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*.

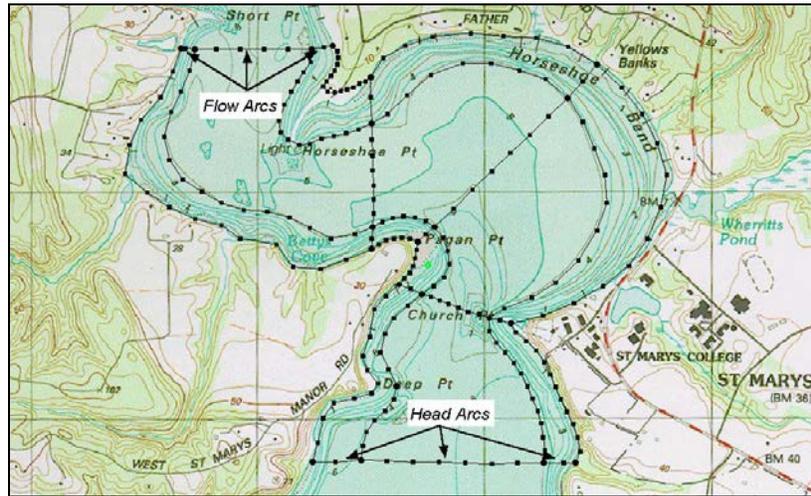


Figure 10 The arc groups to create.

11.2 Assigning the Boundary Conditions

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

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, and click the *Options* button.
4. If using *FESWMS*, the *FESWMS Nodestring Boundary Conditions* dialog will appear. Select the *Flow* option under *Specified Flow/WSE Options*. Enter in a *Constant* flowrate of 40,000 cfs.
5. If using *RMA2*, the *RMA2 Assign Boundary Conditions* dialog will appear. Select *Specified flowrate* as the *Boundary Condition Type*. Enter in a *Constant* flowrate of 40,000 cfs. Toggle on *Perpendicular to boundary* to force the flow to enter the mesh perpendicular to the inflow boundary.
6. Click the *OK* button to close both dialogs.

To assign the water surface boundary condition:

1. Double-click the arc group at the outflow cross section.
2. In the *Arc Group Attributes* dialog, select the *Boundary Conditions* option, and click the *Options* button.

3. If using *FESWMS*, select the *Water surface elevation* option under *Specified Flow/WSE Options*. Enter in a *Constant* water surface elevation of 20 ft
4. If using *RMA2*, select *Water surface elevation* as the *Boundary Condition Type*. Enter in a *Constant* water surface elevation of 20 ft.
5. 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*.
2. Select *New*. This creates a new material named *material 02*
3. Select *New* again.
4. Double click on *material 02* and rename it “Main Channel”
5. Rename the other 2 materials “Left bank” and “Right bank”. You may change the colors/patterns as you wish.
6. Click *OK*.
7. Choose the *Select Feature Polygon*  tool from the *Toolbox*.
8. Double-click on any of the polygons.
9. In the *2D Mesh Polygon Properties* dialog, make sure the *Material* section shows the correct material for the polygon, as shown in Figure 11.
10. Click the *OK* button to close the *2D Mesh Polygon Properties* dialog.
11. Repeat these steps to make sure the correct material type is assigned to each of the feature polygons. The following figure shows the materials that should be assigned to each polygon.

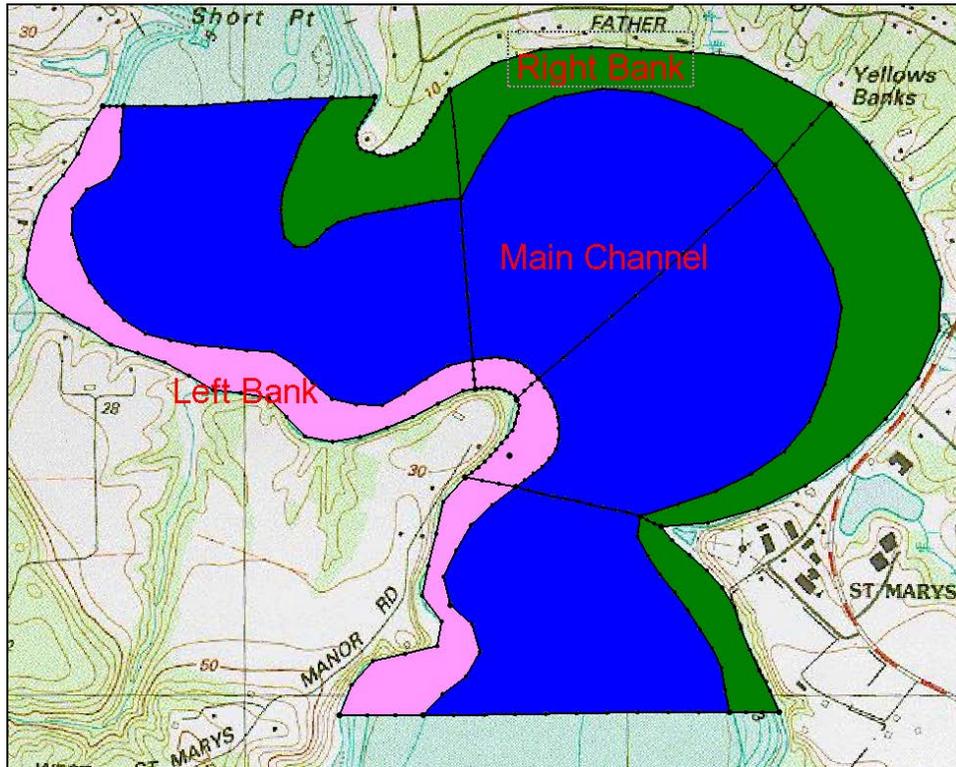


Figure 11 Polygons with defined material types.

12.1 Displaying Material Types

With the materials assigned to the polygons, you can fill the polygons with the material colors and patterns. To do this:

1. Click the *Display Options*  macro from the *Toolbox*.
2. If not active, select the *Map* tab on the left side of the dialog.
3. Turn on the *Polygon fill* option and make sure the *Fill with materials* option is selected.
4. 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, we are ready to generate the finite element mesh. To do this:

1. We want to convert the entire conceptual model to a mesh. Therefore, nothing should be selected. If individual polygons were selected, only those polygons would be converted to mesh segments. Make sure no objects are selected by clicking in the *Graphics Window* away from the river channel.
2. Select *Feature Objects | Map ->2D Mesh*.
3. Click the *OK* button to start the meshing process.

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:

1. Click the *Display Options*  macro from the *Toolbox*.
2. If it is not active, select the *Map* tab.
3. Turn off the display of *Arcs*, *Nodes*, and *Polygon fill*.
4. Click the *OK* button to close the *Display Options* dialog.
5. To hide the image click on the toggle box next to the “stmary” image icon 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 12. With the feature objects and image hidden, the mesh can be manipulated without interference, but they are still available if mesh reconstruction is desired.

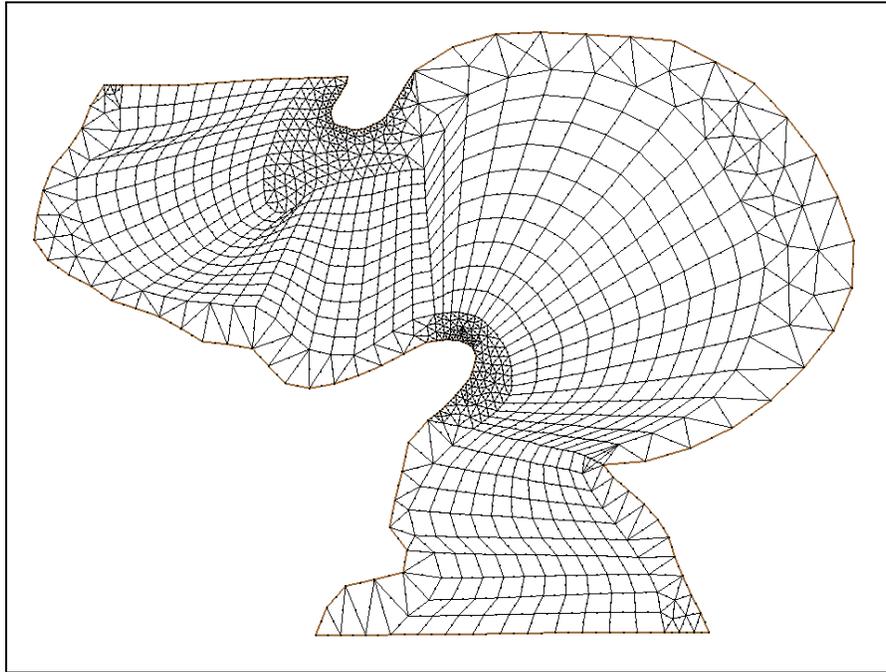


Figure 12 The finite element mesh that was created.

14 Editing the Generated Mesh

When a finite element mesh is generated from feature objects, it is not always the way you want it. 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 the *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 we had not read in the bathymetric data before generating the mesh. Normally, you would read in the survey data, and associate it with the polygons to assign bathymetry to your 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:

1. Select *File / Open* and open the file *stmary_bathy.h5*.

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 you to manually interpolate 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 you are in the *Scatter*  module.
2. Select *Scatter / Interpolate to Mesh*.
3. In the *Interpolation* dialog, make sure *Linear* from the *Interpolation* drop down list is selected. (For more information on *SMS* interpolation options, see the *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.

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, right click on this object and select *Delete*.

16 Renumbering the Mesh

The following can only be done if the section about applying boundary conditions has been done. 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:

1. Switch to the *Mesh*  module.
2. Choose the *Select Nodestring* tool  from the *Toolbox*.

3. Select the flow nodestring at the top of the mesh by clicking inside the icon that is at the middle of the nodestring.
4. Select *Nodestrings / Renumber*.

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, several files are saved. Separate files are created for the map, scatter 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*.
2. Save the file as *stmaryout.sms*.
3. Click the *Save* button to save the files.

18 Conclusion

This concludes the *Overview* tutorial. You may continue to experiment with the *SMS* interface or you may quit the program.