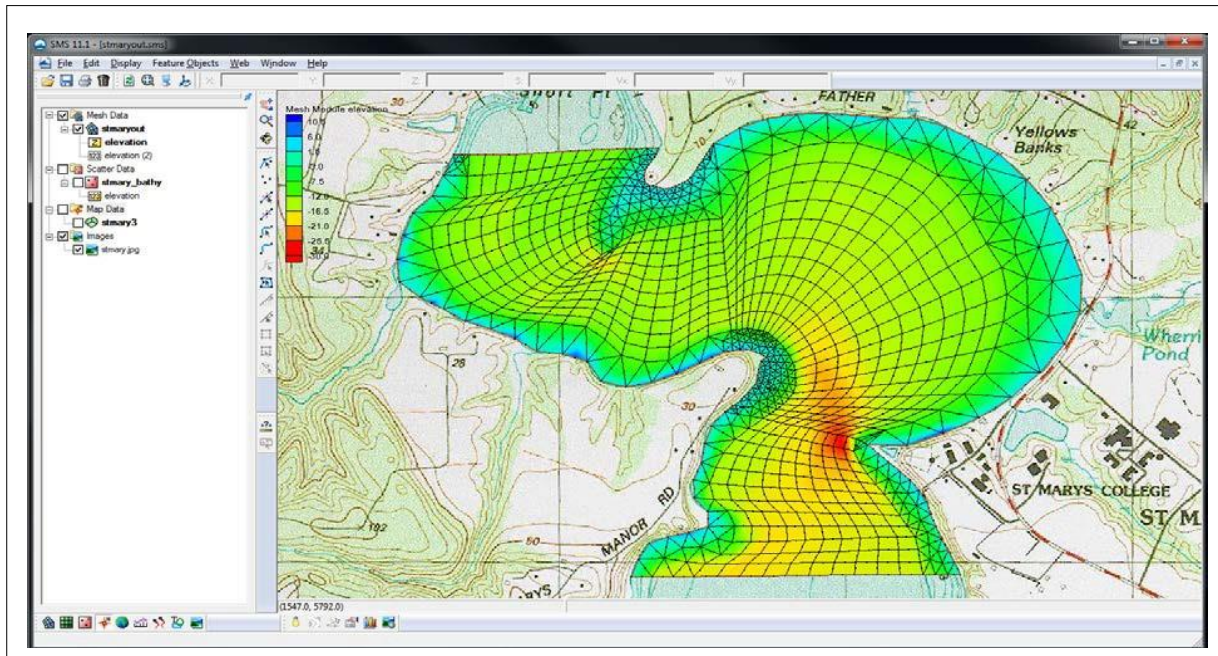


SMS 12.2 Tutorial Overview



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
- Mesh Module
- Scatter Module
- Map Module

Time

- 45–60 minutes

AQUAVEO™



1	Getting Started	2
2	Requirements.....	3
3	The SMS Screen	3
3.1	The Main Graphics Window	4
3.2	The Toolbars	4
3.3	The Project Explorer	5
3.4	Time Steps Window	6
3.5	The Edit Window	6
3.6	The Menu Bar.....	7
3.7	The Status Bars.....	7
4	Using a Background Image	7
4.1	Opening the Image	7
5	Using Feature Objects	8
6	Creating Feature Arcs	9
7	Manipulating Coverages.....	11
8	Redistributing Vertices.....	11
9	Defining Polygons.....	12
10	Assigning Meshing Parameters.....	13
10.1	Creating a Refine Point for Paving.....	13
10.2	Defining a Coons Patch	14
11	Assigning Materials.....	16
11.1	Assigning Materials to Polygons	17
11.2	Displaying Material Types	17
12	Converting Feature Objects to a Mesh	18
13	Editing the Generated Mesh	19
14	Interpolating to the Mesh	19
15	Converting Feature Objects to a 2D Grid.....	20
15.1	Creating a Grid Frame	20
15.2	Editing the Grid Frame	21
15.3	Creating a New Dataset	22
15.4	Generating a Cartesian Grid	22
16	Saving a Project File	24
17	Conclusion	24

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 section 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.

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 12.2” folder.

4. Click on **SMS 12.2**.
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 Mesh and Grid modules must be available under the current SMS license. To check if these modules are enabled:

1. Click on the *Help* menu in SMS and select **Register** to open the *Register* dialog.
2. A list of components and the status of each are displayed in the dialog that appears.
3. Turn off *Show only enabled modules* to show both enabled and disabled components.
4. If desired, change the registration according to the project needs by clicking on the **Change Registration** to bring up the *Registration Wizard*.
5. Complete the steps in the *Registration Wizard* dialog to change the registered components.
6. Click **Close** in the *Register* dialog when done.

In general, purchasing any version of SMS should have these modules enabled.

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.

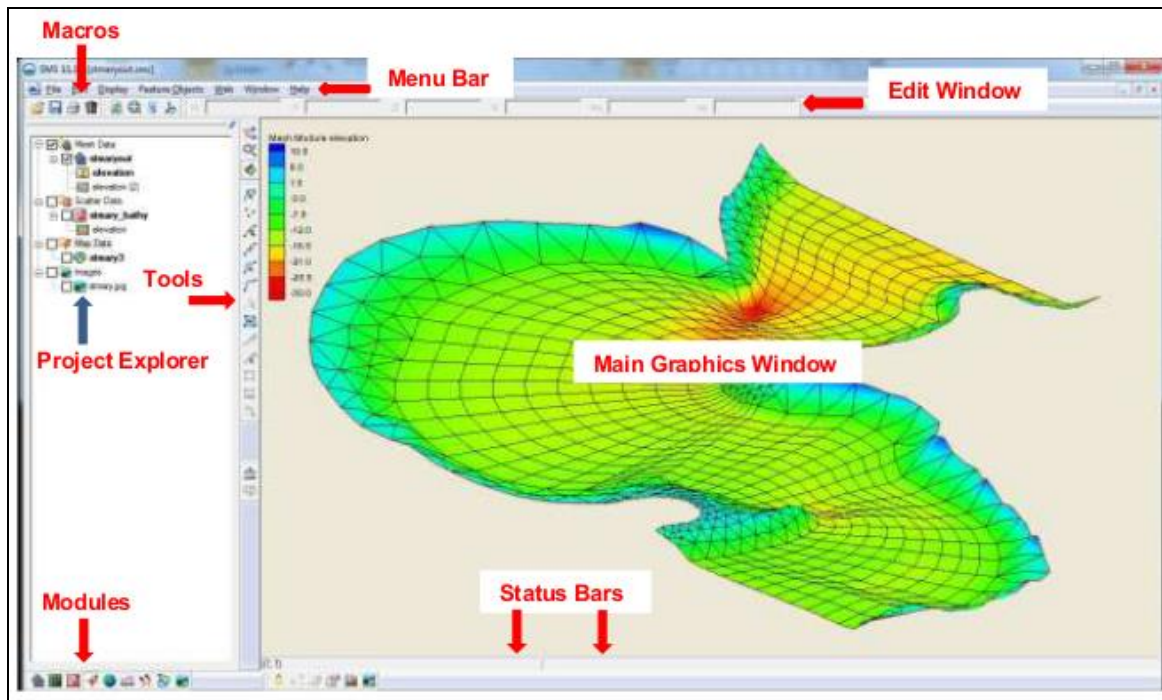


Figure 1 The SMS screen

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. 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 viewing 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, it is possible to also transform, copy, or manipulate the entity.

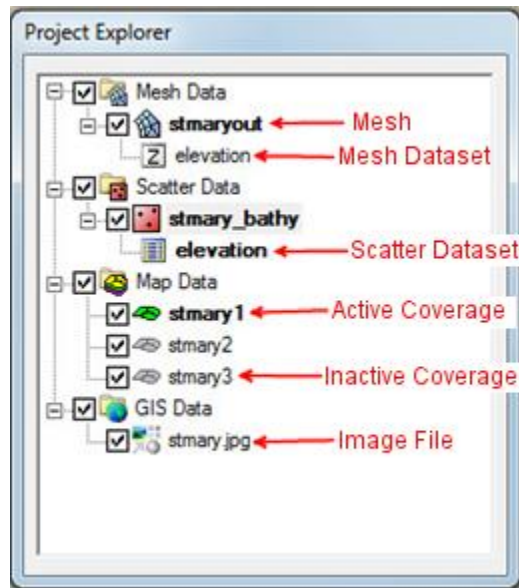


Figure 2 The Project Explorer window

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.

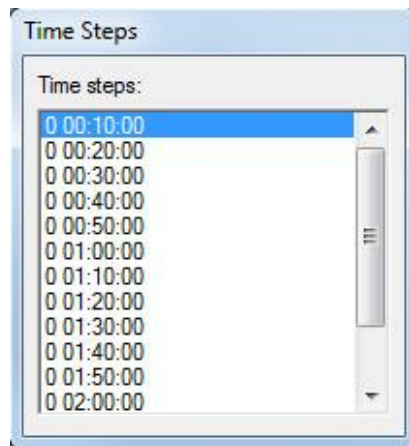


Figure 3 The Time Steps window

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.

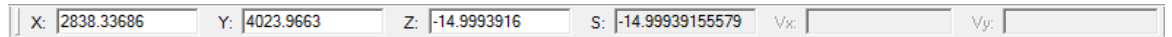


Figure 4 The Edit Window

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.

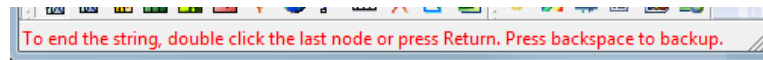


Figure 5 Status bar showing prompt

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.

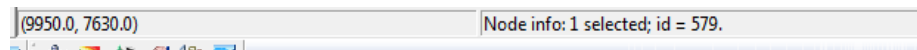


Figure 6 Status bar showing information for a selected entity

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, 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.

Notice 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.

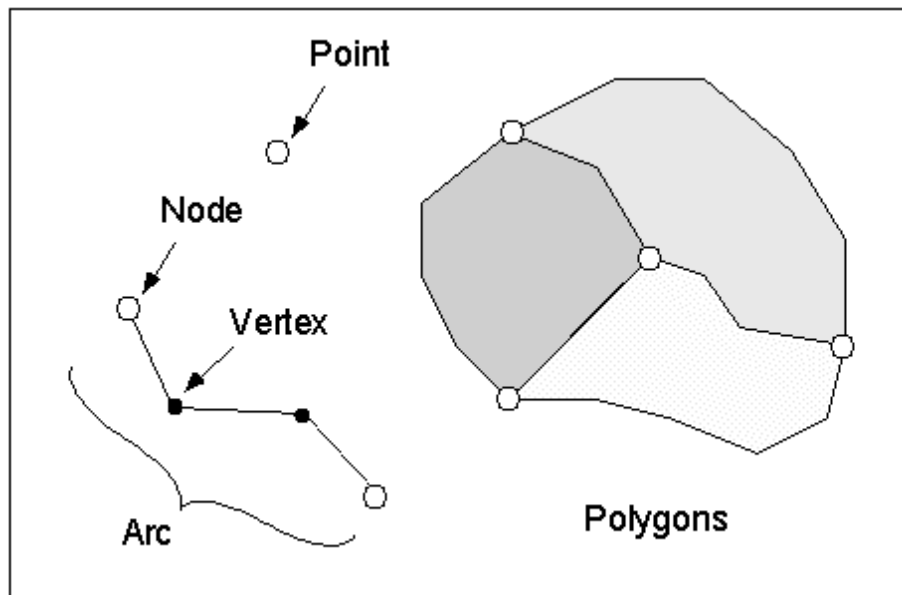


Figure 7 Feature Objects

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, new coverages will be created 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 dynamic toolbox.
3. Click out the left riverbank, as shown in Figure 8 (if necessary, use the **Zoom**  tool to get a closer view). 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.

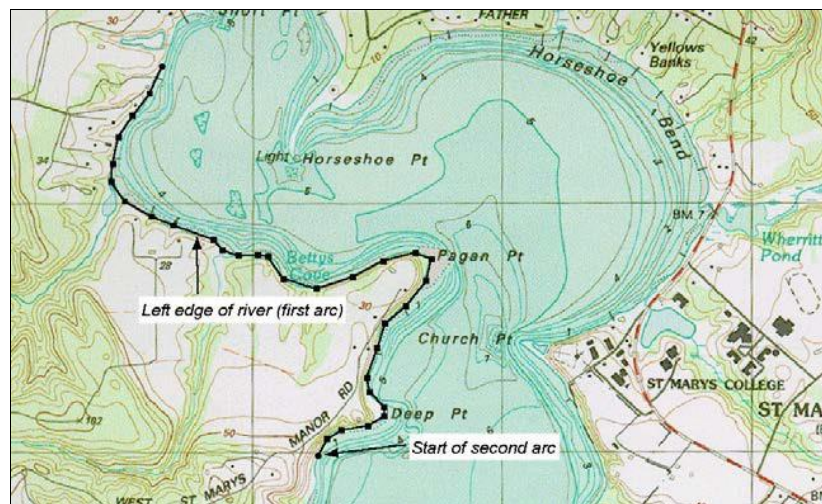


Figure 8 Creation of the first feature arc

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.

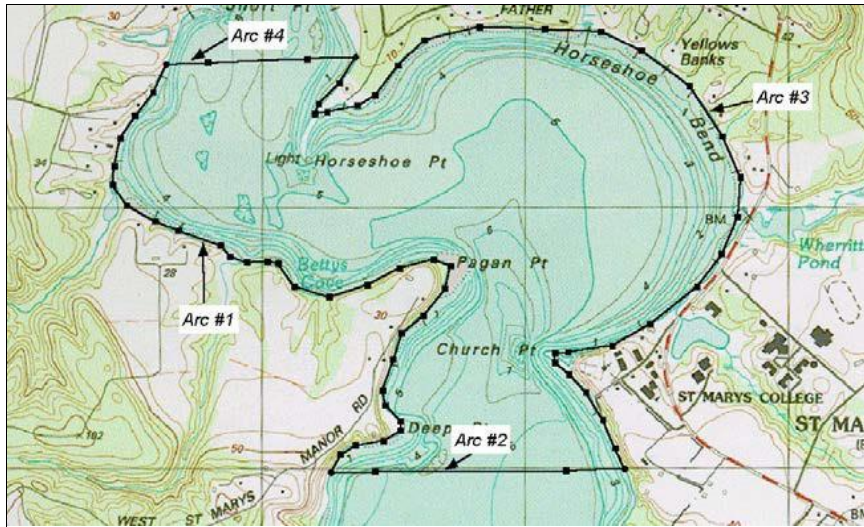


Figure 9 All feature arcs have been created

There is now a defined main river channel. When creating other models, normally 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** to bring up the *Open* dialog.
2. Select the file “stmary1.map” from the data files folder for this tutorial.
3. Click **Open**. A new coverage is created from the data in the file.

Notice that coverage is called “stmary1” is under the “Area Property” coverage. If the “Area Property” coverage was empty SMS would have 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.
5. Check the box next to “stmary1” to the data can be seen in the Graphics Window.

The display should look something like Figure 10.

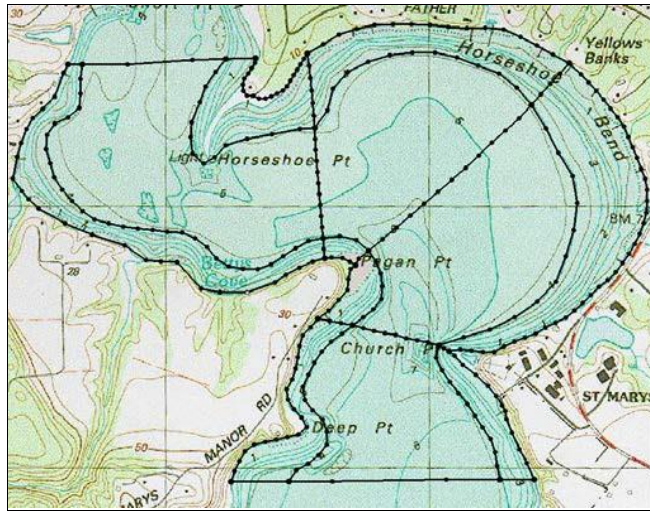


Figure 10 The stmary1.map feature object data

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 option 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.

Coverages can also be organized in folders by right-clicking on the Map Data item and selecting the **New Folder** command. Coverages can be clicked and dragged under folder items in the Project Explorer. The folders can then be turned on or off to show or hide all coverages in that folder.

8 Redistributing Vertices

To create the feature arcs, a line of points were simply clicked out over the image. The spacing of the vertices along the arc may not have been noticed. 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.
3. The addition of the conceptual model split *Arc #1* into four segments; hold the *Shift* key down to select all four segments.
4. Select *Feature Objects* / **Redistribute Vertices**. The *Redistribute Vertices* dialog shows information about the feature arc segments and vertex spacing.
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 ft apart from each other (or 200 m apart if working in metric units).
6. Click **OK** to redistribute the vertices along the arc.

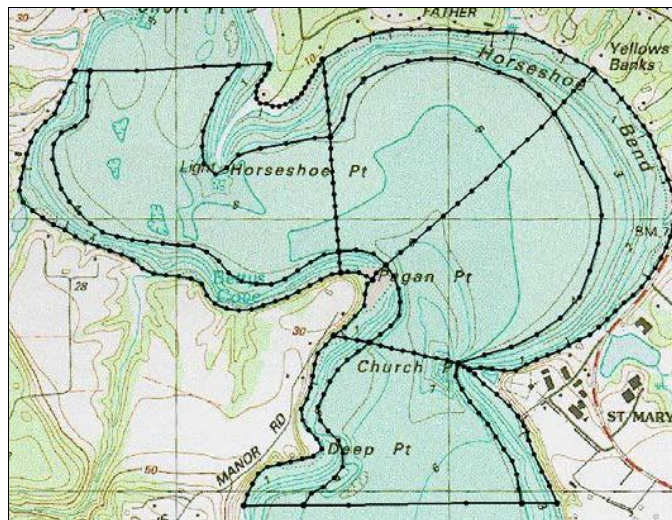


Figure 11 Redistribution of vertices along arcs

After using the *Redistribute Vertices* dialog, the display will refresh, showing the specified vertex distribution. The arc will still be highlighted, because it is still selected.

7. 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

Open another map file, which has the vertices redistributed on all the arcs.


1. Select *File* / **Open** to bring up the *Open* dialog.
2. Select the file “stmary2.map” from the data files folder and click **Open**.
3. Turn off the display of the “stmary1” coverage.

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

4. Right-click on the coverage “stmary2” in the Project Explorer.
5. From the menu, choose *Type* | *Generic* | **Mesh Generator**.

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:

6. Make sure no arcs are selected by clicking in the Graphics Window away from any arcs.
7. Select *Feature Objects* | **Clean** to bring up the *Clean Options* dialog.
8. Click **OK** in the *Clean Options* dialog. This will make sure there are no problems with the feature objects that were created.
9. 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 three or four 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 12. This will bring up the *Refine Attributes* dialog.
3. Make sure the *Refine Point* option is checked.
4. Enter an *Element size* 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.

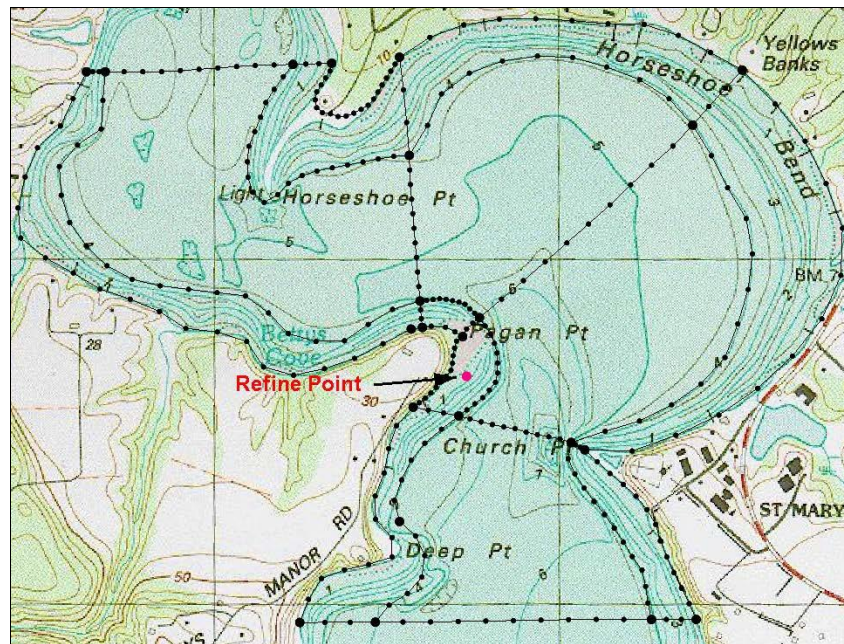


Figure 12 The location of the refine point

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.

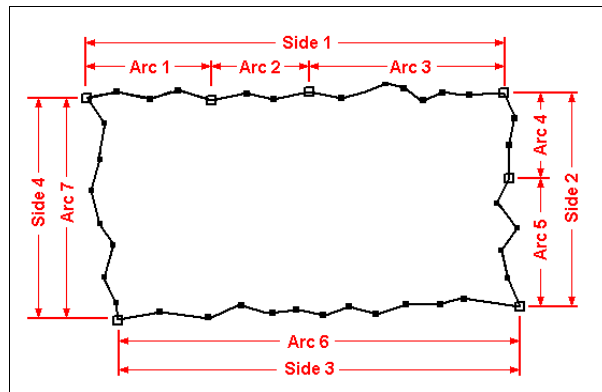


Figure 13 Four sides required for a rectangular patch

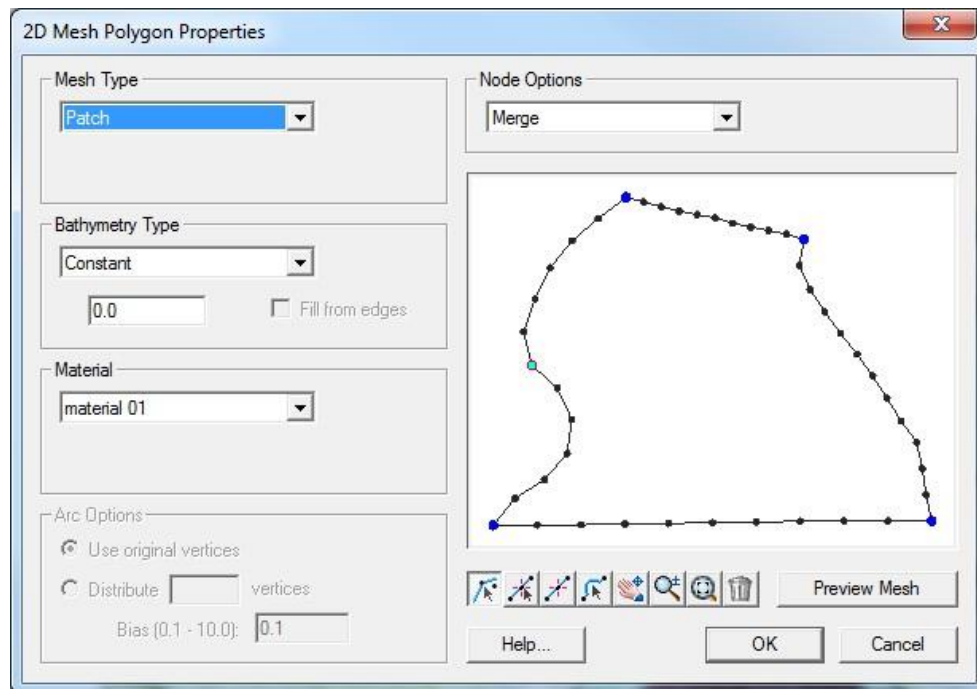




Figure 14 The Feature Polygon Attributes dialog

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 14 above). The *2D Mesh Polygon Properties* will 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 assigning the meshing type to be “Patch” before merging the node, SMS pops up a message box indicating that three or four sides need to exist for a patch.) If wanting to preview the patch, click the **Preview Mesh** button.
6. Click the **OK** button to close the *2D Mesh Polygon Attributes* dialog.

11 Assigning Materials

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 to bring up the *Open* dialog.
2. Select the file “stmary3.map” from the data files folder for this tutorial and click **Open** to import the file.
3. Right-click on the coverage “stmary3” in the Project Explorer.
4. From the menu, choose *Type* | *Generic* | **Area Property**.


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.

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** to bring up the *Materials Data* dialog.
2. Select the **New** button. This creates a new material named “material 02”.
3. Select **New** again to create “material 03”.
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”.
7. If desired, change the colors/patterns for all three materials by using the *Pattern* drop-down menu to the right of each material.
8. Click **OK** to close the *Materials Data* dialog.

11.1 Assigning Materials to Polygons

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 15 below.
3. Right-click on any of the selected polygons and select **Attributes...** to open the *Land Polygon Attributes* dialog.
4. Select *Material* and change it to “Left Bank” from the drop-down menu.
5. Click the **OK** to close the *Land Polygon Attributes* dialog.
6. Deselect the polygons by clicking anywhere on the image outside the polygons.
7. Repeat the previous steps 1–5 for the Main Channel and Right Bank, being sure to select the appropriate material in the *Land Polygon Attributes* dialog (“Main Channel” for the Main Channel polygons, and “Right Bank” for the Right Bank polygons).

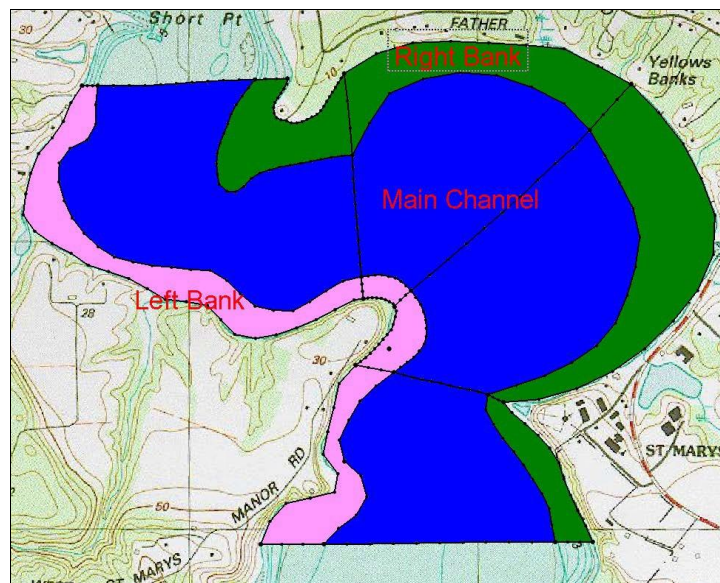



Figure 15 Polygons with defined material types

11.2 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 to bring up the *Display Options* dialog.
2. 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.
6. After viewing the materials, repeat the previous steps to turn off the *Fill* option before continuing with the tutorial.

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

12 Converting Feature Objects to a Mesh

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


Start with a new map file that has everything set up to generate the mesh. Import this file by doing the following:

1. Select the *File* | **Open** command to bring up the *Open* dialog.
2. Select the file “stmary4.map” from the data files folder for this tutorial and click **Open** to import the file.
3. Right-click on the coverage “stmary4” in the Project Explorer.
4. From the menu, choose *Type* | *Generic* | **Mesh Generator**.

To generate the mesh 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. Accept all default options and 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 hide the feature arcs and the image. To hide the feature objects, do as follows:

1. To hide the feature objects, uncheck the toggle box next to the Map Data folder in the Project Explorer.
2. To hide the image, uncheck the toggle box next to the “stmary” image icon under the “GIS Data” folder in the Project Explorer.
3. 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 16. With the feature objects and image hidden, the mesh can be manipulated without interference, but they are still available if mesh reconstruction is desired.

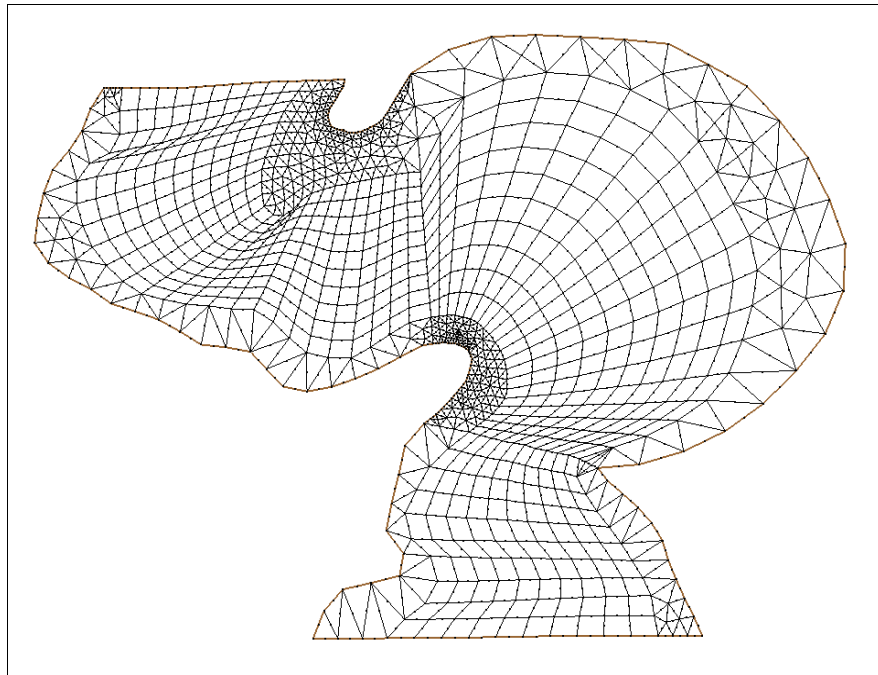


Figure 16 The generated finite element mesh

13 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.

14 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 and click **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:

3. Make sure the “stmary_bathy”  scatter module is active.
4. Select *Scatter* / **Interpolate to Mesh** to open the *Interpolation* dialog.
5. Make sure “Linear” is selected from the *Interpolation* drop down list. (For more information on SMS interpolation options, see *SMS Online Help*.)
6. Turn on the *Map Z* option at the lower left area of the dialog under *Other Options*.
7. Click the **OK** button to perform the interpolation.
8. 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 see if the scatter data was correctly interpolated to the mesh, do the following:

9. Select the **Rotate**  tool then click and drag in the Graphics Window to view how the elevation data was interpolated to the mesh.
10. Click on the **Plan View**  macro when done rotating the mesh.

15 Converting Feature Objects to a 2D Grid

The process of creating a grid feature objects is similar to the process of creating a finite element mesh. A grid can be created using any coverage that works with a grid model.


15.1 Creating a Grid Frame


Before features objects can be converted to a 2D grid, the area of the grid needs to be defined. The grid area is determined using a grid frame created in a map coverage. This is done using a grid frame.

Start by doing the following:

1. Turn on the toggle box next to the Map Data folder and turn off the box next to the Mesh Data folder in the Project Explorer

2. Select *File* | **Open** to bring up the *Open* dialog
3. Select the “stmary5.map” file and click **Open** to import the file.
4. In the Project Explorer, select the “stmary5” item to make it active then right-click and select *Type* | *Generic* | **CGrid Generator**.

The **Create 2D Grid Frame**  tool is now active and can be used. A grid frame is created by clicking in three locations in the Graphics Window. Each location will become a corner of the grid frame. SMS will automatically calculate the fourth corner based on the location of the other three corners.

5. Select the **Create 2D Grid Frame**  tool and click in location 1 as shown in Figure 17.
6. Click on location 2 then click on location 3 to complete the grid frame.

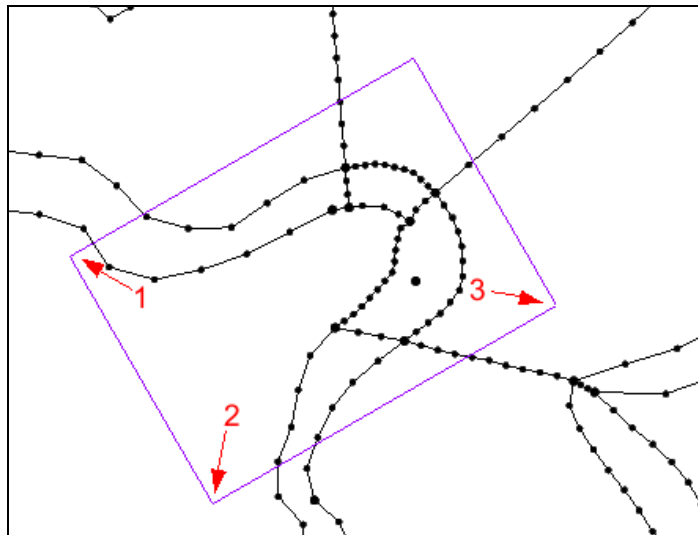




Figure 17 Grid frame locations

15.2 Editing the Grid Frame

The grid frame can be edited by using the **Select 2D Grid Frame**  tool. This tool is active when a grid frame exists in a coverage. It can be used to manually resize the grid frame.

1. Select the **Select 2D Grid Frame**  tool then click and drag the right side of the grid frame to resize it.
2. Click on the circle in the lower right corner of the grid frame then click and drag to rotate it.

Manual resizing of the grid frame is useful, but not very precise. A more accurate way to edit the grid frame is by using the *Grid Frame Properties* dialog.

3. Using the **Select 2D Grid Frame**  tool, double-click on the grid frame. The *Grid Frame Properties* dialog will appear.

4. Enter the following parameters to make the grid frame consistent:
 - *Origin X*: 4400
 - *Origin Y*: 765
 - *Angle*: 30
 - *I size*: 2500
 - *J size*: 1800
5. For the *Cell size*, enter “100” in both locations. This will be used when generating the 2D grid.
6. When done, click **OK** to close the *Grid Frame Properties* dialog.

The grid frame should now appear similar to Figure 17. The grid frame has now been resized and is nearly ready for conversion.

15.3 Creating a New Dataset

The mesh created earlier in this tutorial used bathymetric survey data which was interpolated to the mesh after the mesh was generated. This data is not useable for the Cartesian grid. However, the data can be used to generate a more acceptable dataset using the *Dataset Toolbox*.

The *Dataset Toolbox* holds a number of tools that are used to generate new datasets from existing data.

1. Select the “stmary_bathy” dataset to make the Scatter module active.
2. Select the *Data | Dataset Toolbox* command to bring up the *Dataset Toolbox* dialog.
3. Under *Tools*, select the *Data Calculator* option.
4. Under *Datasets*, select “d1.elevation” and click the **Add to Expression** button.

In the *Calculator* field, the number “d1” is shown. This represents the elevation dataset.

5. Click on the multiplication * button then type “-1” in the *Calculator* field to make the final equation read “d1*-1”.
6. In the *Output dataset name* field enter “depth”.
7. When done, click the **Compute** button. A new dataset will be generated inverting the elevation values so they can be used as depth values.
8. Click **Done** to close the *Dataset Toolbox* calculator.

There is now a new scatter set that can be used when generating the 2D grid.

15.4 Generating a Cartesian Grid

With the grid frame defined and depth dataset ready, now convert the feature objects to a Cartesian grid.

1. Right-click on the “stmary5” coverage in the Project Explorer and select the *Convert* | **Map**→**2D Grid** command. This will bring up the *Map*→*2D Grid* dialog.

Most of the options in the *Map*→*2D Grid* dialog were set earlier in the *Grid Frame Properties* dialog. Had these options not been specified then, the grid frame dimension and options could be specified now.

When the mesh was created earlier in this tutorial, the elevation was interpolated to the mesh after the mesh was generated. The Cartesian grid generation process allows datasets to be assigned during the generation process using the *Map*→*2D Grid* dialog.

2. Under *Depth Options*, change the *Source* to “Scatter Set” then click the **Select** button to bring up the *Interpolation* dialog.
3. Select the “depth” dataset under the *Scatter Set To Interpolate From* section.
4. Leave the other options with the default settings and click **OK** to close the *Interpolation* dialog.
5. Click **OK** to close the *Map*→*2D Grid* dialog and generate the grid.

A Cartesian grid has now been generated. To better view the Cartesian grid, complete the following:

6. Hide all Scatter Data, Map Data, GIS Data, and Mesh Data by unchecking the box next to each item.
7. Select *Display* | **Display Options** to bring up the *Display Options* dialog.
8. Select *Cartesian Grid* from the list on the left.
9. Click the **All Off** button then turn on the *Contours* option.
10. Under the *Contours* tab, change the *Contour method* to “Color Fill”
11. Click **OK** to close the *Display Options* dialog.
12. Select *Display / View* | **Oblique** to see how the depth dataset was interpolated to the grid. It should appear similar to Figure 18.

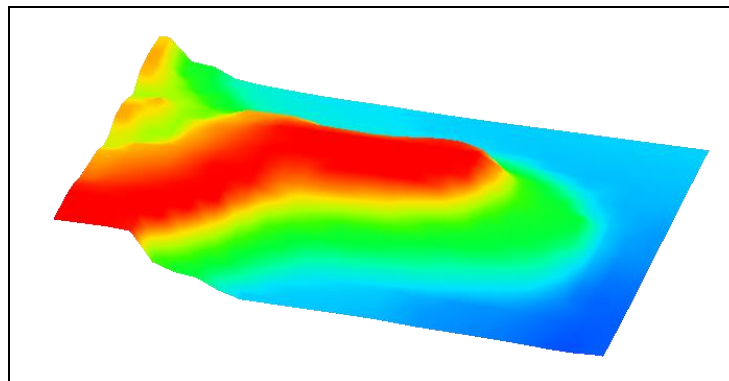




Figure 18 Oblique view of grid with contour options enabled

13. If desired, select the **Rotate**  tool then click and drag in the Graphics Window to further view how the depth was interpolated to the grid.

14. Click on the **Plan View**  macro when done rotating the grid.

16 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 *Save* dialog.
2. Enter a *File name* of “stmaryout.sms” then click the **Save** button to save the files.

17 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
- Assigning materials
- Basic mesh generation
- Basic dataset creation
- Basic grid generation
- Saving a project file

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