SMS User Manual 11.2, Vol. 2

Modules

Contents

3. Modules

Modules

SMS Modules

The commands in SMS are divided based on the types of data they operate on. As you switch from one module to another module, the Dynamic Toolbar and available menu commands change. This allows you to focus only on the tools and commands related to the data you are currently working with in SMS. Switching from one module to another can be done instantaneously to facilitate the simultaneous use of several data types when necessary. Only one module is active at any given time. However, the data associated with a module (e.g. a 2D finite element mesh) is preserved when the user switches to a different module. Activating a module only changes the Dynamic Toolbar and available menu commands.

Module Selection

There are several ways to switch from one module to another. These include:

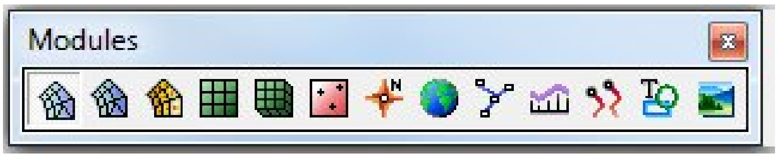
- Select an entity in the *project explorer*. The module containing the active entity becomes active.
- Right-click on the *project explorer* and select the **Switch Module** command.
- Click on the module icon in the module toolbar. The module toolbar is displayed at the bottom of the project explorer by default.

(Note: Switching modules should not be confused with changing the current model inside of a module. When a new model is selected, the tools and menus may change, and the data will be converted as much as is possible. However, some data may be lost.)

Modules in SMS:

- Mesh Module
-  Curvilinear Grid Module
- Cartesian Grid Module
- 3D Cartesian Grid Module
- Scatter Module
- Map Module
- GIS Module
- 1D River Module
- 1D Grid Module
- Particle Module
- Images
- CAD Data

Module Toolbar



Module Toolbar

The *Module Toolbar* is used to switch between modules. Only one module is active at any given time. However, the data associated with a module (ex. a 2D finite element mesh) is preserved when the user switches to a different module. Activating a module simply changes the set of available tools and menu commands.

Annotations

Annotation Objects

The XMS application family provides a series of tools to annotate the data in an application for presentations, animations and screen shots.

These tools (annotation objects)are accessed through the Annotations Module and include:

- Images
- North Arrows
- Scale Bars
- Text
- Lines
- Ovals
- Rectangles

Screen vs World Space Layers

All annotation layers either contain objects referenced to world or screen coordinates. Objects referenced to world coordinates will change size and position on the screen with the underlying data. This is useful to identify specific locations in your model such as peir locations. Objects associated with screen coordinates do not move on the screen with the underlying data. This is useful for titles, legends such as north arrows and scale bars, and logos. Some types of annotations can only be created in screen space layers including North Arrows, Images, and scale bars.

If the first annotation object you create, could be part of a screen or world space layer SMS will ask which type of layer you wish to create and add the object to. You can create additional layers by right clicking on the *Annotation Data tree* item and selecting ***Create Screen Space Layer*** or ***Create World Space Layer***. Layers are differentiated by including an 'S' for screen space layers or 'W' for world space layers in their icons in the project explorer.

If multiple layers exist, any newly created annotation object will be placed in the "current" layer.

Annotation Object Attributes

The extents of annotation objects defined by a frame. The user defines this frame initially when creating the annotation object by left clicking at any point on the screen and dragging a rectangle with the mouse (left button still down). The display will show the frame while you drag with the mouse. (Points and lines defining degenerate frames are not allowed.) When the user creates a annotation, if the frame is too big for the window, it will be resized appropriately. Annotations can't be resized or moved even partially outside of the borders of the window. If the user resizes a annotation through a quick mouse drag and the cursor lands outside the window, the annotation will be redrawn to take up all the window space in that direction.

This frame bounds the region of the screen where the object will appear with the modeling data. The user interacts with the object by interacting with its frame and specifying its attributes or properties (see the section on selection below). The frame anchors the annotation object on the screen. This anchoring defines both the size and position of the object. The x-location, y-location, x-size and y-size are all defined independently as either a pixel value or percentage of the screen.

Horizontally, the user can position the left edge, the right edge or the center of the object. If the user positions the left edge, the object position is defined relative to the left edge of the screen. If the user positions the right edge, the object position is defined relative to the right edge of the screen. If the user positions the center of the object, the object position is defined relative to the horizontal center of the screen.

For example, the left side of the frame may be specified as 100 pixels from the left edge of the screen. Alternatively, the user may specify that the right edge of the frame should be 10% of screen width from the right edge. Finally, the user may specify that the center of the object is 100 pixels to the right of the center of the screen.

The vertical position and sizes of the object are similarly specified in the anchoring attribute of the object.

All annotation objects also have attributes. The specific attributes depend on the type of object. The attributes define color, line thickness, fill properties, associated images, etc.

Screen Space Images

A screen space image is simply a graphics icon mapped to the screen. A typical application would be to display a company, department, or municipality logo next to the numeric model being displayed in the graphics window.

Attributes of the screen space images include:

- General anchoring attributes
- The image file being displayed as part of the project
- Whether the image is being displayed as a scaled (distorted object), scaled based on its original aspect ratio, or locked at another aspect ratio.
- Transparency – The image properties dialog have a transparency checkbox. When checked it will cause the image to be redrawn with the most used color in the image. When that it is checked, it also causes the color checkbox and the tolerance edit field to become available. If the color checkbox is checked, it will activate the color button and the color button will have the latest chosen image color painted on it or the most used color in the image, if it has not been activated before. Clicking on the down arrow part of the color button causes a color popup to be displayed with swaths of the 40 most used colors in the image or all the colors in the image, if the image has less than 40 colors. Clicking on one of those colors will cause the image to be redrawn with that color made transparent in the image. The tolerance edit field allows for variation in the matching of the red, green and blue components. The tolerance field ranges in allowable values from 0.0 to 1.0. 0.0 means the red, green and blue components must exactly match. Values higher than 0.0 indicate the degree of variation from the given color. Clicking the transparency checkbox to the off state causes the image to be redrawn with no transparency.

Scale Bars

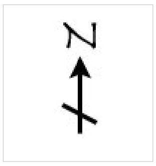
A scale bar occupies a fixed size of the screen to display the relative size of the objects in the simulation. The user defines the minimum width of the scale bar section (in pixels), along with a minimum and maximum height of the scale (also in pixels). The XMS application adds a "Units" label (meters in the image shown below) and labels for the model distance related to the scale divisions.

The XMS applications will compute a well conditioned number to use as the scale increment that fits in the specified scale bar extents.

Attributes of the scale bar include:

- General anchoring attributes.
- The minimum spacing between distance labels.
- The minimum division width (in pixels). The XMS application determines the number of divisions based on the minimum divisiion width and the width of the frame.
- The minimum and maximum height of the scale bar object.
- The font (style, color, size) used to label the scale bar.
- Whether or not the area behind the scale bar will be filled, and if so, with what color.

North Arrows



North arrow objects consist of automatically rotating screen space images. When an XMS application is installed, at least one default North Arrow image will be included in the application's home directory. Users may create or download as many north arrow icons as desired. These icons are displayed at the specified location (anchored with the standard options), but will rotate as the view direction changes so that the "up" direction of the icon always aligns with the "North" or positive "Y" direction.

Attributes of north arrows include:

- See screen space image attributes.

Text

Text can be created in world or screen space layers.

You can enter text by clicking in the graphics window with the **Create Text** tool active.

You can set the following attributes for text:

- Font – including size
- Color
- Background – fill behind with the background color or another color
- Border and border thickness

Lines/Arrows

You can create Lines/Arrows using the **Create Line** Tool. Lines/Arrows can be created in screen or world space layers.

The attributes available for Lines/Arrows include:

- Type – Dashed or solid w/ thickness
- Color
- Arrowheads – location (beginning, end, both) and size

Ovals

You can create ovals by dragging a box with **Create Oval** tool active. Ovals can be created in world or screen space layers.

The attributes that can be specified for ovals include:

- Line type – dashed/solid with width
- Line color
- Fill and color or No Fill

Rectangles

Rectangles can be created in screen or world space layers. Their attributes are the same as those available for ovals.

Selection

The first toolbar button is used to select and set attributes for annotation objects. This requires that objects exist to be selected. In this case when you press this tool and then left click in the annotation object, the object frame will be drawn around the annotation. In addition to the frame, the XMS application displays grab handles on the corners and edges of the frame. The user modifies the rectangular shape of the annotation by dragging one of the grab handles and changes the position of the object by dragging the annotation (click at any point in the object interior).

When this tool is active the right mouse button will pop up a menu of all dialogs for that particular type of annotation. This will include at least "Attributes" which will display a dialog of the objects attributes for editing.

Viewing Annotations At Specific Time Intervals

Available in SMS 11.1 and in development for GMS, annotations can be setup to be viewed at specific time intervals. To setup annotations so they only are displayed at specified time intervals do the following:

- Right click on the Annotation layer in the tree then select *Properties...*.
- This dialog will display the *Annotation Layer Properties* dialog.
- Check the *Apply time range* checkbox
- Modify the "begin" and "end" time controls to specify the range for when annotations are visible.
- Click **Ok**.

Now film loops and time steps will only display the annotation when its within the range specified.

3.1. 1D Grid Module

1D Grid Module

The 1D grid module contains an interface for the GenCade shoreline morphology model.

1D Grid Module Tools

These tools allow constructions of a 1D grid, shorelines associated with the grid, and structures such as seawalls, groins and breakwaters associated with that shoreline.

- **Select Point**
- **Create Point** – Create a point along the coastline.
- **Select Detached Breakwater**
- **Create Breakwater**
- **Select Jetty or Groin**
- **Create Jetty or Groin**
- **Select Seawall**
- **Create Seawall**

See 1D Grid Module Tools for more information.

1D Grid Module Menus

The following menus are available in the the 1D Grid module.

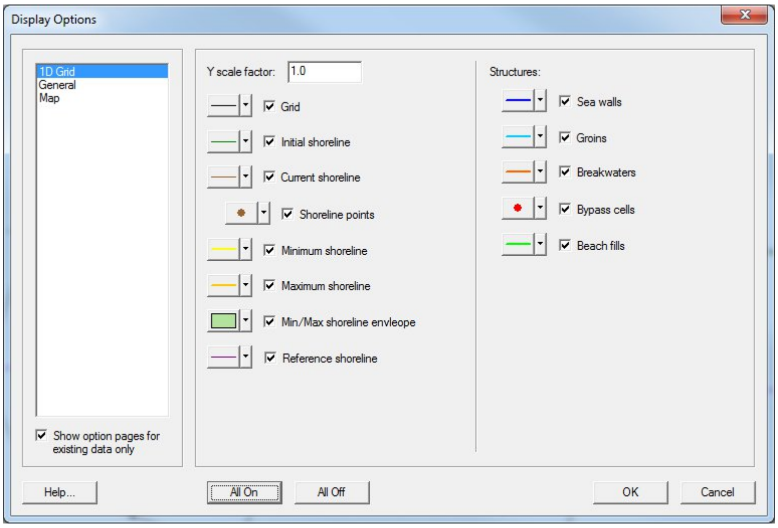
- Standard Menus – See Menu Bar for more information.
- Data – See Scatter Data Menu for more information.
- GenCade – See GenCade Menu for more information.

Related Topics

- 1D Grid Display Options
- Modules
- GenCade

1D Grid Display Options

The properties of all 1D Grid data that SMS displays on the screen can be controlled through the *One Dimensional Grid* tab of the *Display Options* Dialog. This dialog is opened by right-clicking on the GenCade Data entry in the *Project Explorer* and selecting the **Display Options** command. (It can also be accessed from the from the *Display* menu or the *Display Options* Macro.)



Display Options dialog showing *1D Grid* options

The entities associated with the 1D Grid module with display options are shown below. Some of these entities also show an **Options** button to the right. For these entities, additional display controls are available. The available one dimensional grid display options include the following:

Grid Objects

- Y Scale Factor – magnification in the direction perpendicular to shore
- Grid display
- Initial Coastline display
- Current Coastline display (based on dataset)
    - Coastline points
- Minimum/Maximum Coastline and Envelope (zone covered by transient coastline and extremes based on dataset)
- Reference Coastline

Structure Objects

- Sea Walls
- Groins
- Breakwaters
- Bypass Cells
- Beach Fills

Related Topics

1D Grid Module

1D Grid Tools

The 1D grid tools are available in the 1D grid module when working with the GenCade model. These tools allow the user to edit the features on a 1D grid. These festures include:

- Shorelines associated with the grid
- Structures along the shoreline:
  - Seawalls
  - Groins
  - Breakwaters

It is recommended that you manage a GenCade project through a conceptual model in a GenCade coverage. The conceptual model allows more flexibility when specifying the structures because it works in real world space. These tools work on the 1D grid using grid cell indices and distances from the grid to locate the objects. The GenCade menu also provides commands to edit the objects.

The tools include:

- **Select Point –** This tool allows the user to select a point on the initial shoreline defined for the grid and drag it closer to or farther away from the grid. Since the distance along the grid is not variable for the selected point, the edit only affects the local "Y" value of the grid point. Since the grid usually consists of many points along the shoreline, editing the shoreline with this tool can be tedious.
- **Create Point –** This tool is currently disabled. With other 1D grid models that have been supported in the SMS package, creating grid points using this tool was supported. The only 1D grid model currently included in the package (GenCade), does not support this feature. This tool would be used to create a point along the initial coastline.
- **Select Detached Breakwater –** This tool is used to edit a breakwater positioned along the grid. With this tool active, a click in the graphics window selects an endpoint of a breakwater, and dragging the mouse with the end point selected moves the endpoint of the breakwater. The depth, transmission and permeability of the breakwater must be assigned using either the *GenCade*|**Detached Breakwaters** command or by assigning these attributes to an arc in the conceptual model.
- **Create Breakwater –** This tool can be used to create a breakwater in a simulation. Clicking in the graphics window with this tool active defines a starting point for a breakwater. SMS will then draw a "rubber band line" from this location to the cursor until a second location is clicked, terminating the breakwater. Attributes for the breakwater must be assigned using either the *GenCade*|**Detached Breakwaters** command or by assigning these attributes to an arc in the conceptual model.
- **Select Jetty or Groin –** This tool is used to edit the length of a groin or jetty positioned along the grid. With this tool active, a click in the graphics window selects an endpoint of a groin , and dragging the mouse with the end point selected moves the endpoint of the groin, thus changing its length. Other attributes of the groin/jetty must be assigned using either the *GenCade*|**Groins** or *GenCade*|**Inlets** commands or by assigning these attributes to an arc in the conceptual model.
- **Create Groin –** This tool can be used to create a groin in a simulation. Clicking in the graphics window with this tool active defines the end point for a groin. SMS will connect this location to the grid defining the groin. Attributes for the groin must be assigned using the GenCade|Groins command or by assigning these attributes to an arc in the conceptual model.
- **Select Seawall –** This tool is used to edit the shape of an existing seawall along the grid. With this tool active, a click in the graphics window selects a point in a seawall. Dragging the mouse with the point selected modifies the seawall. As with grids, seawalls often include multiple segments making them tedious to edit using this approach.

- **Create Seawall** – This tool can be used to create sea wall segments along the grid. With this tool active, a click in the graphics window starts the creation of a segment. A second click terminates the segment. When the segment is complete, SMS determines if this segment overlaps existing segments and trims the existing segments to the new segment if an overlap exists.

Related Topics

- 1D Grid Module

3.2. 1D River Module

1D River Module

The purpose of the River Module (also referred to as the 1D Module) is to provide tools to perform one dimensional (Step Backwater) hydraulic modeling within SMS. The module include tools to process digital terrain (scattered datasets) and map data (coverages) to build the basic geometry necessary for a 1D Hydraulic Model. Much of the information for developing models with these tools is described in the information on River Coverages in the Map module.

The *1D River Hydraulic* tools are used to create, edit and apply networks of river reaches and cross-sections. The created networks must be associated with a specific 1D model (currently the only major model supported is HEC-RAS. The module includes tools for visualizing results computed for cross sections and reaches.

The general process for developing a model consists of the following steps:

1. Preparing a background digital terrain model that represents the river channel bathymetry and surrounding flood plain with enough detail to substantiate the modeling objectives.
2. Developing a 1D-Hydraulic Centerline coverage including the centerline and optional bank arcs.
3. Creating the cross section arcs at important/required locations along the section of river being modeled.
4. An Area Property coverage can be used to map roughness values to line properties on the cross sections
5. Extracting cross sections from the TIN and establish the 1D Model
6. Export the GIS data and finish defining HEC-RAS (or other models).

Alternatively, It is possible to establish the hydraulic model without extracting cross section information from a TIN. Cross sections which have already been surveyed can be assigning to an arc. This, along with geo-referencing the data is done using the cross section editor from the *River Tools* menu in the Map module.

The heart of the 1D River Hydraulic Module is a database of cross sections. A set of tools called the *Cross-Section Database Manager* allow the user to edit and manage the cross sections. All cross sections referenced from the 1D River Hydraulic Module or the Map Module must be part of this database. Cross sections may be added to the database by importing data from an external source, hand editing in the *Cross-Section Database Manager*, or extraction from topographic data in the Map Module.

River networks are displayed as a schematic of reaches and cross sections. Networks may be georeferenced or purely symbolic. Each cross-section is represented by a cross-section icon displayed at the location of the cross section if the section is georeferenced, and reach icons, which are displayed on top of the first cross section in each reach. The icons can be moved according to the user’s preference by selecting the icon and dragging it to the desired location.

The cross-section attributes can be changed by using the *Select Cross-Section* Tool. Double- clicking on the desired cross-section opens the *River Cross Section Atts* Dialog which can be edited similar to the Map Module. The only difference is that the reach the cross-section is attached to cannot be changed. Likewise the reach attributes can also be changed within the 1D River Hydraulic module.

1D River Module Tools

The River Tools is not necessarily a model, but provides the tools and commands necessary to build 1D Hydraulic modeling data from feature object coverages. The River Tools primarily support the HEC-RAS model, but will be used to support additional hydraulic models in future versions.

The River tools provide the ability to define a hydraulic model using a 1D-Hydraulic Centerline coverage and a 1D-Hydraulic Cross-section coverage. The layout of the feature objects defining the centerline and cross sections establishes the direction, the stationing, and the topology (connectivity between cross-sections) of a hydraulic model. Further, using these two coverages cross sections may be automatically extracted from a digital terrain model and then edited, merged, or combined with other cross section information to provide the geometric basis of the model. An area property coverage can also be used to map materials (Manning's roughness coefficients) to the cross section based on some type of aerial distinction (land use or soils) that may be available. The diagram below illustrates how these coverages are used to establish a hydraulic model.

The river tools also allow you to interpolate cross sections to establish more cross section information in between surveyed or extracted cross sections.

Results data from HEC-RAS and other hydraulic models can be read back in and used to perform a flood plain delineation. The flood plain delineation algorithm in SMS works better with a denser set of resulting water surface elevation points and so there are river tools that allow a water surface elevation computed at a cross section to be interpolated (copied since it will be the same value) along cross section arcs, or along a centerline.

- **Select Cross-section**
- **Select Centerline**

Scatter Module Menus

See 1D River Module Menus for more information.

RivHyd Post-Processing

After a simulation has been successfully run the solution file contains data at the cross sections. This data normally includes velocity and water surface elevation. Depending on the analysis program used, other data may also be available. The solution data may vary across the cross section. These solution files may be read into SMS by using *File* | **Open**.

Datasets

Data read in from a solution file and associated with the 1D Hydraulic Module are stored as datasets with values at each cross section. In some cases, a cross section may have multiple values such as varied velocities in overbank and channel regions. These datasets can be managed through the *Project Explorer*.

Solution data at various cross-section and reach points for the active dataset may be exported to a text file by selecting *Data* | **Export Solution Data**. This will bring up the River Hydraulic *Solution Export* dialog box.

2D Plots

In order to see the model results, 2D plots must be generated in a plot window. (Future releases of SMS will support the mapping of 1D results onto a Triangulated Irregular Network representing the 1D simulation). The two types of plots that can be created are:

- Cross-section plots which show model results for one or more cross-sections.
- Profile plots which displays model results along a reach. Values for the profile plot are the minimum, average, and/or maximum of the results of individual cross-sections along the reach.

Both plots are created through the *Plot Wizard*.

RivHyd Data Export

The time steps to export are chosen in the left side of the dialog. Generally the original point locations for the cross-sections and reaches is sufficient, but if more dense information is desired the spacing can be set using the controls on the right side of the dialog. The data will be linearly interpolated along the cross-section or reach between data points. The filename can be chosen at the bottom of the dialog. In order to read the data into SMS as a scatter point file, the extension should be *.pts.

Related Topics

- Area Property Coverage
- 1D River Conceptual Model
- HEC-RAS
- 1D River Hydraulics Data Browser
- Modules

1D River Hydraulics Data Browser

The top list box shows the 1D River Hydraulic solutions. A solution is a list of functions. When a solution file from a numerical simulation is read in, the functions may be added to an existing solution or to a new solution. New solutions can be created and existing solutions deleted with the buttons on the right of this box. Solution names (except for the Generic solution) can be changed by clicking on desired solution and typing the name you want it to have. The box in the middle of the browser displays the scalar datasets in the currently selected solution. The names for these functions may be changed in the same manner as the solutions names. Datasets may also be moved to other solutions. The bottom box shows the time steps in the currently selected dataset. Clicking on the info button will bring up a dialog box which displays information about the selected time step of the selected dataset.

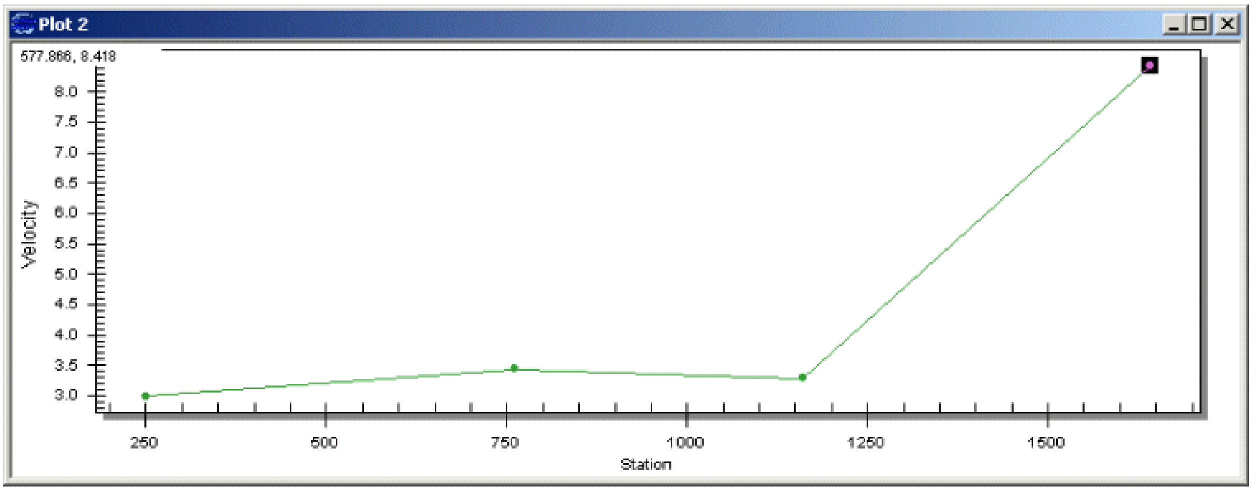
Related Topics

- 1D River Module

1D River Hydraulics Profile Plots

A 1D profile plot can contain either the currently selected river reaches or specific reaches specified in the second frame of the plot wizard. In addition, the plot may set to show the currently active dataset and time step, or the user may specify specific datasets and time steps.

The plot shows the dataset values at each cross-section along the reach. If values vary across the cross sections, the plot may be set to show the minimum, average, and/or maximum values at the cross section.



Related Topics

- Plot Window

1D River Module Menus

The following menus are available in the the 1D River Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- *Data*
  - **Create Visualization Tin**
  - **Rivhyd to Scatter**

Model Specific Menus

- *HEC-RAS*
  - **Delete Simulation** – The link SMS maintains to the current HEC-RAS simulation project file is deleted.
  - **Material Properties...** – SMS uses an Area Property coverage to map the roughness line properties to segments of a cross section. The cross sections store the material property ID's and in order for SMS to correctly associate a material ID with an actual roughness they must be defined using the *Material Properties* dialog.
  - **Model Control...**

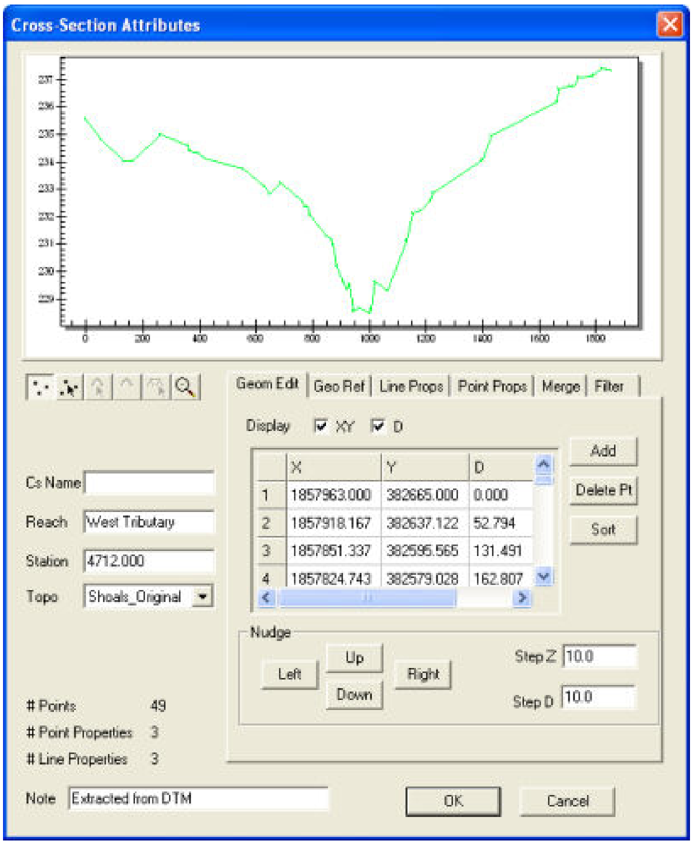
Editing Cross Sections

For the new 1D Hydraulic Cross Section coverage, the cross section geometry is stored in text database file on disk. When extracting cross sections they are saved to a new (or existing) database file. However, extraction of cross sections from digital terrain models is not the only way that they can be created, nor is extraction always the only thing that needs to be done. For example other ways cross sections can be entered for use include: including importing from a spreadsheet, or entering manually. In such cases, and many times after extraction from a digital terrain model there are edits that must be performed in order to prepare the cross sections for hydraulic modeling.

You can edit cross sections in one of three ways:

1. If you double click on an arc in a 1D Hydraulic Cross Section coverage you can assign a cross section from a database. After assigning the cross section you can also enter the editor for that cross section.
2. You can open a cross section database for editing (or create a new database) using the **Manage Cross Sections** command.
3. You can also open an existing cross section database using the *File* | **Open** command.

The operations described in the following paragraphs can be done using the cross section editor shown in the figure below.



General Properties

In order to identify information about the cross section in the database a name (not required), a reach, a station, and the name of the topographic data used to extract the cross section (if applicable) can be defined. A note about the cross section can also be defined. Not all of these attributes are critical for the development of a hydraulic model, but they are useful in managing the cross section within a database.

Editing Geometry

Cross section points can be added, or values edited when the *Geom Edit* tab of the editor is active. XY values are available when the actual 3D position of each point on the cross section is known. The more traditional D-Z pairs define the distance from the starting point and a corresponding elevation.

Geo-Referencing

Geo-referencing information provides the spatial (x-y) location of the cross section and included geometry. This information is inherent in the 3D coordinates, when extracting cross sections from a digital terrain model. However, if the cross section geometry is taken from a survey then the actual x-y-z coordinates of the points may not be known. In order to use the data within SMS for flood plain delineation, a proper geo-reference must be provided.

A cross section can have one of the following georeferencing definitions: All points specified (i.e. extracted cross sections will be of this type), Use two points (i.e. the coordinates of the beginning and ending location along the cross section defined), Use one point an angle (i.e. the centerline location is known and some angle relative to it defined), or no geo-referencing defined.

The geo-referencing is defined from the *Geo Ref* tab in the cross section editor.

Line Properties

Line properties define segments of material properties along the cross section. When using an area property coverage during extraction from a digital terrain model these properties are automatically marked and defined. However, they can also be established manually from within the *Line Props* tab in the cross section editor. To manually define the properties, use the "Insert Breakpoint" tool to specify the beginning and end locations on the cross section plot for each property. These locations and values can be edited in the Line Props spreadsheet.

Point Properties

Point properties include thalweg, left bank, and right bank (other properties can be defined but are not mapped/saved to hydraulic models from within WMS) locations. When using a centerline and bank line arcs from a 1D Hydraulic Centerline coverage during extraction these points are marked. You can also have WMS "Auto Mark" these points by looking for the lowest elevation (thalweg), and appropriate breaks in elevation/slope (banks). Point properties are edited from within the *Point Props* tab in the cross section editor.

Merging

It is possible to combine a surveyed cross section with a section extracted from a terrain model for the flood plain (i.e. the terrain model does not contain enough detail to define the cross section of the river) using the tools in the *Merge* tab in the cross section editor. Two different cross sections can be merged, with rules for locations and precedence defined in order to create a new cross section.

Filtering

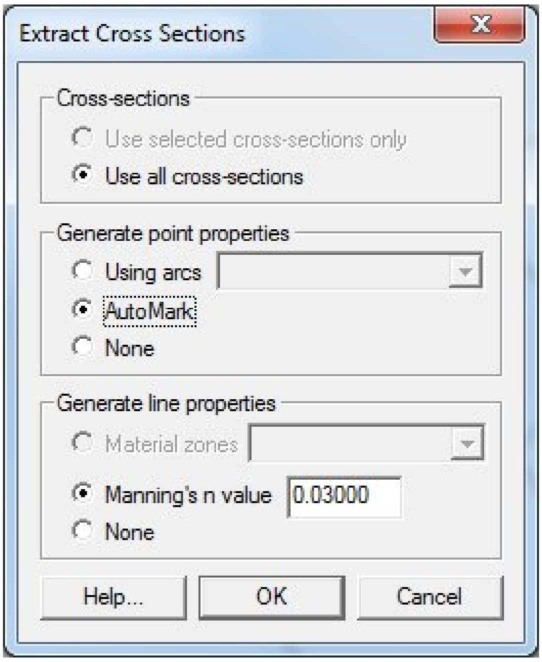
It may be that there are more points defining the cross section than are necessary (or that the hydraulic model is capable of processing). The *Filter* tab in the cross section editor allows you to specify rules for filtering "insignificant" points along the cross section. This can be particularly important when extracting cross sections from a dense digital terrain model.

Related Topics

- 1D River Module Tools

Extracting Cross Sections

The **Extract Cross-sections** command uses the cross section arcs and a digital terrain model (TINs are the only source that can currently be used) to extract the elevations at vertices of the feature arc cross-sections, or at the intersection points with the triangles.



Extract Cross Sections dialog

Cross-sections for individual arcs may be extracted by selecting the arc(s) before choosing the **Extract Cross-sections** command. If not cross-sections are selected then the *Use All Cross-sections* option is used.

Point properties (thalweg, left bank, right bank) can be defined from a 1D-Hydraulic Centerline coverage, or by AutoMark. The AutoMark option will examine the elevations of the extracted cross sections and try to infer the thalweg (low point) and the left and right bank points (change of slope) automatically.

Line properties can be determined from an area property coverage by intersecting the cross-section arcs with the area property polygons and marking them in the cross section database.

Cross Section Database

When extracting the cross sections you will be prompted for the name of a cross-section database file. SMS stores all of the cross-section information in a text database file. The cross section database can also be edited independently using the *Cross Section Editor* tools. Extracting cross sections with feature arcs is only way to generate cross-section information, they also can be imported from spreadsheet files (cut and paste), or entered manually.

Related Topics

- 1D River Module Tools

Interpolate Cross Sections

Using the **Interpolate Cross Sections** command in the *River Tools* menu you can create any number of additional cross sections between two selected cross sections. This allows you to provide more detail to the hydraulic model. The interpolated cross sections are derived from the geometries of the selected cross-sections. The stationing of the new cross sections are established by evenly distributing the length between the selected cross sections.

New Feature Arcs are created for the interpolated cross sections.

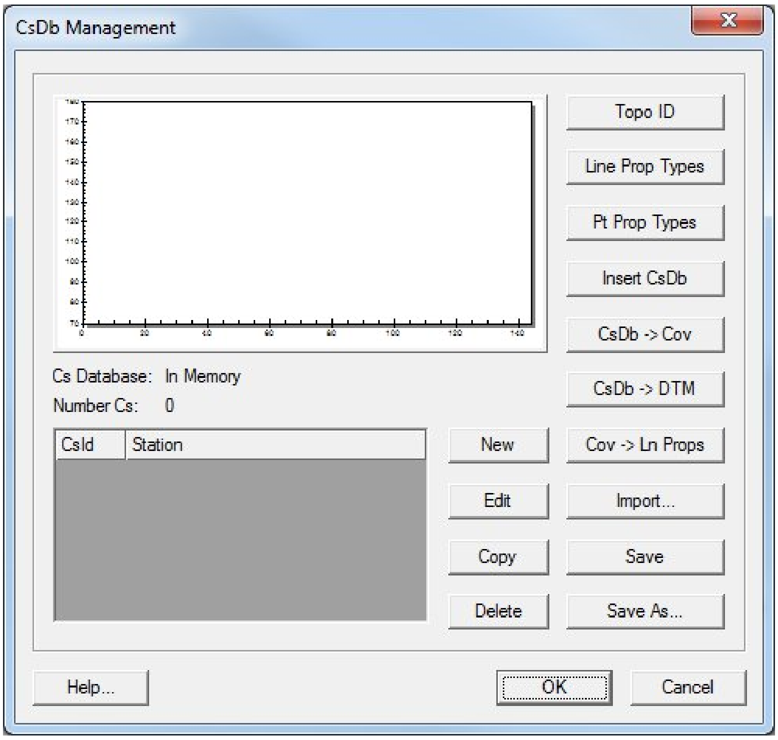
Related Topics

- 1D River Module Tools

Managing Cross Sections

For the new 1D Hydraulic Cross Section coverage, the cross section geometry is stored in text database file on disk. When extracting cross sections they are saved to a new (or existing) database file. This database was the basis for the development of the cross section data in the ArcHydro data model. Cross sections in the database can be used for the development of hydraulic models such as HEC-RAS.

The **Manage Cross Sections** command allows you to create a new database or open an existing database to add geometries, edit existing ones, and provide proper geo-referencing information. It is also possible to open a cross section database using the **Open** command from the *File* menu.



CsDb Management dialog

Cross Section Database Definition

When setting up a new database the following attributes can be defined:

- Topo ID – a topographic identifier and description that identifies where the cross section database was derived from. You should create a new Topo ID for each database.
- Line Prop Types – By default WMS uses only a Material ID, but other properties could be defined for general use (they will not immediately be used by supported hydraulic models).
- Point Prop Types – By default WMS uses thalweg, left bank, and right bank but other point properties could be defined for general use.

The cross section database management dialog also allows you to create a new cross section, edit/copy/delete an existing cross section, insert an entire database (merge databases together), convert a cross section database to a coverage (of course the georeferencing of cross sections must be provided for the cross section to be included in the coverage), create a digital terrain model from the cross section geometry, and converting the coverage to line properties.

Related Topics

- 1D River Module Tools

3.3. 3D Cartesian Grid Module

3D Cartesian Grid Module

This module is currently being developed for future interfaces.

Related Topics

- 3D Cartesian Grid Display Options
- SMS Modules

3D Cartesian Grid Display Options

The properties of the 3D Cartesian Grid data SMS displays on the screen can be controlled through the *Display Options* dialog. The entities associated with the 3D Cartesian Grid module with display options are shown below. Some of these entities also show an **Options** button to the right. For these entities, additional display controls are available. The available 3D Cartesian Grid display options include the following:

Model specific options

Each model may include model specific display options.

Related Topics

- 3D Cartesian Grid Module
- Display Options

3.4. Cartesian Grid Module

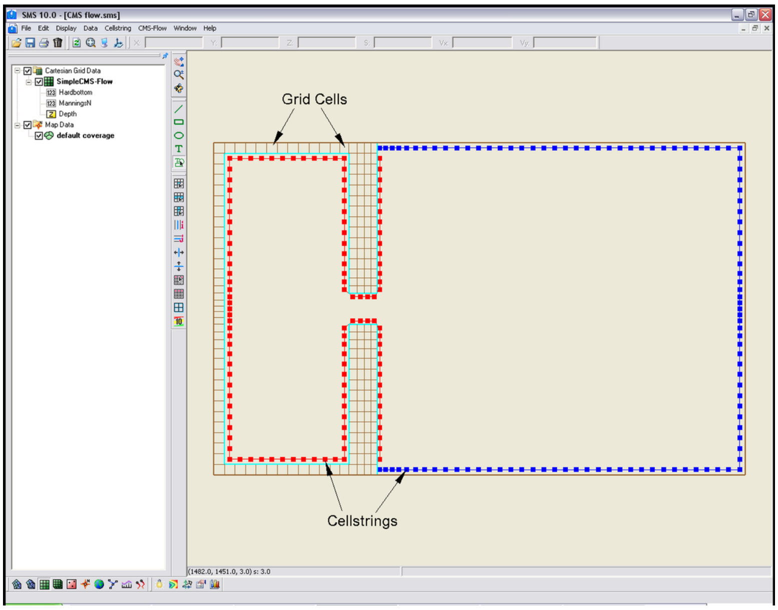
Cartesian Grid Module

At a glance

- Used to create, edit, and visualize rectilinear grids
- Datasets can have values at cells, corners, and midsides

The 2D Cartesian Grid Module contains tools used to construct 2D Cartesian finite difference grids. These grids consist of cells aligned with a rectilinear coordinate system.

Some models limit the grid to be defined with square cells, others limit to constant sized rectangular cells, while others add the flexibility of having variable sizes to the cells (variable row height or column width.



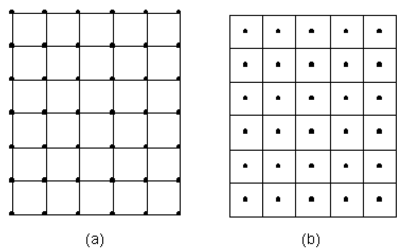
Cartesian Grid Example

It is strongly recommended that grids be created through the Map Module. The grid module currently includes interfaces for:

- BOUSS-2D – phase resolving Boussinesq wave energy and circulation model
- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone
- CMS-Wave – wave energy model
- STWAVE – wave energy model
- TUFLOW – Coastal, Riverine, and Urban hydrodynamic model with emphasis in flooding applications

Grid Types

Multiple types of grids are supported in the 2D Grid module. Generally, a grid can be classified as either a mesh-centered grid or a cell-centered grids. With a mesh-centered grid, the data values are stored at the corners of the grid cells. The BOUSS-2D model is an example of this type. With a cell-centered grid, data values are stored at the cell centers. Both STWAVE and CMS-Wave are example of this type. In additions, a model may generate data at the faces of the cells. In the most complicated cases, a model will use and/or output data in multiple locations. CMS-Flow is an example of this type.



Types of 2D Grids Supported by models SMS. (a) Mesh-Centered Grid (b) Cell-Centered Grid.

When a dataset is imported to a cell-centered grid, there is one value in the dataset for each cell.

In Cartesian grids, row and column boundaries are straight. Each cell center or grid node can have a unique elevation. The grid can also be rotated about the Z axis if desired.

Creating and Editing 2D Grids

Creating 2D Grids

The two main techniques used to create 2D grids are: the ***Create Grid*** command and the *Map → 2D Grid* command.

Create Grid

A new grid can be created by selecting the ***Create Grid*** tool from the *Cartesian Grid Tools*. With this tool active, the user can create a grid by clicking on three points in the graphics window. The first click defines the origin of the grid, the second click defines the orientation of the grid and length of the *I* axis and the third click defines the length of the length of the *J* axis. Once the user clicks three times defining the three points, the *Map → 2D Grid* dialog appears. The values include:

- **Grid Geometry –** This section allows the user to specify the origin, orientation and size of the grid. The fields of these quantities are populated with default values based on the three points. The orientation is measured as an angle from the positive X axis.
- **Cell Options –** This section allows the user to specify the number of cells in each direction in the grid. Several options are available. The user can specify sizes in the I (Delta U)and J (Delta V) directions or a number of columns and rows. If the *Use Grid Frame Size* toggle is checked, the grid will exactly match the dimensions specified in the *Grid Geometry* section. If that option is not checked, the last row and column may extend beyond the specified lengths. This allows the user to specify exact grid size, or exact cell size.
- **Depth Options –** The elevations or depths assigned to each cell or node can be specified as a single value, or the user may select a dataset to interpolate from.
- **Current –** For models that support currents, the user may also specify if current field either as a constant, or interpolated from a vector dataset.

The type and orientation of the grid generated is controlled by the current Cartesian Grid Model.

For some models, specific grid helps are available via a button at the bottom of the dialog.

Map → 2D Grid

The *Map → 2D Grid* command is used to construct a 2D grid using a grid frame feature object in a the current coverage. When the ***Map → 2D Grid*** command is selected, the dialog appears as described above.

If one or more refine points are defined in the conceptual model, the number of rows and columns in the grid will be automatically determined when the grid is created. Thus, these fields cannot be edited by the user and will be dimmed. If refine points are not defined, the user must enter the number of rows and columns.

Editing 2D Grids

Each of the cells in a 2D grid can be active (water) or inactive (land). An inactive cell is ignored when contours or vectors are displayed on the grid and by the numeric engine during computation. If a cell has the potential of becoming active (due to wetting/drying or a similar process), it should be classified as active. Cells status is specified by selecting the cell and assigning a status through the model menu.

Rows and columns can be added to an existing grid that supports variable row/column size by using the **Insert Row**, **Insert Coumn**, **Drag Row**, or **Drag Coumn** tool. (See 2D Grid Tool Palette)

Smoothing 2D Grids

It may be useful to smooth the spatial data stored on a 2D grid for a number of reasons. These reasons include:

- In order to conserve the amount of disk spaced required to store a DEM, many DEM formats store elevations rounded to the nearest integer value. This causes elevation changes to occur in discrete steps rather than smoothly, as would be the case in nature. In regions of low relief, rounded elevations can cause an area to be artificially "flat."
- Surveys may include anomalies. Smoothing algorithms blend these bad data points into the surrounding values.
- Datasets may include spurious noise either from physical conditions such as waves or numerical filtering. Smoothing can dampen these variations.

When you right click on the grid in the *Project Explorer*, operations for the grid appear in a pop up window. One of these is the smooth operation.

Converting 2D Grids

2D Grids may be converted to other types of data used in SMS, such as a Scattered Dataset of 2D mesh. 2D Grids can be converted by right clicking on the grid in the *Project Explorer*.

Cartesian Grid Module Tools

See Cartesian Grid Module Tools for more information.

Cartesian Grid Module Menus

See Cartesian Grid Module Menus for more information.

Related Topics

- Cartesian Grid Display Options
- Spectral Energy
- Cartesian Grid Find Cell
- SMS Modules

Cartesian Grid Module Overview

Cartesian grid module

Overview

The 2D Cartesian Grid Module contains tools used to construct 2D Cartesian finite difference grids. These grids consist of cells aligned with a rectilinear coordinate system.

Some models limit the grid to be defined with square cells, others limit to constant sized rectangular cells, while others add the flexibility of having variable sizes to the cells (variable row height or column width. It is used to create, edit, and visualize rectilinear grids. Datasets can have values at cells, corners, and midsides.

Functionalities

The 2D Cartesian Grid Module contains tools used to construct 2D Cartesian finite difference grids. These grids consist of cells aligned with a rectilinear coordinate system.

Some models limit the grid to be defined with square cells, others limit to constant sized rectangular cells, while others add the flexibility of having variable sizes to the cells (variable row height or column width.

It is strongly recommended that grids be created through the Map Module. The grid module currently includes interfaces for:

- BOUSS-2D – phase resolving Boussinesq wave energy and circulation model
- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone
- CMS-Wave – wave energy model
- STWAVE – wave energy model
- TUFLOW – Coastal, Riverine, and Urban hydrodynamic model with emphasis in flooding applications

Grid Types

Types of 2D Grids Supported by models SMS. (a) Mesh-Centered Grid (b) Cell-Centered Grid. In Cartesian grids, row and column boundaries are straight. Each cell center or grid node can have a unique elevation. The grid can also be rotated about the Z axis if desired.

Creating and Editing 2D Grids

Create Grid. A new grid can be created by selecting the **Create Grid** tool from the Cartesian Grid Tools. With this tool active, the user can create a grid by clicking on three points in the graphics window. The first click defines the origin of the grid, the second click defines the orientation of the grid and length of the I axis and the third click defines the length of the length of the J axis. Once the user clicks three times defining the three points, the *Map → 2D* dialog appears.

Editing 2D Grids. Each of the cells in a 2D grid can be active (water) or inactive (land). An inactive cell is ignored when contours or vectors are displayed on the grid and by the numeric engine during computation. If a cell has the potential of becoming active (due to wetting/drying or a similar process), it should be classified as active. Cells status is specified by selecting the cell and assigning a status through the model menu.

Rows and columns can be added to an existing grid that supports variable row/column size by using the **Insert Row**, **Insert Column**, **Drag Row**, or **Drag Column** tool. (See 2D Grid Tool Palette)

Smoothing 2D Grids

It may be useful to smooth the spatial data stored on a 2D grid for a number of reasons. These reasons include:

- In order to conserve the amount of disk spaced required to store a DEM, many DEM formats store elevations rounded to the nearest integer value. This causes elevation changes to occur in discrete steps rather than smoothly, as would be the case in nature. In regions of low relief, rounded elevations can cause an area to be artificially "flat."
- Surveys may include anomalies. Smoothing algorithms blend these bad data points into the surrounding values.
- Datasets may include spurious noise either from physical conditions such as waves or numerical filtering. Smoothing can dampen these variations.

When you right-click on the grid in the Project Explorer, operations for the grid appear in a pop up window. One of these is the smooth operation.

Converting 2D Grids

2D Grids may be converted to other types of data used in SMS, such as a Scattered dataset of 2D mesh. 2D Grids can be converted by right-clicking on the grid in the Project Explorer.

Project Explorer

The following Project Explorer mouse right-click menus are available when the mouse right-click is performed on a Cartesian Grid Module item.

Cartesian Grid Module Root Folder Right-Click Menus

Right-clicking on the Cartesian Grid module root folder in the project explorer invokes an options menu with the following options:

- Display Options

Cartesian Grid Item Right-Click Menus

Right-clicking on a Cartesian Grid item in the Project Explorer invokes an options menu with the following module specific options:

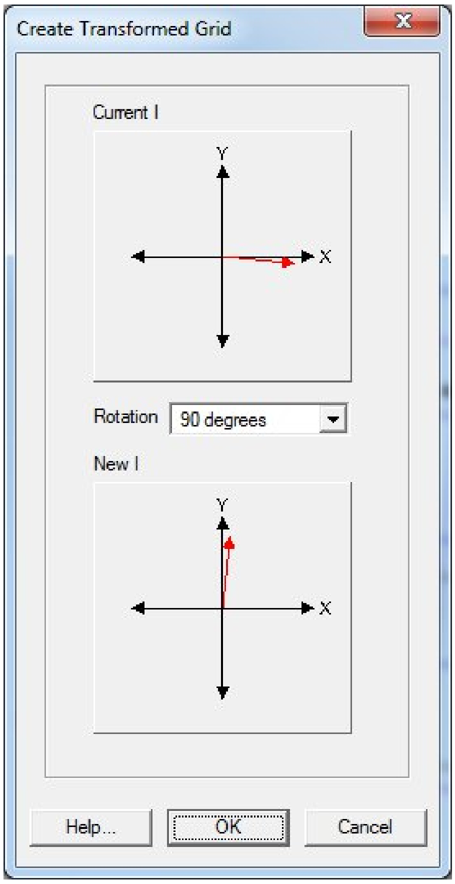
- Smooth – Opens the Cartesian Grid Smoothing Options Dialog.

Model Specific Right-Click Menus

- **Create Transformed Grid**

  Opens the *Create Transformed Grid* dialog. Creates a copy of the grid with a rotated origin. Used to change the I direction for wave models.

  Related Models: CMS-Wave, STWAVE



Create Transformed Grid dialog

Menus

The Cartesian Grid Module *Data* Menu commands include:

Model Specific Menus

- BOUSS-2D
- CMS-Flow
- CMS-Wave
- STWAVE
- TUFLOW

Model Commands

- Steering Module
- Switch Current Model

Dataset Commands

- Data Calculator
- Dataset Toolbox
- Create Datasets
- Map Elevation
- Zonal Classification

Visualization Commands / Options

- Contour Options
- Vector Options
- Film Loop

Data Conversion Commands

- Grid → Scatterpoint
- Grid → Map
- Grid → Mesh

Grid Commands

- Find Cell

Module Tools

When active, the Cartesian Grid module has several tools located in the tool palette. See Cartesian Grid Tools to learn more.

How do I?

To learn more about how to use the Cartesian Grid Module go to the Tutorials section of the Aquaveo website at: http://www.aquaveo.com/software/sms-learning-tutorials [1].

Related Coverages

The grid module currently includes interfaces for:

- BOUSS-2D – phase resolving Boussinesq wave energy and circulation model
- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone
- CMS-Wave – wave energy model
- STWAVE – wave energy model
- TUFLOW – Coastal, Riverine, and Urban hydrodynamic model with emphasis in flooding applications

Cartesian Grid Coordinates

A projection can now be associated with a Cartesian grid. The data for the grid will be stored in this projection; however, the grid can still be displayed in any projection chosen by the user. When the SMS project's projection ("working projection") is changed, the grid will be converted "on the fly." While the display will be changed, the data will remain in the original projection. This method will reduce rounding errors in the data introduced when converting coordinates.

Editing the Grid

When the grid is displayed in a projection different than its own, it will not be editable. The "working projection" must match that of the grid to be able to edit. The grid's right-click command, **Work in grid projection**, will set the "working projection" to the grid's projection.

Changing the Grid Projection

When a grid is created, the projection is defaulted to the "working projection". The grid's projection can be changed using the **Projection...** and **Reproject...** commands in the grid's right-click menu.

Floating Projection

If a grid is read in from a file that does not specify a projection, the grid will "float" in whichever projection is the working projection. If a grid is floating, the **Projection...** command in the right-click menu will be followed by "floating". To assign a projection to the grid, select the **Projection...** command and select a projection.

Related Topics

- SMS:Cartesian Grid Module

Cartesian Grid Data Menu

Most of the SMS modules have a *Data* Menu, but the items in this menu are different for each module. The *Cartesian Grid Module Data* Menu commands include:

Model Commands

- Steering Module – Launches the steering tool.
- Switch Current Model – Allows user to change current active model.

Dataset Commands

- Data Calculator – Can be used to perform mathematical operations with datasets.
- Dataset Toolbox – Contains tools for working with datasets.
- Create Datasets – Opens a dialog that can be used to create functions for the entire mesh or active scatter set.
- Map Elevation – Allows use of another functional dataset as the mapped elevation function.
- Zonal Classification – Tool to identify areas that meet as set of criteria.

Visualization Commands / Options

- Contour Options – opens a dialog where dataset specific contour options can be defined.
- Vector Options – opens a dialog where options to generate vectors can be edited.
- Film loop – opens the *Film Loop Setup* wizard.

Data Conversion Commands

Grid to Scatterpoint

The **Grid → Scatter** command in the *Data* menu (Cartesian grid module) is equivalent to the **2D Grid → 2D Scatter** command in the right-click menu on a grid object in the project explorer. It is used to convert the grid cell corners into a scattered dataset (scatter module).

Each cell corner in the grid is converted to a scatter vertex. SMS computes one dataset from the elevation dataset of the grid and one dataset for each functional dataset on the grid. If the Cartesian grid is cell centered (data at the center of the cell), SMS averages the values of the four surrounding cells to compute a value for the scatter set vertex. If the Cartesian grid is a mesh centered grid (BOUSS2D), SMS assigns the value from the cell corner to the scattered vertex.

Each cell in the grid is converted to two triangles in a TIN.

This command allows the visualization of the data on a matching geometric object.

Grid to Map

The **Grid → Map** command in the *Data* menu (Cartesian grid module) is equivalent to the **2D Grid → Map** command in the right-click menu on a grid object in the project explorer. It is used to convert attributes of the Cartesian grid into feature objects on a coverage (map module). All generated features are added to the current or active coverage. If a new coverage is desired, it should be created prior to issuing this command.

The command includes the following options:

Land/Water Boundary → Arcs

This option only applies to Cartesian grids which support cell attributes supporting land and water cells (CMS-Flow). When this option is selected, the cell faces between cells of these opposing types are converted to feature arcs. For grid cells that do not support cell attributes, this options functions identically to the *Grid Boundary → Arcs* option.

Grid Boundary → Arcs

This option generates a feature arc along all cell edges of the boundary of the Cartesian grid.

Observation Cells → Points

This option can be applied to either arc generation option above. It only applies to Cartesian grids that support the cell attribute of observation cells. If this toggle is selected, a feature point is created at the centroid of each cell with the observation point attribute.

This command has minimal usefulness since the feature objects generated are step functions. It is recommended that other data sources, such as a scatter set or a shapefile be used to define these features, but if only the numerical model exists, this command can be used to help construct a conceptual model.

Grid to Mesh

The **Grid → Mesh** command in the *Data* menu (Cartesian grid module) is equivalent to the **2D Grid → 2D Mesh** command in the right-click menu on a grid object in the project explorer. It is used to convert the grid cell corners into a mesh or unstructured grid object (mesh module).

Each cell corner in the grid is converted to a mesh node. SMS computes an elevation for the mesh node as the average of the four surrounding cells in the grid for cell centered grids, or the elevation of the corner for mesh centered grids.

Each cell in the grid is converted to two triangular elements.

This command allows the visualization of the data on a matching geometric object.

Grid Commands

Find Cell

The **Find Cell** command from the *Data* menu is used to locate a cell either with a specific i,j location, or near a specific location. When this command is executed the *Find Cell* dialog opens.

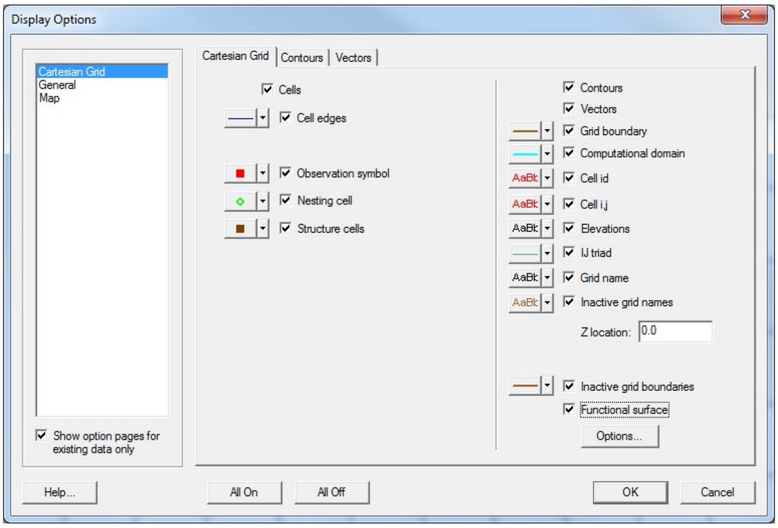
When the *Find by* (I, J)option is selected, the cell with the specified $i, j$is highlighted in red. If there is no cell with the specified i, j, an error message is given. Conversely, when the *Find by nearest* (x, y)*coordinates* option is selected, the cell containing the specified coordinate is highlighted with red. If no cell contains the x, ylocation, an error message is given. With either of these methods, the found cell becomes selected in addition to being highlighted.

Related Topics

- Cartesian Grid Module

Cartesian Grid Module Display Options

The properties of all mesh data that SMS displays on the screen can be controlled through the *Cartesian Grid* tab of the *Display Options* dialog. This dialog is opened by right-clicking on the Cartesian Grid Data entry in the *Project Explorer* and selecting the **Display Options** command. (It can also be accessed from the from the *Display* menu or the *display options* Macro.)



Display Options dialog showing *Cartesian Grid* options

Cell display

SMS allows the user to specify the level of quality for displaying cells. This is done at the top of the *display options* dialog by choosing 1 quad, 4 quad, 4 triangles or 8 triangles as the display option. Aquaveo developers have found that 1 quad is the fastest display option, but in situations of high relief, the cell may be distorted from a flat quad and this representation can leave gaps in the display. For such situations, 4 quads generally solves the display issues, but takes a little longer. For highest quality of display, and to assure matching of the cell display with the contours, 8 triangles can be used.

The cells are colored based on their types. Land cells are colored separately from water cells. Special cells are marked with symbols. These could be observation stations (probes), or specially marked cells for another purpose.

Cell String display

Some models support cell strings (M2D, BOUSS2D), at which specific attributes or boundary conditions are specified. Each cell string type has its own display attributes.

Grid display

Other entities associated with the Cartesian Grid module with display options are shown below. Some of these entities also show an **Options** button to the right. For these entities, additional display controls are available. The available mesh display options include the following:

- **Contours** – The contours are drawn for the active scalar dataset. All standard contour display options are supported for cartesian grid contours.
- **Vectors** – The cartesian grid vectors are drawn for the active vector dataset. All standard vector display options are supported.
- **Grid Boundary** – A line around the perimeter of the cartesian grid can be drawn. This is useful when the cells are turned off. User specifies line color and thickness.

- **Computational Domain** – It can be useful to delineate the active (or water cells) from the rest of the grid. This option allows for specification of a line to outline the computational cells.
- **Cell id**
- **Cell i, j** – The i,j coordinate can be drawn in each cell.
- **Elevations** – The current scalar value can be drawn in each cell.
- **IJ triad** – Arrows can be displayed at the origin of the grid showing the i and j directions.
- **Display Inactive Grids** – SMS supports the ability to store/manipulate multiple Cartesian grids at the same time. This allows for functionality such as nested grids in STWAVE and steering between STWAVE and M2D. However, only one grid can be edited at a time. This is the "active" grid. The outline of other grids can be displayed using the *Display Inactive Grid* option.
- **Functional Data Surface**

Model specific options

Each model may include other display options associated only with that model or options slightly modified from those described above. For example, the types of cell strings supported by each model are different. This is reflected by slightly different options for each model. Details on these entities can be found in model specific documentation.

Related Topics

- Cartesian Grid Module
- Display Options

Cartesian Grid Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Cartesian Grid module is active. Tools specific to a model interface are described with the corresponding model. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Cartesian Grid module tool palette. Depending on the current model, and the type of grids it supports, some of these tools may not be available.

Tool	Tool Name	Description	Right-Click Menu
	Select Cell	The **Select Cell** tool is used to select a grid cell. A single cell is selected by clicking on it. A second cell can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple cells can be selected at once by dragging a box around them. A selected cell can be de-selected by holding the *SHIFT* key as it is clicked. When a single cell is selected, its Z coordinate is shown in the *Edit Window*. The Z coordinates can be changed by typing in the edit field, which updates the depth function. If multiple cells are selected, the Z Coordinate field in the *Edit Window* shows the average depth of all selected cells. If this value is changed, the new value will be assigned to all selected points. With one cell selected, the *Edit Window* shows the point i,j location. With multiple cells selected, the *Edit Window* shows the number of selected cells. The number and size of the cells can be changed in the *Model Control*.	When one or more Cartesian grid cells are selected, and there is a scatter set in the project, a right-click in the graphic window will bring up a menu. One of the options is **Interpolate Bathymetry...**. This option brings up the *Interpolation* dialog where the desired source scatter dataset can be selected. When **OK** is clicked, SMS will interpolate the selected cell(s)' elevations based on the chosen scatter dataset.

	Select Row	The **Select Row** tool is used to select cell rows. Rows are selected in the same manner as selecting individual cells.	N/A
	Select Column	The **Select Column** tool is used to select cell columns. Columns are selected in the same manner as selecting individual cells.	N/A
	Split Grid Column	Inserts a new column into an existing grid. This tool splits an existing column into two columns at the location selected by the user.	N/A
	Split Grid Row	Inserts a new row into an existing grid. This tool splits an existing row into two rows at the location selected by the user.	N/A
	Drag Column Boundary	Edit column boundary. This tool makes one column narrower while making its neighbor wider.	N/A
	Drag Row Boundary	Edit row boundary. This tool makes one row taller while making its neighbor shorter.	N/A
	Select Cell String	Select a “Cell String”. Allows assignment of boundary conditions.	N/A
	Create Cell String	Create a “Cell String”. This tools allows the user to define a string of cells for later assignment of boundary conditions or flux observations.	N/A

Related Topics

- SMS:Cartesian Grid Module

Grid Frame Properties

The *Grid Frame Properties* dialog is accessed when the **Create 1D Grid Frame** tool or the **Create 2D Grid Frame** tool is used. It can also be accessed after a grid frame has been created by using **Select 1D Grid Frame** tool or the **Select 2D Grid Frame** tool then right-clicking on the grid and selecting the **Properties** command from the right-click menu.

The *Grid Frame Properties* dialog allows the user to specify the attributes applied to the grid frame when performing a **Map → 2D Grid** operation. These properties are as follows:

- Origin
- Orientation
- Directional properties (u and v direction)
    - Define cell sizes – specified uniform cell sizes
        - Cell size – the cell size in the specified direction
        - Number of cells – number of cells in the specified direction
    - Use refine points – refine points will be used to generate the grid
        - Maximum cell size – the max size the should exists when growing
        - Maximum bias – the max growth ratio to be used when growing
        - Use inner growth – specifies whether the cell sizes should grow between two refine points
    - Grid size – the grid dimension in the specified direction

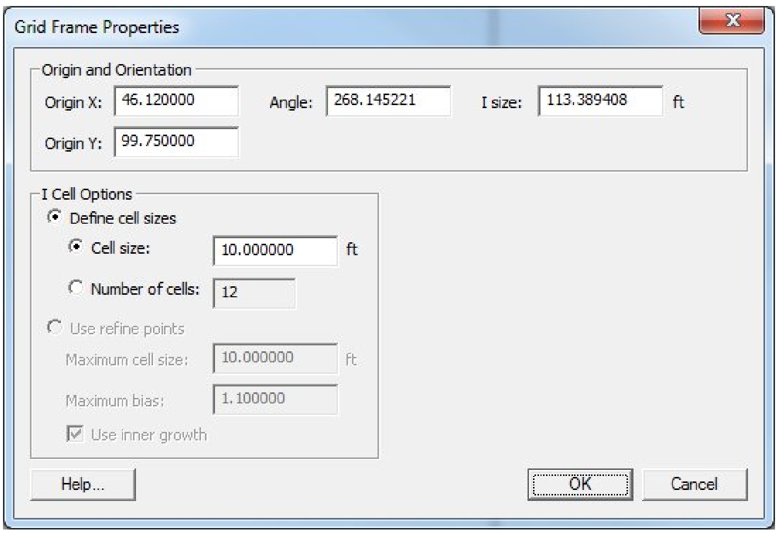
When the user specifies "Define cell sizes", there are a few options available. These options are:

1. Specify cell size – specify the cell size and the number of cells will be computed.
2. Specify number of cells – specify the number of cells and the cell size will be computed.

If the grid is to have square cells, the v direction cell size will always be linked to the u direction cell size.

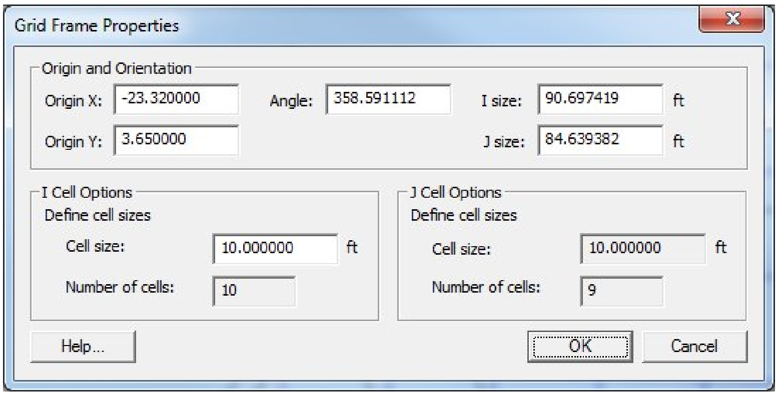
The *Grid Frame Properties* dialog varies depending on the active coverage.

1D Grid Frame Properties

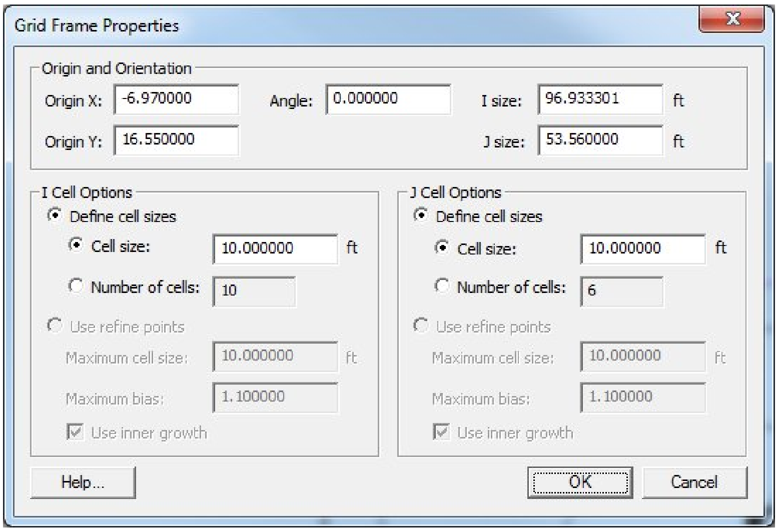


GenCade *Grid Frame Properties* dialog

2D Grid Frame Properties



TUFLOW *Grid Frame Properties* dialog



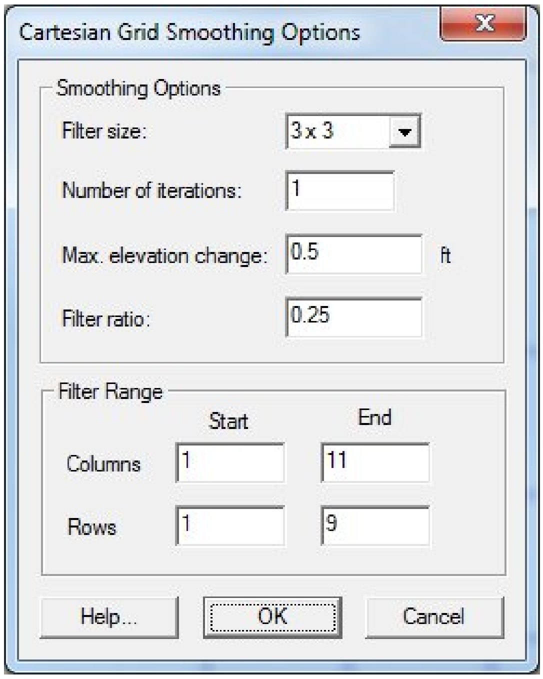
Grid Frame Properties dialog with CMS-Flow coverage

Related Topics

- Grid Frame Dialog
- Map Module Tools

Grid Smoothing

The *Cartesian Grid Smoothing Options* dialog is opened by right-clicking on a Cartesian Grid item in the *Project Explorer* snd selecting the **Smooth...** menu command. The following options can be specified:



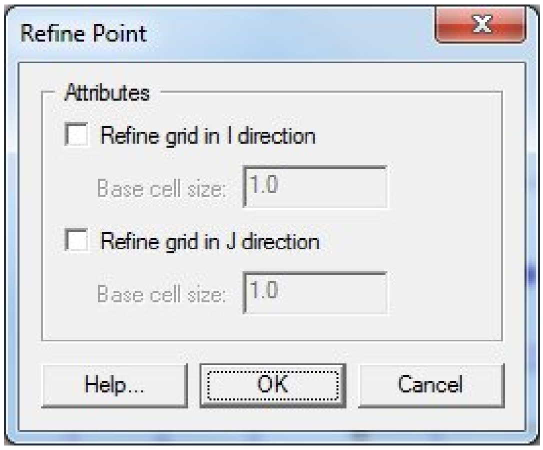
Cartesian Grid Smoothing Options dialog

- **Filter size –** This determines how many neighbors are included when smoothing the grid. Options are 3x3 and 5x5.
- **Number of iterations –** This specifies how many passes should be made with the smoothing algorithm.
- **Max. elevation change –** This value specifies the maximum allowable elevation change per iteration for each cell.
- **Filter ratio –** The new cell elevation is computed using the original elevation (at the beginning of the iteration not the whole process) and the "blurred" elevation. The filter ratio defines how far the elevation is changed between the original elevation and the "blurred" elevation. A filter ratio of 1.0 would replace the existing elevation with the "blurred" elevation. A filter ratio of 0.0 would be pointless as it wouldn't change the elevations. A filter ratio of 0.5 would give a new elevation that is the average of the original elevation and the blurred elevation.
- **Filter Range –** The start and end index values specify the extents of the smoothing. Defaults to the grid extents.
  - **Column start –** The column index on which to start the smoothing process.
  - **Column end –** The column index on which to end the smoothing process.
  - **Row start –** The row index on which to start the smoothing process.
  - **Row end –** The row index on which to end the smoothing process.
  - **Only modify selected cells/cell locations –** If this option is selected, only the cells or cell locations (if model uses elevations at centers, faces, and corners) that are selected are smoothed. Cells or cell locations not selected may be used to compute "blurred" elevations but their elevations are never modified.

Related Topics

- Cartesian Grid Module Right-Click Menus
- Cartesian Grid Module

Refine Point Dialog

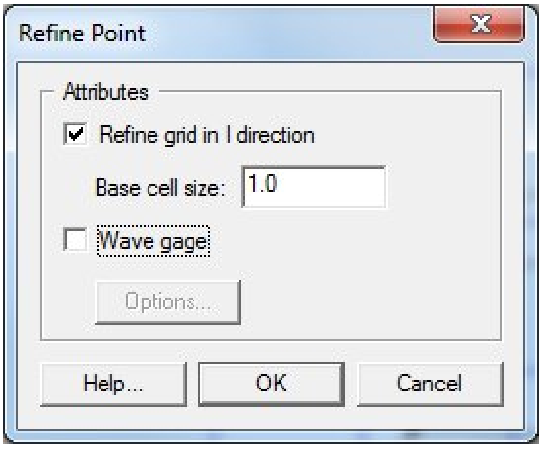


Refine Point dialog

The *Refine Point* dialog is used to set the attributes for a refine point represented by a feature point in a Cartesian Grid model coverage. Attributes that can be specified for each refine point include:

- Refine grid in I direction(checked = on)
  - Base cell size – Specify the cell I size in the vicinity of the refine point.
  - Refine grid in J direction(checked = on)
    - Base cell size – Specify the cell J size in the vicinity of the refine point.
    - Wave Gage
      - Options –

Only refine points located within a grid frame are used when the **Map → 2D Grid** command is executed. Refine points are not available for all models, since some Cartesian Grid models require uniform cell sizes.



Refine Point dialog

Related Topics

- Feature Objects Menu

3.5. Curvilinear Grid Module

Curvilinear Grid Module

The curvilinear grid module contains tools used to work with curvilinear grid data. Curvilinear grids consist of nodes that are grouped together to form cells. These nodes and cells define the computational domain of the numerical model. In addition to nodes and cells, a curvilinear grid may store additional information such as material values assigned to elements and boundary conditions assigned to nodes. In general, this additional information is used as input data for the numerical model.

Nodes

Nodes are the basic building blocks of cells in a curvilinear grid. Nodes store elevation and other dataset values. Nodes can also be used for building nodestrings and assigning boundary conditions. The density of nodes helps determine the quality of solution data and can be important to model stability.

Cells

Cells are used to describe the area to be modeled. Cells are formed by joining exactly four nodes. No more than four cells may join at a single node. If four cells join at a single node, the node cannot be a boundary node. Cells are identified by a unique i, j index.

Delete Cells

1. Click on the **Select Element** tool for curvilinear grids or Vtk Meshes.
2. Select the cells to be deleted.
3. Right-click on the selected cell and select *Delete* or hit the *DELETE* key.
4. For curvilinear, the Delete will fail and error message will be displayed if the grid would become invalid if deletion occurred.

Add Cells

Cells can only be added to an existing Curvilinear grid.

1. Click on the **Create Element** tool for cuvilinear grids.
2. Click and hold on any element on the edge of the grid and drag the displayed arrow across the boundary edge, then release. A new element will be created.

Nodestrings

A collection of nodes can be formed into a nodestring. Nodestrings are most commonly used to assign boundary conditions such as a flowrate or water-surface elevation. Nodestrings can also be used for mesh renumbering, forcing break lines, and boundary smoothing. Finally, a nodestring can store attributes pertinent to a location such as the total flow nodestring.

Delete Nodes (Vtk Mesh only)

1. Click on the **Select Node** tool for Vtk mesh.
2. Select the nodes to be deleted.
3. Right-click on the selected nodes and select *Delete* or hit the *DELETE* key.

Merging Two Curvilinear Grids

To merge two curvilinear grids you must have at least one segment that is common (shared) between the two grids.

To do a merge:

- Hold down the *CTRL* key and select two curvilinear grids from the tree item.
- Right click and select **Merge Curvilinear Grids**.

A new curvilinear grid is created from the two selected grids.

Curvilinear Grid Models

The curvilinear grid module currently includes interfaces for:

- LTFATE

Curvilinear Grid Module Tools

See Curvilinear Grid Module Tools for more information.

Curvilinear Grid Module Menus

When one or more active nodestrings have been created, and the **Select Nodestrings** tool is selected, a set of menu's becomes available by right-clicking on the mouse. The menu items operate on the active nodestrings:

- **Delete Selected –** deletes the selected nodestrings.
- **Merge Selected –** will merge two or more selected nodestrings to form a single nodestring. Nodestrings must share the same endpoints to be merged.
- **Clear Selections –** unselect all of the selected nodestrings.
- **Select All –** selects all nodestrings.

Curvilinear Grid Display Options

See Curvilinear Grid Display Options for more information.

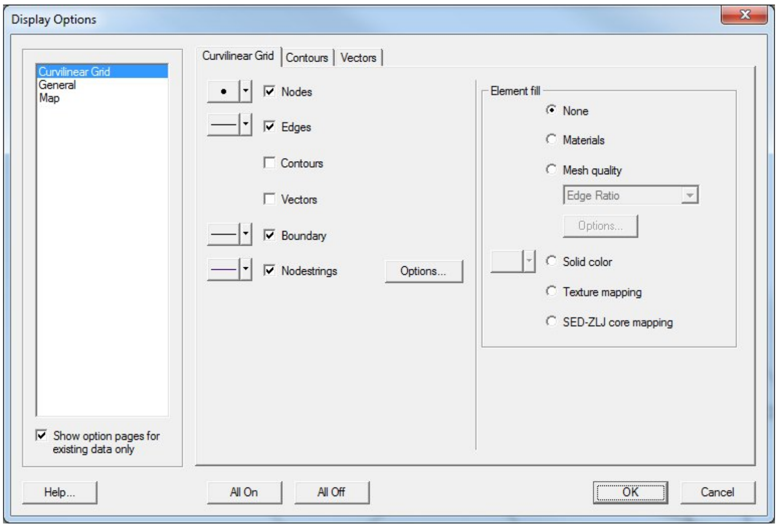
Related Topics

- Curvilinear Grid Display Options
- SMS Modules

Curvilinear Grid Display Options

Curvilinear Grid Module Display Options

The properties of the curvilinear grid data SMS displays on the screen can be controlled through the *Display Options* dialog. The entities associated with the curvilinear grid module with display options are shown below. Some of these entities also have an associated *Options* button. For these entities, additional display options are available. The available curvilinear grid display options include the following:



Display Options dialog showing *Curvilinear Grid* options

- **Nodes** – A circle is filled around each node. The user can specify the radius and color of these circles. The **Options** button is used to set the display of nodal boundary condition data. The dialog that opens when this button is clicked depends on the current numerical model.
- **Edges** – Cell edges.
- **Contours** – Contours are drawn for the active scalar dataset. Use the contours tab to change contour options.
- **Boundary** – A line is drawn around the perimeter of the curvilinear grid.
- **Element Fill** – Elements can be filled using the following options:
  - **None**
  - **Materials** – Elements are filled using the material assigned to the element.
  - **Mesh quality** – Elements are filled using a user specified mesh quality metric. For a description of the mesh quality metrics, please see the VERDICT Manual [1] which contains the mathematical definition of each quality metric. The VERDICT website contains further information on the VERDICT [2] mesh verification code library.
  - **Solid color** – Elements are filled using a solid color.
  - **Texture mapping** – An image is draped over the mesh elements.

Model Specific Options

Each model may include model specific display options. These appear at the bottom of the *display options* dialog.

Related Topics

- Curvilinear Grid Module
- Display Options

Curvilinear Grid Module Tools

The following tools are active in the dynamic portion of the *Tool Palette* whenever the Curvilinear Grid Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the *Graphics Window* with the cursor depends on the current tool. The table below describes the tools in the Curvilinear Grid tool palette.

Tool	Tool Name	Description	Right Click Menu
	Select Curvilinear Grid Node	The **Select Curvilinear Grid Node** tool is used to manually select and edit an individual node location. Currently, the only reason to select a grid node is to adjust the shape of the adjacent (4) elements. The status bar at the bottom of the screen displays the i, j, and id of the selected node. The current position of the node is displayed in the edit fields at the top of the screen. (Dragging of curvilinear grid nodes was added for SMS 11.2.)	N/A
	**Select Cell**	The **Select Cell** tool is used to select a cell (or element) of a curvilinear grid. Currently, the only reason to select a cell is to delete it from the grid and create a hole (or notch) in the grid. This would be done when the cell covers an island or other region that should be excluded from the computational domain. The status bar at the bottom of the screen displays the i, j, and id of the selected cell as well as the area of the cell. (Deleting of curvilinear grid cells was added for SMS 11.2.)	
	Select Nodestrings	The **Select Nodestrings** tool is used to select nodestrings. Nodestrings currently serve no purpose for curvilinear grids.	
	Create Nodestrings	The **Create Nodestrings** tool is used to create node string. Nodestrings currently serve no purpose for curvilinear grids.	N/A

Related Topics

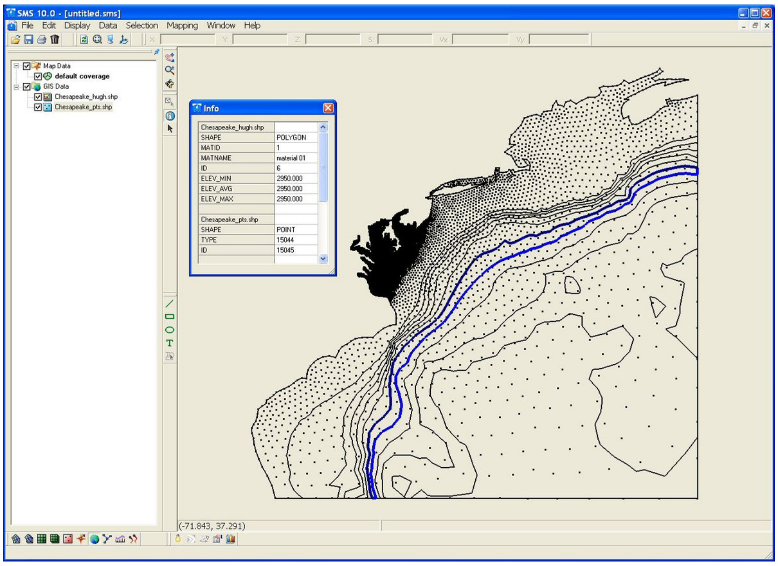
- Curvilinear Grid Module

3.6. GIS Module

GIS Module

At a glance

- Open and visualize GIS data
- Supports ESRI and MapInfo formats
- Uses Mapobjects for ESRI files if available to use ArcGIS visualization options
- GIS data can be converted to feature data (map module)



GIS Data in SMS

The GIS module has been separated from the Map module in order to define a more integrated and separate approach to linking with GIS data. The GIS module has two separate modes, although the primary functions are available in either mode. The main reason that the GIS data has been separated from the map module is to allow users to handle large files more efficiently when creating hydrologic models. For example the GIS module allows you to import large files and then select and convert to feature objects only the portions that are needed. The conversion to feature objects is both time consuming and potentially memory intensive, and so managing your GIS data in this fashion is more efficient (the equivalent GIS functionality is to clip out just the parts of the data you need for import).

Some of the key functionality available in either mode includes:

- Efficient management of large datasets
- Graphical selection of features
- Mapping of selected features to feature objects in map coverages
- Viewing attribute tables
- Joining additional attribute tables based on a key field (i.e. joining the hydrologic soils group attribute to a STATSGO/SSURGO shapefile).

Using the GIS Module with a License of ArcView®

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

To use SMS with ArcView®, do the following:

- Activate the GIS Module.
- Enable ArcObjects by selecting Data | Enable ArcObjects.
- Open the desired shapefile by selecting Data | Add Shapefile Data... and browse for the file. The file should now appear in the Project Explorer.
- Right-click on the *imported shapefile* and select *Properties*. The *ArcGIS Properties* window will appear.
- Click on the *Symbology* tab and the shapefile properties can be edited.

Using the GIS Module without a License of ArcView®

Most of the same functionality that exists for users with licenses of ArcView® is available to users without. The primary differences are that you can only read layers that are in the shapefile format, and all of the display and symbology available with ArcView® is not available. Points, lines, and polygons are displayed in a single color and not filled. Further some of the queries for selection are not supported without a license to ArcView®.

Importing Shapefiles

Shapefiles can be visualized in SMS as well as be converted to feature objects or scatter data. This can be done by using either the **Shapes → Feature Objects** or **Polygons → TIN** command in the *Mapping* menu. It is important to check for bad polygons when converting shapefile data. These may be polygons with zero area or with duplicate nodes. This problem can be fixed by using the **Clean** command in the *Feature Objects* menu. If using the **Clean** option does not fix the problem initially, try increasing the tolerance until all problematic feature objects are removed.

For additional information, see Importing Shapefiles.

GIS Module Tools

See GIS Module Tools for more information.

GIS Module Menus

See GIS Module Menus for more information.

Related Topics

- Map Module
- ArcObjects
- Shapefiles
- Modules

Importing Shapefiles

ARC/INFO or ArcView® shape files provide an easy method to import GIS data into SMS. Unfortunately the shape file format is extremely redundant, meaning that points or lines that are shared by lines or polygons are multiply defined.

Therefore, in order for you to convert a shape file to a SMS coverage it may take up to several minutes (depending on size) to build the correct line or polygon topology. This was very problematic in previous versions because SMS often bogged down when reading moderately large files. This is one of the primary reasons that the new GIS module has been developed and with or without a license to ArcObjects shapefile data can now be managed better by SMS.

With the addition of the GIS module there are now two different ways to import shapefile data.

Direct Conversion of Shapefile Data to Coverages

The first is the traditional method which allows you to load a shapefile layer directly into a coverage.

You can then map attribute fields of the shapefiles database (*.dbf) file to their pertinent SMS parameters .

Using the GIS Module to Convert Shapefile Data to Coverages

When opening a shapefile in the GIS module using the **Add Shapefile Data** or **Add Data** commands SMS first reads the points/lines/polygons into a simple display list and does not try to "build" topology by connecting arcs at nodes, and eliminating shared edges of polygons as required when creating a coverage. This makes the display and selection of the polygons much easier and more efficient. You can then select only the polygons you wish to convert to a coverage and map them. In this way you will only be building topology for the selected polygons.

Cleaning Imported Shapefile Data

If you intend to use the data from the shape file in more than one session you should save it as a SMS map file after importing/mapping the first time. Further, after importing the shape files you may wish to consider the following:

1. Clean the feature objects in order to snap nodes within a certain distance, intersect arcs, and eliminate dangling arcs.
2. Build Polygon so that SMS can define the appropriate conceptual model. After intersection of arcs, reordering of streams, etc. it is often necessary to rebuild the polygon topology so that the topologic structure is consistent.

Related Topics

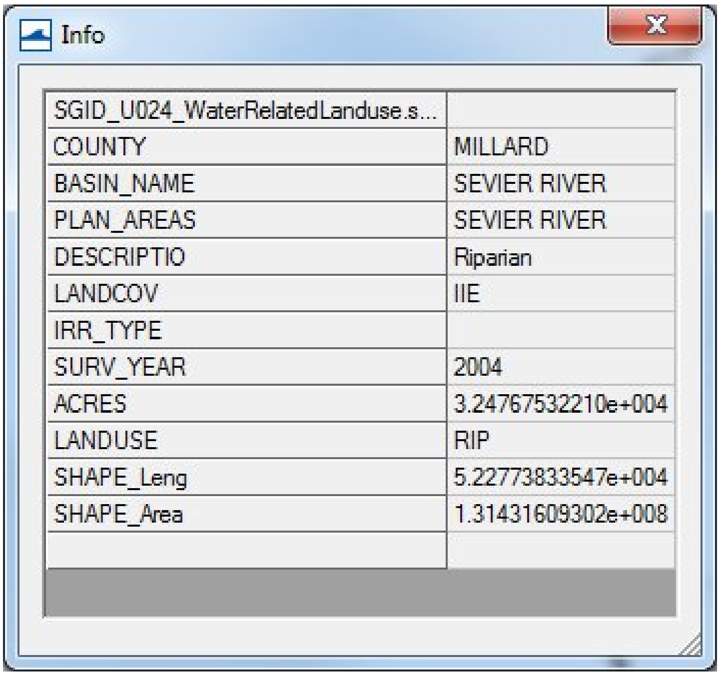
- Shapefiles

GIS Module Tools

The following tools are active in the dynamic portion of the *Tool Palette* whenever the GIS module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the *Graphics Window* with the cursor depends on the current tool. The table below describes the tools in the *GIS tool palette*.

Tool	*Tool Name*	*Description*
	Select ArcObject	The **Select ArcObject** tool is used to select ArcObjects in the *Graphics Window*. This tool is only available if ArcView is installed locally on the user's desktop.
	Get ArcObject Attributes	The **Get ArcObject Attributes** tool is used to select GIS objects and display information relating to that object in an *Info* dialog. This tool is only available if ArcView is installed locally on the user's desktop.
	Select ArcObject	The **Select ArcObject** tool is used to select shape objects in the *Graphics Window*.

Get ArcObject Attributes Tool Info Dialog



Example of the *Info* dialog that appears when using the **Get ArcObject Attributes** tool

Clicking on an object when using the **Get ArcObject Attributes** tool will bring up the ArcObject Attributes *Info* dialog.

Related Topics

- GIS Module

GIS Module Menus

The following menus are available in the the GIS Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

Data

Most of the SMS modules have a *Data* menu, but the items in this menu are different for each module. The GIS Module commands include:

GIS Data Menu Commands

- **Enable ArcObjects**

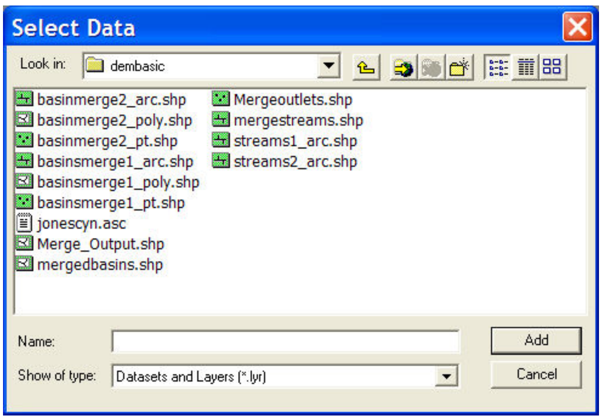
  ArcObjects is a development platform provided by ESRI [1] that allows developers of other applications (such as SMS) to incorporate ArcView/ArcGIS capability directly within their application. SMS can use ArcObjects to access some of the same functionality in SMS that is available in ArcView, providing you are running on a computer that has a current license of ArcView.

  The *Data* | **Enable ArcObjects** command queries the ESRI license manager for ArcView/ArcGIS to see if a license exists. If a valid license is found then the ArcView functionality within SMS is enabled and you will be allowed access. If a license is not found then the ArcView specific features remain unavailable.

- **Add Data**

  SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

  The **Add Data** command is only available if ArcObjects have been enabled. The **Add Data** command is used to open Datasets and layers (*.lyr) files into SMS using ArcView®.



Select Data Dialog

  When ArcObjects is enabled you are able to load any of the ESRI supported formats, including shapefiles, coverages, geodatabases, grids, images, CAD files and others, as GIS data layers in SMS. These data can then be converted to feature objects in map coverages.

- **Add Shapefile Data**

  The Add **Shapefile Data** command is only available if ArcObjects have **NOT** been enabled. The **Add Shapefile Data** command is used to open a Shapefile (*.shp) into SMS.

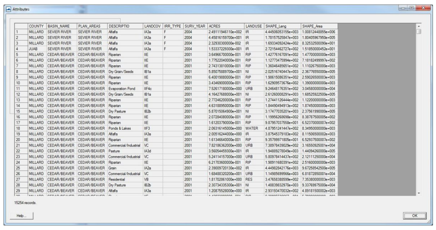
- **Add MIF/MID File Data**

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

The **Add MIF/MID Data** command is used to open a MIF/MID file (*.mif) into SMS using ArcView®.

- **Attribute Table**

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

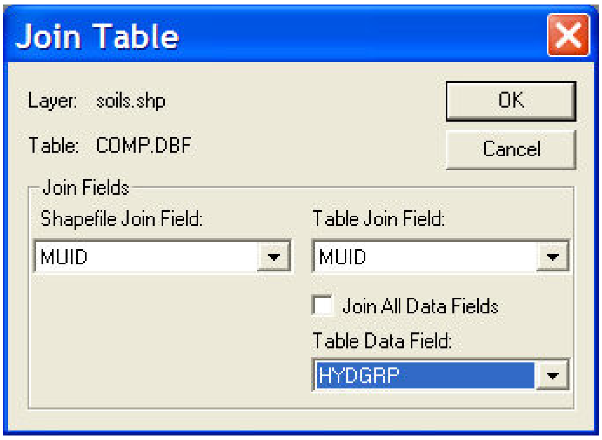


Example of the *Attribute Table* dialog using a shapefile

The **Attribute Table** command opens the *Attribute Table* dialog. The shapefile layer of interest is specified in the *Attribute Table* dialog. Once the shapefile layer has been specified, the *Attributes* dialog will be shown. The attributes for each record in the specified layer can be viewed in the *Attributes* dialog.

- **Join Table to Layer**

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.



Join Table Dialog

The **Attribute Table** command opens the *Attribute Table* dialog. The shapefile layer of interest is specified in the *Attribute Table* dialog. Once the shapefile layer has been specified, the corresponding DBF Table File (*.dbf) must be selected. The *Join Table* dialog is then shown.

The **Join Table to Layer** command, available when right-clicking on a layer in the *Project Explorer*, allows you to join the attributes of one database file (*.dbf) to the features of a GIS layer based on a key attribute field. This is particularly important when the features are stored in a shapefile with a minimal set of attributes, and additional attributes are stored in a separate *.dbf file. The two files are related based on an attribute field named MUID. Other GIS data layers may be similar where the features contain some kind of key indexing field and the attributes are stored in a separate table that can be joined to the features based on the index field values.

After selecting the **Join Table to Layer** command you will be prompted for the database file you wish to join using the standard *select file* dialog. The *Join Table* dialog will then appear and you will be asked to select the *Join Field* from the GIS data layer attributes and the *Join Field* from the table you are joining to the GIS data layer. Often these field names will be the same as in the example below, but they are not required to be the same. The important thing is that they contain similar information from which a join can be made. Finally, you can select to join all of the attributes from the join table or just add a specific field.

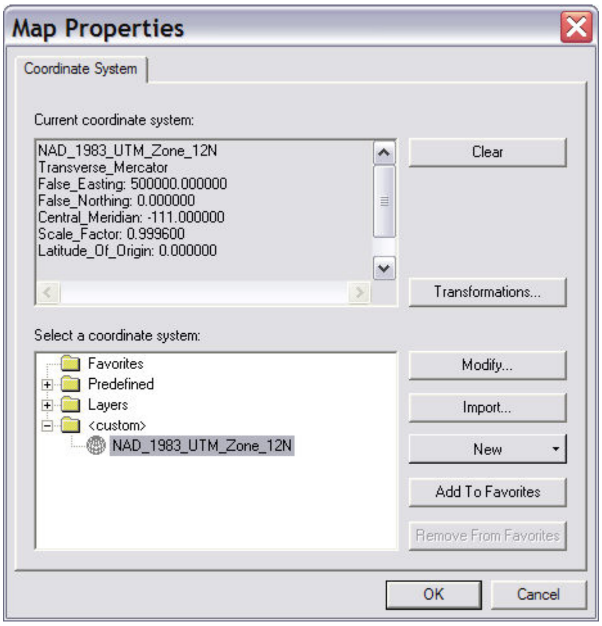
The join does not permanently alter the GIS data layer on the hard drive of your computer.

- **Layer Properties**

  The **Layer Properties** command opens the *Select a layer* dialog. The shapefile layer of interest is specified in the *Select a layer* dialog. Once the shapefile layer has been specified, the *Layer Properties* dialog is shown. See the ESRI ArcGIS documentation [2] for further explanation of the *Layer Properties* dialog. The **Layer Properties** command is only available if ArcObjects have been enabled.

- **Map Properties**

  The *Map Properties* dialog is used when ArcObjects® is enabled to specify the coordinate system that you wish to display/map features from the ArcGIS® data layer. An ArcGIS® data layer should have a currently defined coordinate system associated with it. If the coordinate system is geographic (latitude/longitude), then ArcObjects® is able to "guess" correctly at the projection. Using the coordinate system as defined in the *Map Properties* you can specify the coordinate system you wish to use to display features/rasters. While this does not change the actual geometry of the layer, it will display in the main graphics window according to this projection and any data mapped to coverages will be mapped into the coordinate system specified by the *Map Properties*.



Map Properties Dialog

GIS layers can have an associated global projection. If a layer has an associated projection, the entities will be automatically displayed in the current project projection. GIS projection information can come from multiple sources:

1. **Files** – ESRI shapefiles can have an associated prj file that contains the projection information. MapInfo MIF/MID files contain projection within the MIF file.
2. **Assigned in SMS** – You can assign a projection to a layer by right-clicking on the layer and choosing *Coordinate Conversions*. When you do this, SMS will save a PRJ file or a new set of MIF/MID files with the updated information.

Selection

The GIS Module Selection menu commands are only available if ArcObjects have been enabled. The *GIS Module Selection* menu commands include:

GIS Selection Menu Commands

Command	ArcObjects® Required	Description
Select by Attributes	Yes	Opens the *ArcObjects® Query Wizard* Dialog. See the ESRI ArcGIS documentation [2] for further explanation of the *Query Wizard* dialog.
Select by Location	Yes	Opens the *ArcObjects® Select By Location* Dialog. See the ESRI ArcGIS documentation [2] for further explanation of the *Select By Location* dialog.
Clear Selected Features	Yes	Clears the current selection
Interactive Selection Method	Yes	Change the ArcObjects® selection options in use. Options include: • Create New Selection • Add to Current Selection • Remove from Current Selection • Select from Current Selection
Selectable Layers	Yes	Opens the *ArcObjects® Check Selectable Layers* dialog. The layers to be selected can be specified.

Mapping

The *GIS Module Selection* menu commands are only available if ArcObjects have been enabled. The *GIS Module Selection* menu commands include:

GIS Selection Menu Commands

Command	ArcObjects® Required	Description
Arc Objects → Feature Objects	Yes	Opens the *GIS to Feature Objects Wizard*
Shapes → Feature Objects	No	Opens the *GIS to Feature Objects Wizard*
Feature Objects → Geodatabase	Yes	Saves the Feature Objects as a Personal Geodatabase file (*.mdb)

GIS Module Right-Click Commands

The following right-click commands are available for GIS objects:

- **Convert to TUFLOW Rainfall Boundary Conditions**

  If the format is correct, the GIS data can be converted into TUFLOW rainfall boundary polygons on a TUFLOW 1D/2D BCs and Links coverage. For more information, see TUFLOW Boundary Conditions.

Related Topics

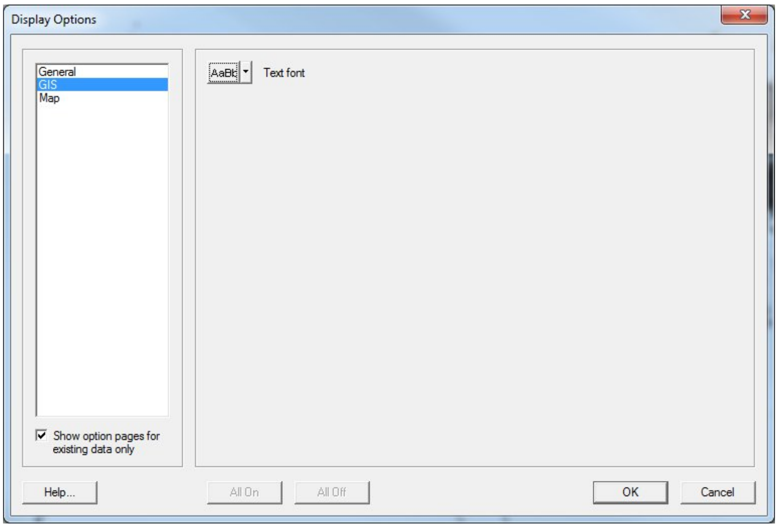
- GIS Module

GIS Module Display Options

The properties of the GIS Module data SMS displays on the screen can be controlled through the *Display Options* dialog. The entities associated with the GIS Module module with display options are shown below. Some of these entities also show an **Options** button to the right. For these entities, additional display controls are available. The available *GIS Module display options* include the following:

- **Text font**

  Controls the display of text. The size, color, style, and font of the text can be adjusted.



Display Options dialog showing *GIS* options

Related Topics

- GIS Module
- Display Options

ArcObjects

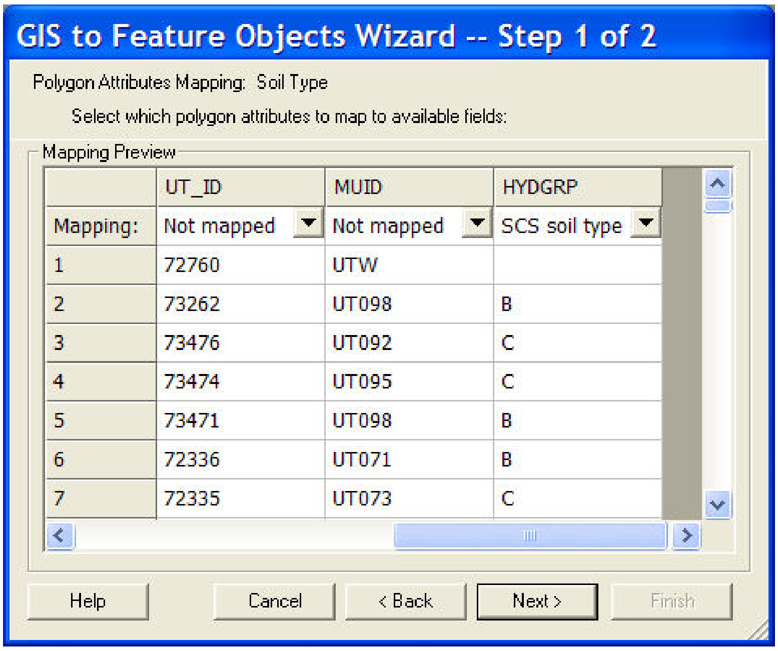
ArcObjects® is a development platform provided by ESRI® [1] that allows developers of other applications to incorporate ArcView/ArcGIS® capability directly within their application. ArcObjects® is used to incorporate ArcView® functionalities into XMS software. This allows users to use ArcView® functionality within XMS software. In order to use ArcView® functionality, a current license of ArcView® must be installed. Without a license, much of the same functionality is available, the primary differences being that only the shapefile format is supported, and many of the selection and display capabilities are minimal.

Related Topics

- GIS Module (GMS)
- GIS Module (SMS)
- GIS Module (WMS)

GIS to Feature Objects Wizard

While future versions of the XMS software may be able to process some commands directly from the GIS data layers, currently the user must map all desired features as part of model development to feature objects in a map coverage. One way to do this is to convert an entire shapefile directly to a map coverage. One problem with this approach is that the extents of the GIS data layer may be much larger (i.e. an entire state) than the area of interest. In this case, it may be more efficient to select only those GIS features (points, lines, polygons) that overlay your study area and map those to feature objects in a map coverage.



GIS to Feature Objects Wizard -- Step 1 of 2

Selecting the *Mapping* | **ArcObjects → Feature Objects** command (with an ArcObjects license), or the ***Mapping* | *Shapes → Feature Objects*** (without an ArcObjects license) launches a mapping wizard which guides you through the process of converting your GIS data layer features to feature objects in a map coverage. Before beginning the mapping process you should first go to the map module and make sure that the currently active coverage is the coverage you wish to map GIS data layer features to. Also, you will associate the GIS attributes with coverage attributes, so you must make sure the coverage attributes are defined before you can do the conversion.

After making sure you will be mapping to the correct coverage select the GIS features which overlay your study area and that you wish to map (this is done with the selection tool(s) in the GIS module). If you wish to map all the features you can choose the *Edit* | **Select All** command, or just execute the Mapping command, and you will be prompted if you want to convert all features since none are selected.

If you have ArcObjects enabled you will see that the *Mapping* | **ArcObjects → Feature Objects** command is activated whereas if you do not have ArcObjects enabled you will see that the *Mapping* | **Shapes → Feature Objects** command is activated. After choosing the appropriate mapping command you will see the *Mapping Wizard* shown below. This wizard will guide you through the rest of the process. The first dialog in the mapping wizard contains instructions and marks the beginning point of mapping for selected features. The first of two steps is to map the GIS attribute fields of the features to the coverage attributes. Common attribute names are automatically mapped.

The second step marks the end of the wizard and after selecting Finish all selected features will be converted to feature objects within the active coverage. Attributes of mapped fields will be saved accordingly as attributes of the feature objects.

Generic Model arc or node mapping

Users can now bring in GIS data (shapefile or MIF/MID) and convert this data to generic model node or arc attributes. First you should have arc and/or node boundary conditions defined in the Generic Model. Once those are defined, you will see them in the drop down box when mapping. Each boundary condition will contain a (on/off) item followed by parameters. Example:

- Hydro→(on/off)
- Hydro→Manning
- Hydro→Flowrate

Mapping "Hydro→(on/off)" will turn on or off the node/arc. Mapping a parameter such as "Hydro→Manning" will automatically assign the node/arc as "on" unless "Hydro→(on/off)" is explicitly mapped to "off".

Related Topics

- GIS Module

3.7. Map Module

Map Module

At a glance

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

- Create and edit GIS like data
- Used to create conceptual models as well as data for other purposes
- Conceptual model is a geometry (mesh/grid) independent representation of the numeric model domain and/or boundary conditions
- Conceptual models can be converted to model geometry and boundary conditions
- Conceptual model makes it easier to create, edit, and alter models

The Map module provides tools for creating, managing and editing feature objects. Feature objects are geometric entities, meaning they have a defined position or shape, and the attributes associated with those entities. The simplest feature object is a feature point, which defines a single location. Increasing in complexity, the next GIS object is an arc, which defines a line or polyline. Areas enclosed by polylines can be classified as a feature polygon.

Feature objects with related attributes are grouped into layers or coverages. The coverage definition includes a "type" that determines the attributes available for the objects in the coverage. See the list of coverage types to learn about the attributes associated with objects in that specific coverage.

The principal application of coverages in the SMS is to facilitate the representation of a numerical simulation in a representation that is independent of a specific discretization (a specific set of nodes and elements or cells). This allows the modeler to interact with a much smaller set of entities and reduces redundant effort in the modeling process.

A secondary application of coverages is to define geometric objects for data extraction from numerical model results.

Conceptual Model

A conceptual model is a high level representation used to define attributes used in the Mesh or Cartesian Grid generation process such as:

- Bathymetry source
- Materials information
- Boundary conditions

Creating Feature Objects

Feature objects are the building blocks of a conceptual model. They define the geometric shapes, locations and extents of objects in the model. There are several mechanisms for creating feature objects including:

- Extracting feature arcs from the contours of a scatter set. See Scatter Contour to Feature for more information.
- Importing from a web data source such as a coastline database. See Import from Web for more information.
- Importing from CAD data.
- Interactive definition (digitizing) using the Map Module Tools.
- Creation as a stamped feature to define built up embankment or dredged channels.

```
   Normal   0          false   false   false
```

Elevations of Feature Objects

In the map module, Nodes, vertices and arcs all have an elevation attribute. That means that you can assign an elevation to individual nodes, points or vertices, or you can assign it to the arc. If you assign an elevation to an arc, the attribute of the nodes and vertices in that arc are updated as well. This will override any z-value specified for individual vertices or nodes on the arc.

The elevations of the map objects can also be assigned using the interpolation from the scatter module. In this case, each object (node, point, vertex, and arc) are assigned an elevation (z-value) based on the scatter set. The location for interpolation of the arc is the mid-point of the arc.

When you digitize in the map module, elevations are assigned as with other digitization in SMS. That means that when you create a node, point, or vertex, it is assigned the default elevation value for digitization. The default elevation is initialized to 0.0. The default changes any time you specify a Z-value. Therefore, if you create a map point or node, and specify an elevation for that selected point, the value you specify is now the default value for newly digitized points, nodes and vertices. (Note: when you create mesh nodes, there is an option to ask for an elevation each time you create a node, but this option is not available for scatter vertices or map module objects.)

When you convert a map coverage to a scattered data set, you have the option of using the arc elevations or the node and vertex elevations for the new scatter set. (There is also an option to use the arc spacing, but that is for a different purpose. It is not an elevation, but is useful sometimes as a function on a scattered data set.)

Map Module Overview

Map Module

Overview

The Map module provides tools for creating, managing and editing feature objects. Feature objects are geometric entities, meaning they have a defined position or shape, and the attributes associated with those entities. Feature objects with related attributes are grouped into layers or coverages. The coverage definition includes a "type" that determines the attributes available for the objects in the coverage. The principal application of coverages in the SMS is to facilitate the representation of a numerical simulation in a representation that is independent of a specific discretization (a specific set of nodes and elements or cells). This allows the modeler to interact with a much smaller set of entities and reduces redundant effort in the modeling process.

The map module also provides the tools to create and edit GIS like data and conceptual models as well as data for other purposes. Conceptual model is a geometry (mesh/grid) independent representation of the numeric model domain and/or boundary conditions and they can be converted to model geometry and boundary conditions. Conceptual model is also a high level representation used to define attributes used in the Mesh or Cartesian Grid generation process such as bathymetry source, materials information, and boundary conditions.

Functionalities

Map Module Display Options

Project Explorer

The following Project Explorer right mouse click menus are available when the right mouse click is performed on a Map Module item.

Map Module Root Folder Right Click Menus

Right clicking on the Map module root folder in the project explorer invokes an options menu with the following options:

- **New coverage**' – Opens the *New Coverage* dialog.
- **Clear Coverages** – Deletes all coverages.
- **Display Options** – Opens the *Display Options* dialog.

Coverage Item Right Click Menus

Right clicking on a Map item in the Project Explorer invokes an options menu with the following module specific options:

- Type – Change the coverage type.

Right click options for the coverage may also include options applicable only to the specific coverage type.

Menus

The following menus are available in the Map Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Feature Objects Menu

How do I?

To learn more about how to use the Map Module go to the Tutorials section.

Tools

The following tools are contained in the dynamic portion of the Tool Palette when the Map Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the map tool palette.

Tool	Tool Name	Description
	Select Feature Point or Node	The **Select Feature Point or Node** tool is used to select stand alone feature points or the ends of arcs. A single point is selected by left-clicking directly on it. Multiple points can be selected at once by dragging a box. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. Additional points can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new points without holding the SHIFT key will first clear the selection list and then add the newly selected points. A selected point can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature points are locked so they are not accidentally dragged. When a single point is selected, its location is shown in the *Edit Window*. The coordinates can be changed by typing in the edit field. The Graphics Window's status bar will display information on the selected items depending on the settings find through the *File* | **Info Options** command in the *File* menu. Selected points can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Arcs attached to the deleted points are deleted. This tool is available when one or more feature points exist.
	Create Feature Point	The **Create Feature Point** tool is used to place new feature point in the current coverage. A single point is created at a time by left-clicking at the coordinate desired. The newly created point is selected to allow Z Coordinate changes in the *Edit Window*. This tool is always available, however, creating a feature point is only allowed while in plan view.
	Select Feature Vertex	The **Select Feature vertex** tool is used to select one or more vertices on an arc. These vertices define the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.
	Create Feature Vertex	The **Create Feature vertex** tool is used to create a new vertex on the interior of an arc. The vertex is created at the current arc location, but can be selected and moved to change the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.
	Select Feature Arc	The **Select Feature Arc** tool is used to select one or more existing feature arcs. This is typically done to assign attributes to an arc or delete the arc. A single arc is selected by left-clicking directly on it. Double clicking on the arc will bring up the arc attributes dialog or that arc. Multiple arc can be selected at once by dragging a box. Additional arcs can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new arcs without holding the SHIFT key will first clear the selection list and then add the newly selected points. A selected arc can be removed from the selection list by holding the SHIFT key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature arcs may have elevations associated with the arc as an entity. This is independent of the node and vertex elevations. When a single arc is selected, its elevation is shown in the *Edit Window*. The coordinates can be changed by typing in the edit field.

		Multiple feature arcs may also be selected to create a feature arc group to associate attributes with a string of arcs rather than a single arc. These arcs must connect end to end. The arc group is created from the **Feature Objects** menu command. The Graphics Window's status bar will display information on the selected items depending on the settings find through the File | Info Options command in the *File* menu. Selected arcs can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Nodes attached only to the deleted arcs are deleted. This tool is available when one or more feature arcs exist.
	Create Feature Arc	The **Create Feature Arc** tool is used to create a new feature arc.
	Select Feature Arc Group	The **Select Feature Arc Group** tool is used to assign attributes to a string of arcs. The group must be created before it can be selected as a group. The attributes of the group then operate just as if the group was a single arc.
	Select Feature Polygon	The **Select Feature Polygon** tool is used to select polygons to assign attributes to them. The attributes associated with the polygon depends on the coverage type. Double-clicking on the polygon will invoke the polygon attributes dialog for the polygon. Multiple polygons may be selected just as with arcs and nodes. The polygons must be created using the **Feature Objects** menu command.
	Select 1D Grid Frame	The **Select 1D Grid Frame** tool only appears when the coverage is associated with a 1D grid model (Cascade or GENESIS).
	Create 2D Grid Frame	The **Create 2D Grid Frame** tool is used to create a new grid frame for the creation of Cartesiona grids. The grid frame is defined by clicking three times in the graphics window. The first click defines the origin. The second click defines the I axis of the grid frame (both extents and direction). The third click defines the extents of the J axis. The direction is set to be perpendicular to the I axis.
	Select 2D Grid Frame	The **Select 2D Grid Frame** tool allows the user to select a grid frame and alter its position, orientation and size.

Related Coverages

Generic Coverages:

- Area Property
- Mapping
- Observation
- Particle/Drogue
- Spatial Data
- Stamping
- Grid Frame Dialog

Model Specific Coverage

- ADCIRC
- ADH
- BOUSS2D
- CGWAVE
- CMS-FLOW
- CMS-WAVE
- FESWMS
- GenCade
- Generic 2D Mesh
- PTM
- STWAVE
- TABS
- TUFLOW

3.7.a. Coverage Types

Coverages

Map Module

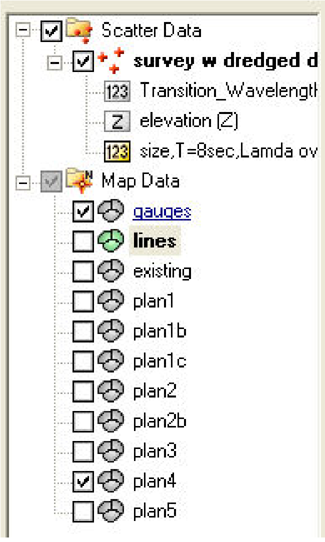
Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

Coverages

Feature objects in the Map module are grouped into coverages. Each coverage has a specific type, which determines the attributes that can be associated with geometric objects in that coverage. The coverages are grouped into conceptual models.

A coverage is similar to a layer in a CAD drawing. Each coverage represents a particular set of information. For example, one coverage could be used to define meshing zones and another coverage could be used to define zones of consistent roughness parameters. These objects could not be included in a single coverage since polygons within a coverage are not allowed to overlap and material zone boundaries don't necessarily coincide with meshing zone boundaries. Alternatively, one coverage could define Cartesian grid parameters for the same zone.

Coverages are managed using the Project Explorer. When SMS is first launched, a default coverage exists. If the user creates feature objects they are placed in the current coverage. When multiple coverages are created, one coverage is designated the "active" coverage. New feature objects are always added to the active coverage and only objects in the active coverage can be edited. The figure below shows several coverages in the Project Explorer. The active coverage is displayed with a green colored icon and bold text. A coverage is made the active coverage by selecting it from the Project Explorer. In some cases it is useful to hide some or all of the coverages. The visibility of a coverage is controlled using the check box next to the coverage in the Project Explorer. In the example below, several design options are not displayed.



Creating a New Coverage

A new coverage can be created by right-clicking on the Map Data folder in the Project Explorer. Select New Coverage from the pop-up menu. The New Coverage dialog will appear and the coverage type will need to be defined. Each coverage type is organized according to whether it is a Generic or a Model type coverage. If a coverage type was selected that has attributes associated with it, they can be changed by clicking the Attributes button. If there are no attributes associated with the selected coverage type the Attributes button will be unavailable. The name of the coverage is also specified here. After clicking the OK button, the new coverage will appear in the Project Explorer.

Coverage Right-click Menu

Right-clicking on a coverage brings up a menu with the following options:

- **Delete** – Removes the selected coverage along with any data in the coverage.
- **Duplicate** – Will create a copy of the coverage.
- **Rename** – Allows the coverage to be given a different name.
- **Convert** – Coverage feature object data can be mapped to other geometric objects or numerical models by selecting one of the Map -> ... options in the Convert submenu.
- **Coordinate Conversion** – The Coordinate Conversion dialog opens, which allows the user to specify a coordinate conversion to be performed on the coverage data.
- **Metadata** – Metadata associated with this coverage can be edited and viewed.
- **Zoom to Coverage** – Frames the graphics window to the extents of the data displayed in the selected coverage.
- **Type** – Sets the coverage type.

Merging Existing Coverages

Occasionally, a user may want independent features of two separate coverages combined into one converage. SMS allows the user to merge these two coverages together. The user should left click one of the coverages listed in the data tree then multi-select the other coverages to be merged. This can be done one at a time by holding the CTRL key, or several adjacent coverages can be selected by holding the SHIFT key and then clicking the last adjacent dataset.

Once all datasets to be merged have been selected, the user simply accesses the right-click menu and chooses "Merge Coverages". A dialog may appear asking the user if they wish to delete the coverages used to create the merged coverage.

Coverage Types

The attributes of entities on a coverages belong to a list of attributes associated with a type of coverage. For example, arcs in mesh coverages have boundary conditions compatible with the specific finite element model they are associated with, and polygons in those coverages include attributes associated with meshing and material types. The following coverage types include the following:

- Generic Input (Preprocessing)
  - Area Property
  - Feature Stamping
  - Spatial Data
- Generic Output (Postprocessing)
  - Observation
  - Particle/Drogue
  - Shoals
- Generic Model Interface
  - Mapping
- Models
  - ADCIRC
  - ADH
  - BOUSS-2D
  - CGWAVE
  - CMS-FLOW
  - CMS-WAVE
  - FESWMS
  - GenCade
  - GENESIS (replaced by GenCade)
  - Generic Mesh
  - PTM
  - STWAVE
  - TABS
  - TUFLOW

Related Topics

- Map Module

3.7.a.1. Generic Coverages

Generic Coverages

Generic coverages can be selected by right-clicking on the map module data and selecting *Type* | **Generic**.

Generic Coverages:

- Area Property
- Mapping
- Observation
- Particle/Drogue
- Spatial Data
- Stamping
- Grid Frame

Related Topics

- Map Module
- Model Specific Coverages

Area Property Coverage

An area property coverage is used to map properties such as Manning's roughness values to the mesh, grid, or cross-sections.

Materials are used to define different values for the property the material represents. For example, if the material represents the Manning's roughness value, materials are defined to represent the different Manning's roughness values to be included in the model (stream, floodplain, field, roadway, etc.). Polygons can then be created to define the stream, floodplain, field, and roadway with the corresponding material assigned.

Materials are assigned a name, color and pattern for display, and model specific attributes.

Many of the data entities constructed and edited in SMS (i.e., elements, cells) have a material ID associated with them. This material ID is an index into a list of material types. Materials contain model specific parameters such as manning's roughness, or bed material grain size. A global list of material attributes is maintained and can be edited using the menu command *Edit* | *Materials*. This command brings up the *Materials Data* dialog where each material is assigned an ID number. This dialog can be used to delete unused materials, create new materials, and assign a descriptive name, color, and pattern to a material. This general information is saved in the material file. The materials defined within the Materials Data dialog are available for all modules. Contents

Dialog Description

The *Materials Data* dialog is accessible from the menu command *Edit* | **Materials** or from model specific material properties windows (ex. ADH), available in the model specific menu. The dialog is resizable by dragging on the window edges.

When a new mesh element or grid cell is created, the material is assigned to the new object based on the materials options in the *Element Options* dialog.

Model specific material properties such as Manning's $n$and Eddy viscosity are edited using commands available in the model specific menu.

Materials Spreadsheet

The materials spreadsheet contains three columns (ID, Name, and Pattern) for the defined materials. All IDs must be unique and the spreadsheet can be sorted by clicking on the column headings. The default *Disable* material cannot be edited (except the display pattern) and will always be at the top of the spreadsheet regardless of sorting. Each material is accompanied by a pattern button in the *Pattern* column. To select a pattern, click on the preview section (left side) of the button to open the *Pattern Attributes* window. To quickly edit only the color, click on the down arrow (right side) of the button, and make a selection in the pop up color palette.

Buttons

- New – Inserts a material into the spreadsheet with the lowest unique ID available and a default name and pattern.
- Delete – Removes the currently selected material from the spreadsheet.
- Copy – Creates and inserts a copy of the currently selected material with the lowest unique ID available and a default "copy of" name.

Legend

- Legend – Check box with the associated **Options...** button controls the display of a legend of the materials in the *Graphics Window*.
- Options... – Opens the *Legend Options* dialog. The options for the legend are edited in the *Legend Options* dialog. These options include:
  - The name to be displayed on the legend.
  - The specification of where on the screen the legend will appear.
  - The font to be used in the legend.
  - The size of each entry in the legend.

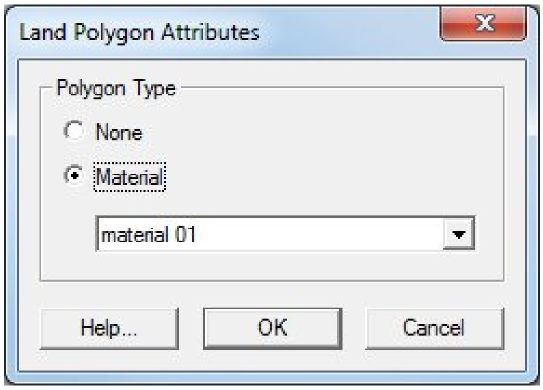
Note: only active materials are included in the legend.

Area Property Polygon Attributes Dialog

The *Land Polygon Attributes* dialog is used to set the attributes for feature polygons. Attributes that can be specified for each polygon include:

- Polygon Type
  - None
  - Material

Materials must be defined in the *Materials Data* dialog before they can be assigned to a polygon.



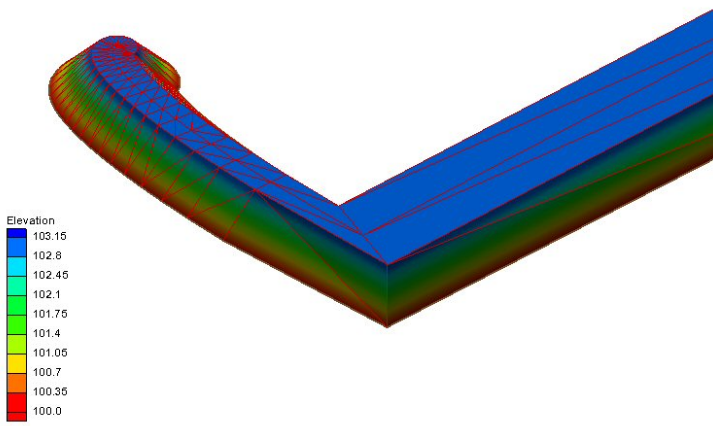
Land Polygon Attributes dialog

Related Topics

- Coverages

Feature Stamping

Feature stamping is the terminology used to refer to the insertion of man made structures into a natural topography or bathymetry set. In common terms, this means adding an embankment (such as a levy) or dredging a channel. A stamped feature usually follows a linear object or centerline. However, it can also be based around a single point to create a mound or pit, or applied to only one side of a closed line (a polygon) to create a flat topped mound or flat bottomed pit.



A stamped embankment with a guidebank

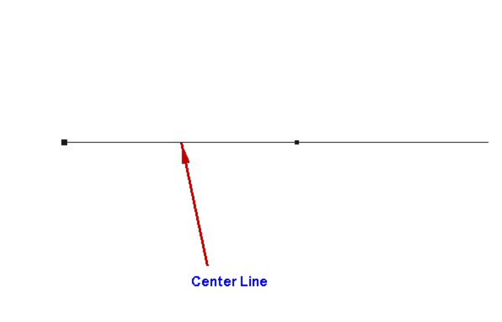
The Process

The basic steps to define a linear stamped feature include:

1. Define the stamping coverage and centerline (or focal point) of the stamped feature
2. Assign attributes to the centerline including
   1. The elevation along the centerline
   2. The cross sections along the centerline
3. Stamp the feature. This converts the centerline and its attributes to:
   1. Another coverage containing all the extents and details of the feature
   2. A scatter set defining the elevation for the feature.

Sample problems in the section below illustrate the procedure.

Define the Coverage and Centerline



A centerline arc – simple

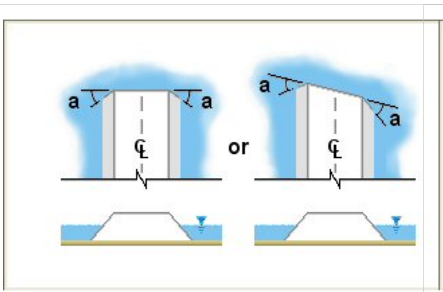
SMS utilizes a coverage of type "Stamping" to create the stamped features. Depending on the application, you may want to have multiple "Stamping" coverages to represent different design alternatives. Each coverage may contain multiple features. Create a "Stamping" coverage by right-clicking on the "Map Data" entry in the *Project Explorer* and selecting the **New Coverage** command. Then right-click on the new coverage and set its type to "Stamping".

The ambient geometry is defined by a scatter set (and one of its associated datasets). This surface determines the cut-off for the sloped banks of a stamped feature. Right-clicking on the coverage and selecting "Properties" brings up the dialog that associates a specific dataset to the stamping coverage. You must also specify whether this surface is defined as elevations (positive up) or depths (positive down). By default, SMS interprets this surface as elevation data.

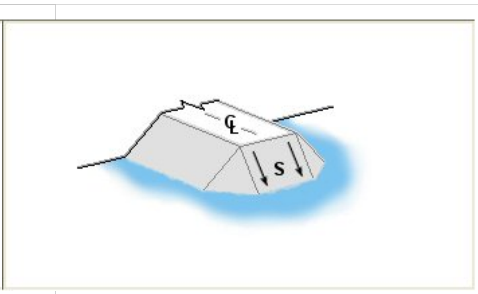
Define Feature Attributes

Any arc or point created in a "Stamping" coverage has attributes to create a stamped feature. Attributes are assigned in the *Stamping Point Attributes* Dialog or *Stamping Arc Attributes* Dialog. Attributes include:

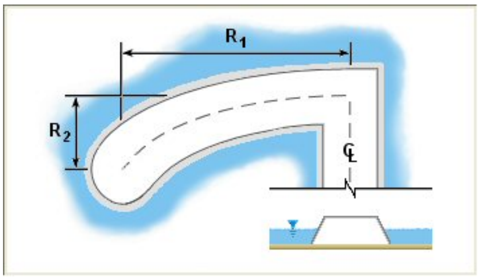
- A feature name – This will be used when SMS creates stamped feature coverages and scatter sets from the stamping coverage.
- The feature type (cut = channel or fill = embankment) – If you have a stamped feature that contains both cut and fill sections, create one coverage as a cut, then duplicate the coverage and change the copies type to fill.
- A base elevation (the top elevation of the embankment or the bottom of the channel) – This can be specified from the node/vertex elevations, as a constant, or extracted from a curve of elevation -vs- distance.
- The cross sectional shape(s) – The cross section can be defined as a template, which is propagated all along an arc, or can be individually specified at each vertex in an arc. The cross section can also be defined individually for the left and right side of the arc. One point on each side of the cross section can be specified as the "shoulder". For a channel, this would be the "toe", but the reference in SMS is the shoulder point. When the arcs representing the shoulder are created, the user has the option to create an arc along this shoulder. If vertical walled structures are desired, the cross section can simply stop at the edge (shoulder). This will result in a feature arc at the edge and a scatter set for the top of the structure.
- The method for treating the ends of the structure – The options include:
  - Wingwalls
  - A sloped abutment (spillthrough)
  - A guidebank



Wing Walls



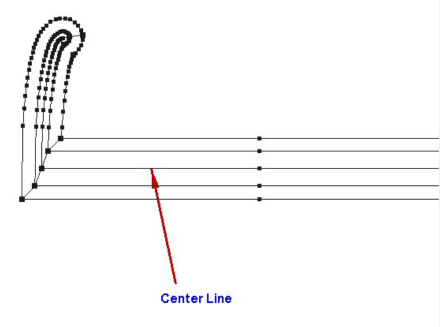
Spillthrough



Guidebank

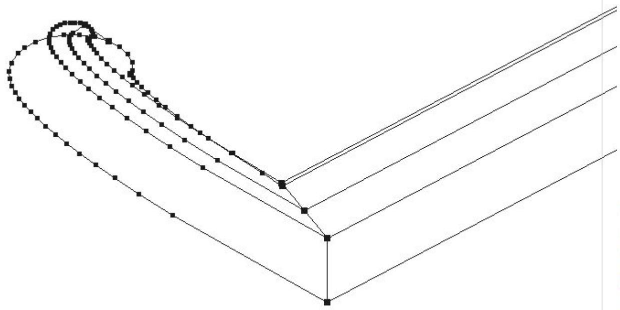
Stamping the Feature

To create the stamped feature, right-click on the "Stamping" coverage and select **Convert → Stamp Features ...**. This brings up a dialog that allows the user to specify what output should be created from the process. Specifically, the process can create:

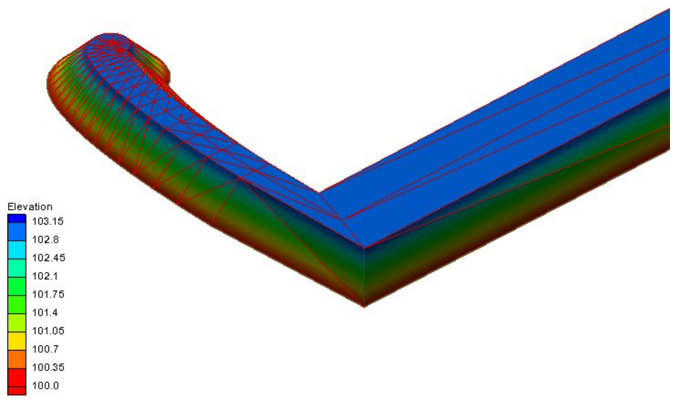


Stamped Feature Arcs – Created from Centerline

- A new scatter set which defines the elevation points for the new structure. This is named based on the stamped feature name. SMS triangulates all these points to create a surface and trims the scatter set to the extents of the stamped feature. The arcs that make up the stamped feature are converted to scatter breaklines to assist in the triangulation.
- A new coverage which defines the breaks and extents of the new coverage. The dialog allows the user to specify the type of the coverage. Generally, it is recommended that the coverage type be selected to match the numeric engine that will evenutally be used for the simulation. Options also exist to determine whether the stamped feature will include the center line, the shoulders, and the cross sections. The extents of the stamped feature are always generated.



Stamped Feature Arcs – Oblique view



Stamped Feature Scatter Set – Oblique view

Case Studies / Sample Problems

There are a wide variety of stamped features that can be created using this tool. The Feature Stamping tutorial in the general section of the tutorials may be helpful for learning to use them.

- Embankments on a flat plain
  - Vertical sides on an embankment
  - Sloped sides on an embankment

External Links

- Emery, R. N. (2005). Refining and Expanding the Feature Stamping Process. Thesis, Brigham Young University. [1]
- Christensen, J. R. (2001). Stamped Features: Automatic Generation of Flow Modifying Structures in Conceptual Models. Thesis, Brigham Young University.
- Zundel, Alan K. and J. Ryan Christensen, "Stamped Features: Creation of Engineered Structures in Conceptual Models", International Journal of Hyrdroinformatics, Vol. 4, No. 1, February 2002, pp. 63-72.

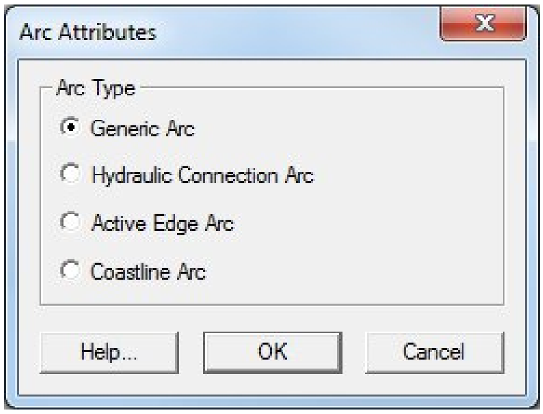
Mapping Coverage

Mapping Arc Attributes Dialog

The *Arc Attributes* dialog is used to set the attributes for feature arcs in a Mapping type coverage. Attributes that can be specified for each arc include:

- Generic Arc
- Hydraulic Connection Arc
- Active Edge Arc
- Coastline Arc

The dialog is reached by double-clicking on an arc with the **Select Feature Arc** tool with the Mapping type coverage active.



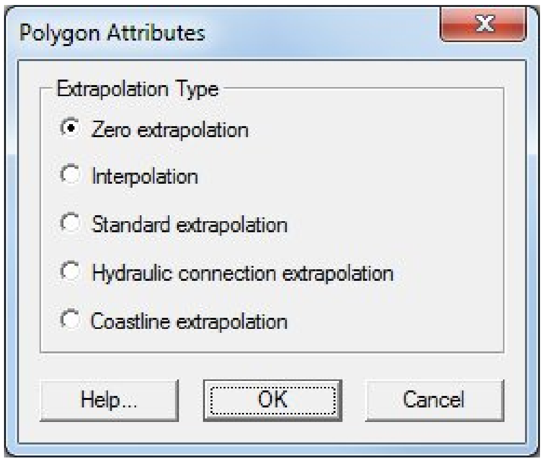
Arc Attributes dialog

Mapping Polygon Attributes Dialog

The *Polygon Attributes* dialog is used to set the attributes for feature polygons in a Mapping type coverage. Attributes that can be specified for each polygon include:

- Extrapolation Type
  - Zero extrapolation
  - Interpolation
  - Standard extrapolation
  - Hydraulic connection extrapolation
  - Coastline extrapolation

The dialog is reached by double-clicking on a polygon with the **Select Feature Poygon** tool with the Mapping type coverage active.



Polygon Attributes dialog

Related Topics

- Feature Objects Menu
- Steering
- Generic Coverages

Observations

SMS contains what is called an observation coverage that is designed to help in model verification and calibration processes. Result verification is an important part of the computer modeling process. SMS includes a number of powerful tools, associated with an observation coverage, that allow users to verify simulation results with observed data. The two tools used for verification and calibration in an observation coverage are observation points and observation arcs. Observation points are used to verify the numerical analysis with measured field data such as water surface elevation or velocity data. They are also be used to see how computed values change with time at a particular location. Observation arcs are used to view the results at a cross section or along the river profile. These tools can be used with any of the SMS models.

Creating an Observation Coverage

To create a new observation coverage:

1. Right-click the *Map Data* item in the *Project Explorer*
2. Select **New Coverage** from the right-click menu
3. Set the coverage type to *Generic* → *Observation* in the *New Coverage* dialog
4. Set the coverage name as desired
5. Click **OK** to exit the dialog

Alternatively, an existing coverage can be changed to an observation coverage by right-clicking on the coverage in the Project Explorer and setting the type to *Observation* using the right-click menu.

Creating an Observation Point

Observation points are created at locations in the model where calibration data such as the velocity or water surface elevation has been measured in the field. Each observation point is used to compare the measured values with the values computed by the model at the point's x, y location. This comparison can assist the modeler in determining the accuracy of the numerical model results. If the numerical model results do not match the observed field data, model parameters such as manning's roughness may need to be modified to obtain more accurate results.

Creating an observation point is just like creating a feature point in any other coverage type. Select the Create Feature Point tool from the Dynamic Toolbar and click the location for the feature point.

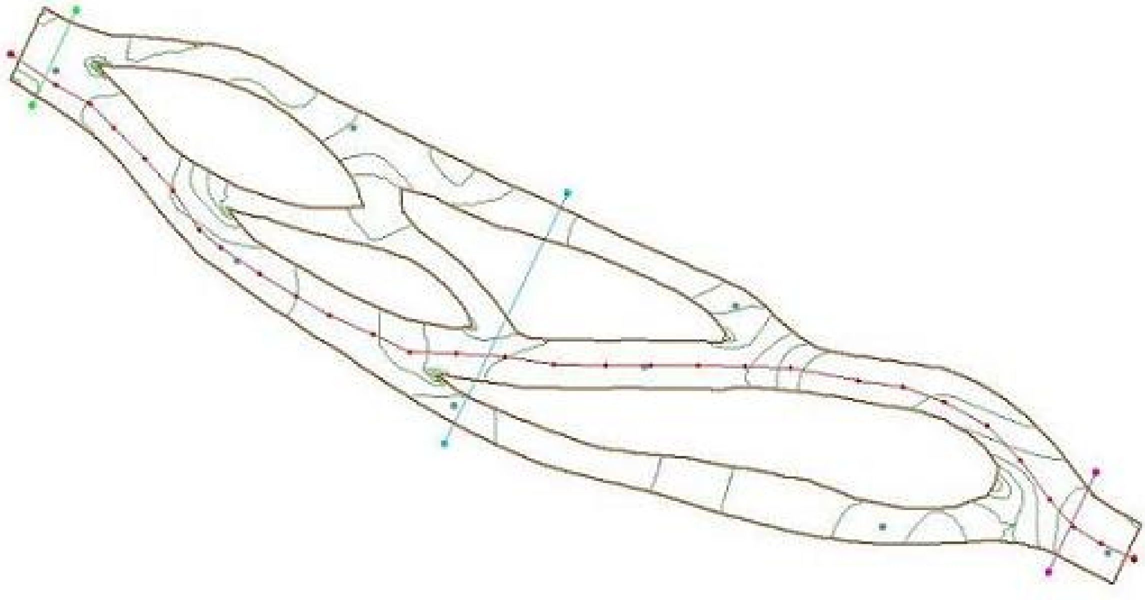
Creating an Observation Arc

Observation arcs are created at cross sections in the model where calibration data such as the flowrate has been measured in the field. Observation arcs compute fluxes across the arc. Therefore, measurements for observation arcs are called *Flux Measurements*. Each obseration arc is used to compare the measured values with the values computed by the model across the vertical plane defined by the arc. This comparison can assist the modeler in determining the accuracy of the numerical model results. If the numerical model results do not match the observed field data, model parameters such as manning's roughness may need to be modified to obtain more accurate results.

Creating an observation arc is just like creating a feature arc in any other coverage type. Select the **Create Feature Arc** tool from the Dynamic Toolbar and click out your arc. Double-click to end the arc. In an observation coverage, profile arcs and cross section arcs may be useful to analyze a simulation's solution.

Setting Observation Object Attributes

Observation point and arc attributes are defined in the Observation Coverage dialog. See Observation Coverage dialog for a description of the *Observation Coverage* dialog.



Viewing Results

In addition to viewing the results of the solution data versus the observed data on the calibration targets, additional plots can be created using the Plot Wizard. See Plot Window for a description of the available plot types.

Calibration

Calibration is the process of modifying the input parameters to a model until the output from the model matches an observed set of data. SMS includes a suite of tools to assist in the process of calibrating a model. Both point and flux observations are supported. Most of the calibration tools can be used with any of the models in SMS.

Measurement types can be defined in SMS. They are defined in the map module and are associated with points and arcs in an observation coverage. Point observations represent locations in the field where some value has been observed. In most cases, the points will correspond to gauges or high water marks and the value will be the elevation of the water (the head) or a flow velocity (and possibly direction). Flux observations represent linear or areal objects such as streams gages where the flow rate has been measured or estimated. Both point and flow observations can be assigned a confidence interval or calibration target.

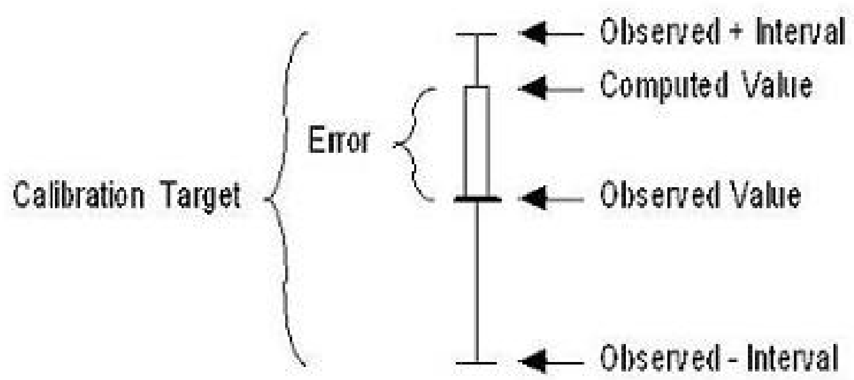
Once a set of observed point and flow values has been entered, each time a model solution is imported, SMS automatically interpolates the computed solution to the observation points. A calibration target representing the magnitude of the residual error is displayed next to each observation point and each flux object as shown below. The size of the target is based on the confidence interval or the standard deviation. In addition to the calibration targets next to the observation points, you can choose to display any of a number of statistical plots.

Calibration Target

If an observed value has been assigned to an observation point or if an observed flow has been assigned to an arc or polygon, the calibration error at each object can be plotted using a "calibration target". A set of calibration targets provides useful feedback on the magnitude, direction (high, low), and spatial distribution of the calibration error.

The components of a calibration target are illustrated in the following figure. The center of the target corresponds to the observed value. The top of the target corresponds to the observed value plus the interval and the bottom corresponds to the observed value minus the interval. The colored bar represents the error. If the bar lies entirely within the target, the color bar is drawn in green. If the bar is outside the target, but the error is less than 200%, the bar is drawn in yellow. If the error is greater than 200%, the bar is drawn in red. The display options related to calibration targets are specified in the Calibration Display Options dialog.

If the active time step is before the first observed time, or after the last observed time, the targets are drawn lighter.



Calibration Display Options

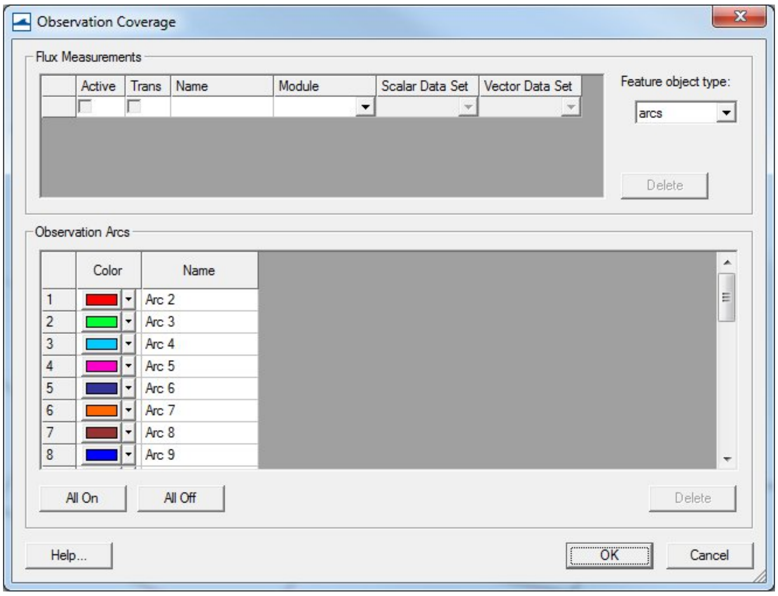
Calibration targets are drawn next to their corresponding map data (point, arc, polygon). The *Feature Objects Display Options* dialog contains a toggle labeled *calibration targets*. Below the toggle is a *Scale* edit field.

The target is drawn such that the height of the target is equal to twice the confidence interval (+ interval on top, - interval on bottom). The Scale edit field allows the user to change the general length and width of the targets independent of the range of the active dataset.

Observation Coverage Dialog

The *Observation Objects* dialog has two sections used to define the attributes of the points and arcs created in the observation coverage. To open the *Observation Coverage* dialog:

1. Make the observation coverage active in the Project Explorer
2. Select the Create Feature Point or Create Feature Arc tool from the Dynamic Toolbar
3. Select a feature point or feature arc
4. Select the menu command *Feature Objects* | **Attributes** or double-click in the previous step



Observation Coverage dialog

Dialog Layout

The options in the dialog will differ slightly based on the feature object type currently being edited (arc or point). The feature object type is specified using the combo box in the upper right of the dialog.

In addition to a unique Name and Dataset(s), two other parameters are used to define the data represented by a measurement: *Trans* and *Module*. When analyzing data that varies through time, select the *Trans* toggle. The Module of a measurement refers to the SMS module where the computed data is stored. (The Module is set by default and normally does not need to be changed.)

Observation Point Attributes

When "points" is selected in the Feature object type combo box, the top section of the dialog is entitled *Measurements* and the bottom section is entitled *Observation Points.* The *Measurements* section is used to specify which datasets correspond with the observed data entered in the *Observation Points* section. The *Measurements* section is used to enter the observed data at each point. Each observation point is assigned the following attributes:

- **Color –** Color of the observation point.
- **Observe –** Turn the observation point on or off.
- **Name –** Name of the observation point.
- **X –** X-location of the point.
- **Y –** Y-location of the point.
- **Observed Value –** Value measured in the field corresponding to the active measurement.
- **Interval –** Allowable error (±) between the computed value and the observed value. Model verification is achieved when the error is within the interval (±) of the observed value.
- **Angle –** When a measurement for observation points is tied to a vector dataset (as is the case with a Velocity measurement) an angle needs to be specified. This angle is an azimuth angle with the top of the screen representing north when in plan view.

Observation Arc Attributes

If "arcs" is selected in the combo box, then the top spreadsheet had the name *Flux Measurements* and the bottom spreadsheet has the name of *Observation Arcs.*

To define an arc measurement, the Name must first be defined. In addition to a unique Name, a scalar and vector dataset must be assigned to it. Two other parameters are also used to define the data represented by a measurement: *Trans* and *Module*. When analyzing data that varies through time, select the Trans toggle. The Module of a measurement refers to the SMS module where the computed data is stored. (The Module is set by default and normally does not need to be changed.)

Each observation arc is assigned the following attributes:

- **Color** – Color of the observation arc.
- **Observe** – Turn the observation arc on or off.
- **Name** – Name of the observation arc.
- **x-origin** – X origin of the arc (used to modify the x-value used in plots).
- **Observed Value** – Value measured in the field corresponding to the active measurement.
- **Interval** – Allowable error (±) between the computed value and the observed value. Model verification is achieved when the error is within the interval (±) of the observed value.

Related Topics

- Coverages
- Plot Window

Particle/Drogue

A particle/drogue coverage is used for Visualization post-processing. The feature points and arc nodes/vertices define the seed locations for generating an animation of particles flowing through a hydrodynamic current. The particles are simulated as massless objects or "drogues" floating in the flow field. SMS computes the paths these particles would follow when driven by the currents of the flow field using numerical integration.

This coverage must be defined before selecting the drogue option when setting up a film loop.

When displaying the resulting animation, the particles may be displayed in a color based on the current velocity of the particle or the distance the particle has traveled. The user also specifies the length of the tail behind the particle (in units of time). Therefore, a longer particle tail indicates a generally faster moving particle.

Drogue plot animations are different from flowtrace animations in that the distances traveled by each drogue represents the actual physical speed of the flow field.

Application

- Residence time calculations. One of the most useful applications of drogue plots is to approximate residence time of a basin or other subregion of a hydrodynamic domain. In order to approximate this, the user would:
  - Distribute a fairly large number of drogue seed points inside the basin (or sub domain) of interest).
  - Generate a drogue plot film loop of the time range of interest. (Currently the hydrodynamic solution must include enough time steps to span the residence time in question. If this is not the case, additional time steps should be added to the data set either by rerunning the simulation for a longer time period or duplicating time steps).
  - Review the number of particles still in the basin (or subdomain) at various time intervals. The percentage of particles, and their position give an indication of overall concentration and local concentration in the basin.

- General visualization of flow field.

External Links:

- “Gas Flow Visualization for Combustion Analysis”, Energy and Fuels, Vol. 7, No. 6, 1993, pp. 891-896. Zundel, A.K., Saito, T., Owen, S.J., Sederberg, T.W., Christiansen, H.N. [1]

Related Topics

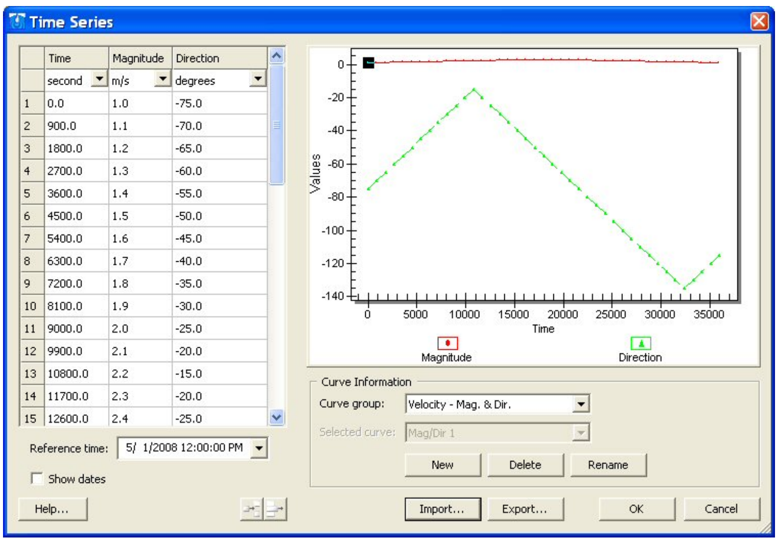
- Coverages
- Animations

Spatial Data

The Spatial Data coverage is designed to store, visualize, and analyze various types of data at node locations. Most commonly, this data would consist of time series curves. The data can be accessed or added to a node by the right-click menu of a single selected node. From this menu the data associated with the node can be created, viewed, modified, or deleted.

The following types of data are supported (right-click options are described):

Time Series – The *time series editor* allows the user to view, edit and import/export transient datasets for the location. Each time series consists of either a scalar value at each time, or a vector value at each time. The vector quantities may be defined as (X,Y)components or ((Magnitude,Direction) pairs. The time values may be displayed as dates or offset values. The time series are stored in a database inside SMS. When the project is saved, the series are saved in the XMDF project file. The series may also be imported/exported using a TSD file format..



Time Series Editor'

- *Edit Data* – This menu option invokes the *time series editor*. The list of time series curves available for this node are listed in a list box. The user selects the desired time series curve, and the values for that curve appear in the spread sheet. The selected time series data may be modified in this spread sheet.
- *View Data* – This menu option operates like the *Edit Data* option, but the spread sheet is set to read only. This prevents accidental modification of the values in the time series.
- *Delete* – This option removes the association between the selected time series and the spatial data node. The time series curve remains accessible in the Time Series database.

- **Compass Plot**
  - *Show Connection Lines* – This option allows the lines connecting the compass plot to its associated spatial data node to be enabled or disabled.
  - *Properties...* – This option brings up the *Compass Plot Properties* dialog for the selected compass plot.

- *Delete* – This option deletes the selected compass plot.

The Spatial Data coverage type is found in the "Generic" coverage type list.

Compass Plot

Compass Plots may be created on a *Spatial Data* node. The *Compass Plot* displays arrows to represent temporally varying vector data. This plot can be used to show a vector quantity, either varying through time or constant, to illustrate quantities such as wind direction, wave direction, or current direction. The plot is associated with a specific point, but that point does not have to be part of a numerical model or physical object. It could be created simply to hold the prevailing wind direction, for which a compass plot would be created.

Direction Convention

For curves with a specified direction, the compass plot uses a "TO" direction convention with North being 0.0 and the angle is measured clockwise. This means that a vector with a direction of 0.0 points North, 90 degrees points East, 180 degrees points South and 270 degrees points West.

Layout

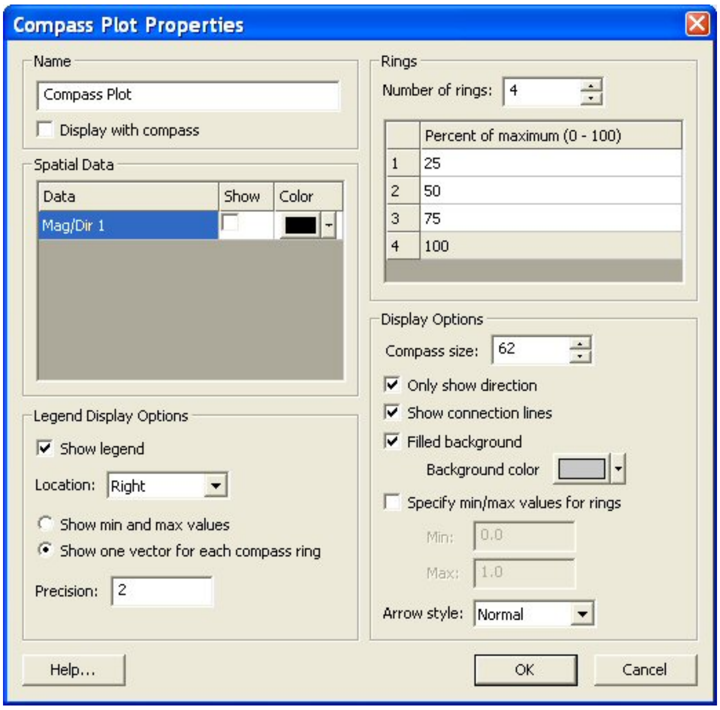
When a plot is created, SMS places on the left side of the screen. The user may select the plot using the **Select Compass Plot** tool  and position it at any location. It is positioned in screen space, so as the user pans or zooms around the modeling domain, the plot stays in a single location. The plot can also be resized graphically or using its attributes dialog.

Creating a Compass Plot

In order to create an use a compass plot, a user will perform the following steps:

1. Create or Select a *Spatial Data* Coverage.
2. Create or Select a *Feature Point* in the *Spatial Data Coverage*.
3. Make sure at least one vector time series curve is stored for the selected feature point.
4. Right-click on the point and select the **Add → Compass Plot** command. This creates a compass plot and brings ups the *Compass Plot Properties* dialog. Each time series to be included in the compass plot must be selected in the *Spatial Data* section of the dialog. Clicking **OK** will cause the dialog to disappear and the compass plot to appear. Properties of the compass plot include:
   - The name of the plot which can optionally be displayed at the top of the compass plot.

- A flag to show/not show a vector for each vector time series stored at the *Spatial Data* point.
- Options for a compass plot legend, including:
  - A flag to show/not show the legend.
  - Set the position of the legend. Options include any side of the plot.
  - The number of vectors to show in the legend. This can be the min/max, or one for each compass ring.
  - The number of digits of precision for the legend.
- Control of the number of rings to show in the plot, and the percent of maximum value for each ring. By default SMS creates the compass plot with four concentric circles, representing 1/4, 1/2, 3/4 and the maximum vector magnitude.



Compass Plot Properties

- Display options including:
  - The pixel size of the compass plot.
  - A flag to show only the vector direction (ignore magnitude).
  - A flag to show connection lines. Since the plot can be selected and drug to any position on the screen, these lines can be useful to show a location the vectors apply to.
  - A flag to set the background color (if desired) for the compass plot. By default, the plot is filled with a gray background.
  - An option to specify the magnitude range to display. Any vector with magnitude above the maximum will appear as a 100% magnitude vector. Below the minimum, the vector will not be displayed.
    - The vector style. (This is a future enhancement. Currently, only "Normal" is supported.

Editing a Compass Plot

In order to edit or adjust a compass plot, a user will perform the following steps:

1. Select the **Select Compass Plot** tool  .
2. Left-click on the selection box of a compass plot and drag to position and resize the plot.
3. Right-click on the plot itself. I menu appears including the following options:

- Delete – This allows the compass plot to be deleted
- Show Connection Lines – this sets the display option of the plot
- Show Legend – this turns on/off the legend for the compass plot
- Legend Location – this is a pull right menu that allows the user to position the legend
- Properties – this invokes the compass plot properties dialog to allow the user to edit any of the attributes.

Related Topics

- Time Series
- Coverages

Spectral Coverage

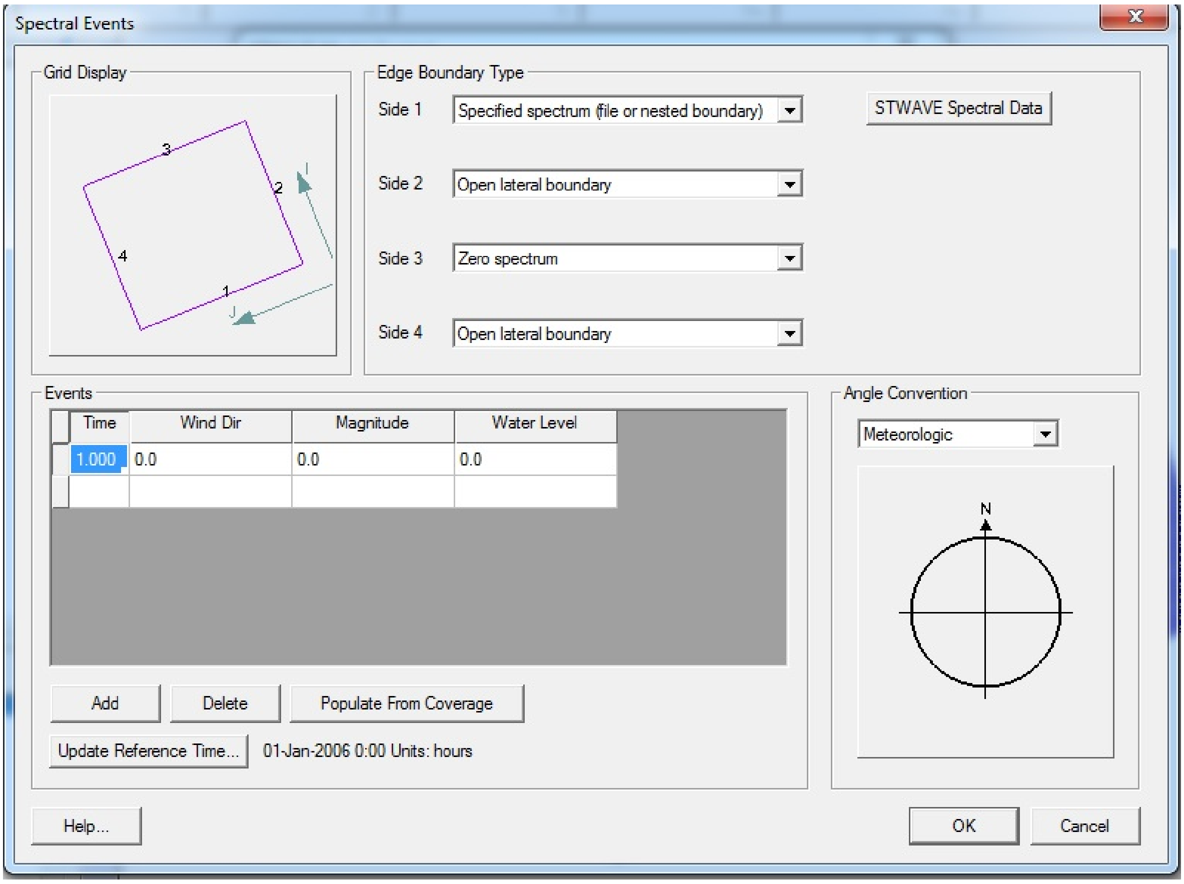
Spatial Varied Boundary Conditions

CMS-Wave and STWAVE have the ability to read in spectral data from various locations defined in a nesting file. Within SMS, this spectral data is defined using a spectral coverage. Each node in this coverage can be assigned to any number of spectral grids and datasets which define the conditions at that location at any specific time.

Creating Spectral Data

To create spectral data at a node in the spectral coverage, right-click on the node and select **Node Attributes...**. This will bring up the *Spectral Energy* dialog, from which spectral grids and spectra can be created. See Generate/Edit Spectra.

Creating a Spectral Event



Spectral Event dialog

Related Topics

- Spectral Energy

3.7.a.2. Model Specific Coverages

Model Specific Coverages

Specific model coverage can be selected by right-clicking on the map module data and selecting *Type* | **Models**.

- 1D Hyd Centerline Coverage
- 1D Hyd Cross-Section Coverage
- 1D River Conceptual Model
- ADCIRC
- ADCIRC Wind Coverage
- ADH
- BOUSS2D
- CGWAVE
- CMS-Flow
- CMS-Wave
- CSHORE
- EFDC
- ESMF
- FESWMS
- GenCade
- Generic Mesh Coverage
- Generic Cartesian Grid Coverage
- PTM
- SED-ZLJ
- STWAVE
- TABS (RMA2/RMA4)
- TUFLOW Coverages
- WAM

Related Topics

- Map Module
- Generic Coverages

1D Hyd Cross-Section Coverage

The 1D-Hydraulic Cross-section coverage is used to identify the cross section stations in the hydraulic model, and can also be used to automatically cut a cross section from an underlying digital terrain model. The attributes of a cross-section feature arc is the cross-section itself, along with the other parameters that define its topology in the model and include: a cross-section ID (internally assigned), the reach name (inherited from the centerline arc it intersects), the station (inherited from the centerline), and any specific model attributes. The 1D-Hydraulic coverage is used in conjunction with the cross sections and digital terrain model in order to determine the thalweg position (from the centerline arc) and the left and right bank points (from the bank arcs).

A cross-section is assigned automatically when cutting the cross sections, or can be assigned manually (imported from a file or entered directly) using the cross section editor.

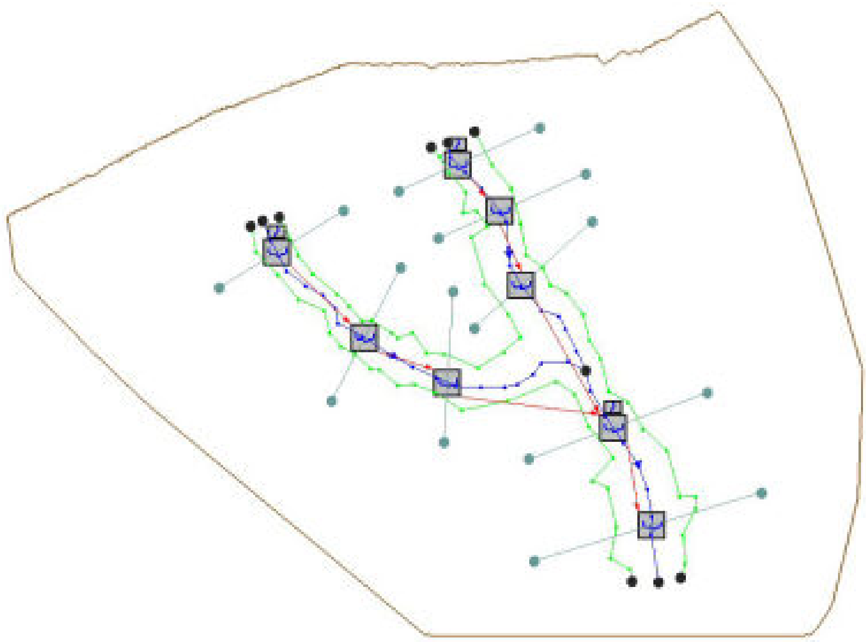
See the help for Editing Cross Sections to learn more about how cross sections are managed and edited.

Related Topics

- 1D River Conceptual Model

1D River Conceptual Model

SMS uses a conceptual model (coverages of centerlines, and cross-sections) to define the hydraulic model, but at some point this conceptual model must be mapped to an equivalent topologic model representation for a Hydraulic model. HEC-RAS, for example, is defined as reaches and cross sections. Each reach and section has appropriate stationing defined from the spatial nature of the conceptual model. When choosing this command a schematic of the river is made for the appropriate model. The example below shows the reach (small boxes at the beginning of each reach) and section icons of the schematic for an HEC-RAS model. In the Hydraulic Modeling module the schematic is generally used as the view of the model.



Related Topics

- 1D Hyd Cross-Section Coverage
- 1D River Module

ADCIRC

An ADCIRC coverage is used to build a conceptual model of an ADCIRC project. The conceptual model defines parameters such as model extents, mesh generation options, and boundary conditions.

ADCIRC Conceptual Model Development

The following steps are generally followed when creating an ADCIRC conceptual model:

Define Coastline

The coastline can be defined in any of the following manners:

- Read in an existing coastline file (*.cst) (see Create Coastline). Coastline files include lists of two-dimensional polylines that may be closed or open. Open polylines are converted to Feature Arcs and are interpreted as open sections of coastline. Closed polylines are converted to arcs and are assigned the attributes of islands.
- Extract a coastline arc from a scatter set using the **Scatter Contour to Feature** command.
- Digitize a coastline arc using the **Create Feature Arc** tool.

Define Ocean Boundary

Once the coastline has been created, the ocean boundary can be defined in any of the following manners:

- Use the **Define Domain** menu command to automatically generate the ocean boundary.
- Digitize the ocean boundary arc using the **Create Feature Arc** tool.

The ocean boundary can take on a rectangular, semi-circular, or circular shape depending on your coastline form. This will close the domain for your project, giving you a defined area where a finite mesh can be created and the ADCIRC model can perform its analysis.

Build Polygons

Build polygons using the **Build Polygons** menu command found in the *Feature Objects* menu.

Choose Mesh Generation Method

At this point, a choice must be made to generate the mesh using the LTEA Toolbox or manual mesh generation methods.

LTEA Toolbox

The LTEA Toolbox can also be used to generate a mesh from a bathymetry scatter set and the ADCIRC coverage created in the previous steps.

Manual Mesh Generation

To manually generate a mesh:

- Use the **Select Feature Polygon** tool and double-click on a polygon to open the *2D Mesh Polygon Properties* dialog. You can also use select a polygon and then select **Attributes** from the right-click menu or select a polygon and use the **Attributes** menu command to open the *2D Mesh Polygon Properties* dialog.
- Set the desired mesh options. See the article on mesh generation for an explanation of mesh generation.

Related Topics

- Define Domain
- Coverages
- ADCIRC
- Boundary Conditions
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Spatial Attributes
- Steering

ADCIRC Wind Coverage

The ADCIRC wind coverage represents a storm, such as a tropical depression or hurricane. This information can be imported from NWS 8 and 9 files (fort.22 and fort.15 files), and exported by SMS for use with ADCIRC. Running ADCIRC directly from SMS also works fine, and various model checks have been implemented to help users catch mistakes and avoid crossing ADCIRC's limitations.

To create a wind coverage, create a new coverage and select the coverage type *Models* | *ADCIRC* | **Wind**. Alternately, you can convert an existing coverage by right-clicking it and selecting *Type* | *Models* | *ADCIRC* | **Wind**.

Coverage Attributes

To access the *Coverage Attributes*, right-click on the coverage and select **Properties...**. The *Coverage Attributes* dialog contains several separate fields that apply to the entire hurricane, such as whether it is symmetric or asymmetric. The storm's symmetry in particular is important to set before editing node properties because it affects which fields are shown by default and regarded as required.

- Wind Model: Choose between symmetric and asymmetric definition of your storm. The wind model will determine which fields are displayed and which are hidden (by default) in the node attributes dialog.
  - Symmetric is the basic and default choice, and assumes a simple storm definition will suffice.
  - Asymmetric gives you a lot more options for defining the storm's shape and orientation.
  - PBL (Planetary Boundary Layer) model isn't currently available.
- Basin and Subregion: For informational purposes, these define where in the world the storm is taking place.
- Annual cyclone number: Does not affect calculations, but is valuable for book keeping.

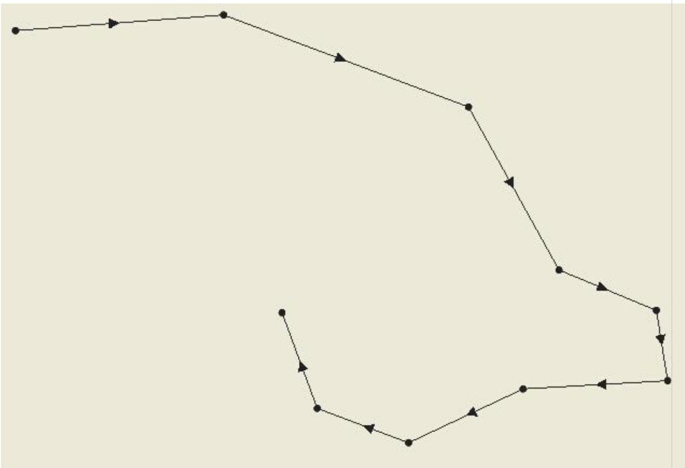
Building a Storm Path

The storm path should be a single continuous line with no breaks or branches. Simply clicking out an arc is sufficient—vertices will be converted to nodes once you enter the *Node Attributes* dialog. Do:

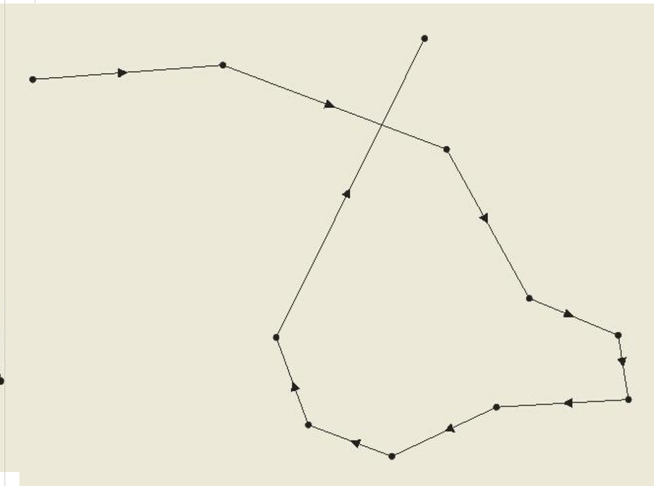
- Operate in Geographic Coordinates
- Make a single path with no breaks
- Create the number of vertices/nodes that you have data for (you can adjust their positions manually with the nodes/vertices selected, or you can set their positions within the *Node Attributes* dialog)

Don't:

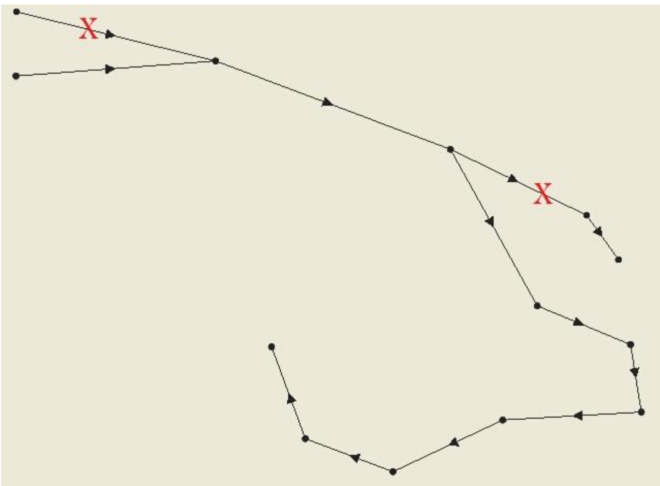
- Create multiple paths in the same coverage
- Split the path in multiple directions
- Create loops with the path



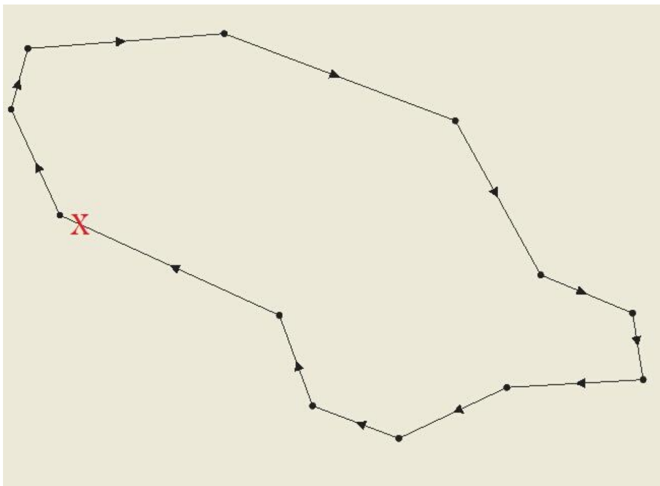
Good example of a storm path



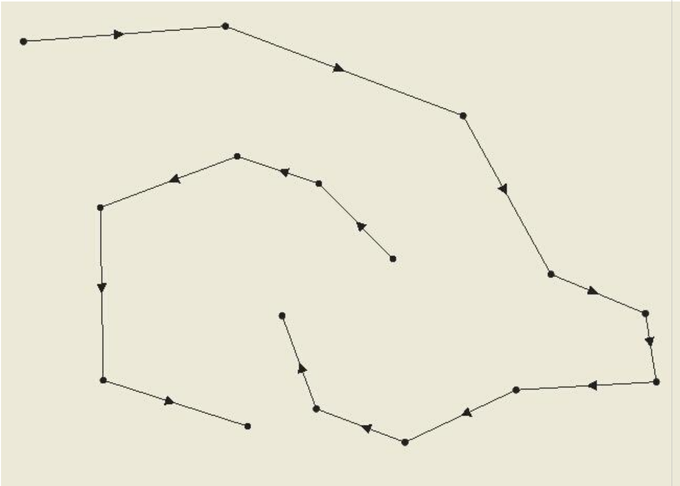
A good path can cross itself as long as it does not connect at the intersection



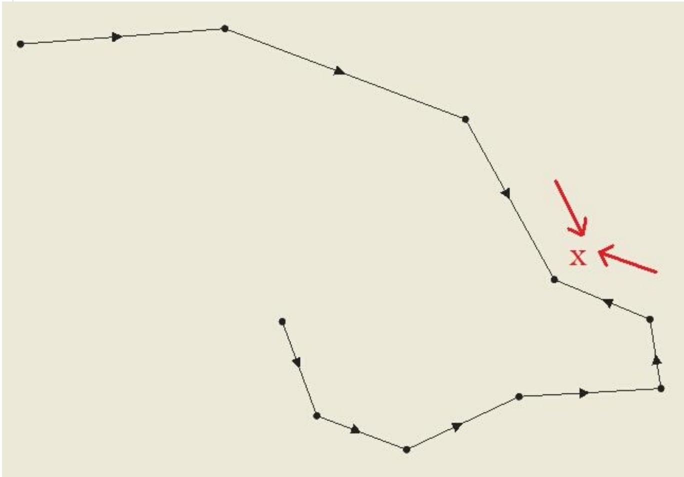
Path should not branch as in this picture



Path should never loop as shown



Only one path can exist. Multiple paths as shown are not allowed.



Different portions of the path cannot converge on each other. The arrows for each section of the path should all point in the same direction for the whole length of the path.

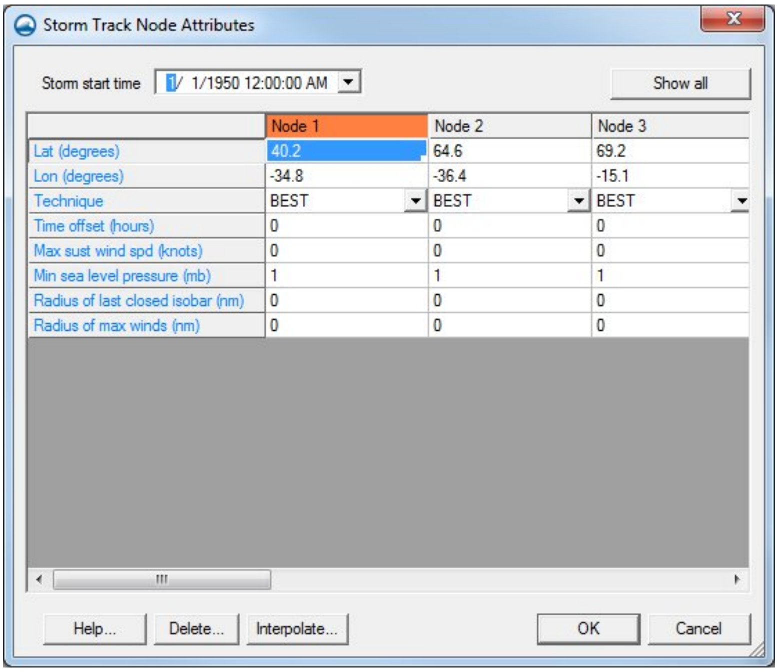
Notes

Consequently, the storm path and nodes information may be obtained in a hurdat file obtained on the NOAA website [1]. Access website and save the Easy to Read version of the file. Extension for file must be saved as hurdat in order to open in SMS.

Node Attributes

Once you've built a storm path, you can define the storm's attributes at each node of the path. Enter the *Node Attributes* dialog by selecting the **Select Feature Point** tool and double-clicking anywhere in the coverage. This opens the *Storm Track Node Attributes* dialog. Whenever you open this dialog, all vertices in the storm path are converted to nodes automatically.

A second way to open the *Node Attributes* dialog is to use the **Select Feature Point** tool to select one or more nodes on the path. Then right-click and select **Node Attributes...**. This will highlight those nodes in the dialog.



Storm Track Node Attributes dialog

Fields colored blue and displayed by default are those that are required for the Wind Model (symmetric or asymmetric) selected in the *Coverage Attributes* dialog. The **Show all/Show only required** button can be used to show all the fields available, even those not used by ADCIRC or for the selected wind model. These are useful for book keeping and completeness, even though they typically do not affect calculations.

The Storm start time sets the starting time for the first node in the storm's path. Each node then defines an offset from this starting time in hours (see below). Year, month, day and hour are important, while minutes and seconds should be left at 0.

The fields in the spreadsheet more or less correspond directly to a field in the fort.22 file [2]:

- Lat and Lon: These define the latitude and longitude of the given node, in tenths of degrees (900 = 90 degrees). You can edit these values directly from the dialog, or you can select the nodes with the **Select Feature Point** tool and edit their X and Y that way. In this dialog, values are always positive and N/S/E/W determines quadrant, whereas the main SMS interface uses negative numbers for South and East.
- TechNum/Minutes (TECHNUM/MM)
- Technique (TECH): ADCIRC recommends that this be set to ASYM when dealing with asymmetric storms
- Time offset (YYYYMMDDHH and TAU): This field combined with the Storm start time above the spreadsheet determine the YYYYMMDDHH and TAU fields in the fort.22. This field is the offset (in hours) from the storm start time.
- Max sust wind spd (VMAX)
- Minimum sea lvl pressure (MSLP)
- Lvl of tc development (TY)
- Wind radius code (WINDCODE): ADCIRC requires that this be full circle for symmetric, and northeast quadrant for asymmetric. This is fairly restrictive, but SMS can convert many of the other options to northeast quadrant automatically. A model check will warn you if some of your selections cannot be converted without losing data.
- Wind Intensity (RAD, RAD1-4): Each node can store wind intensity and radii for the storm shape at 34, 50, 64 and 100 kts.
- Pressure of last closed isobar (RADP)
- Radius of last closed isobar (RRP)
- Radius of max winds (MRD)
- Gusts (GUSTS)
- Eye diameter (EYE)
- Max seas (MAXSEAS)
- Forcaster's initials (INITIALS)
- Storm direction (DIR)
- Storm speed (SPEED)
- Storm name (STORMNAME)
- System depth (DEPTH)
- Wave height for radii (SEAS)
- Seas radius code (SEASCODE)
- Wave height radius 1-4 (SEAS1-4)

Linking the Coverage to the ADCIRC Project

Once you've defined your storm path and all its data, it's time to link the project into ADCIRC. To do this, select the ADCIRC mesh and go to *ADCIRC* | **Model Control**. Choose the *Wind* tab. Select either Dynamic Holland Model (NWS=8) or Asymmetric vortex, Holland gradient wind model (NWS=9). In the *Wind File Options* section, click **Choose coverage...** to select your coverage and link it in. The **Options** button will now open the *Coverage Attributes* dialog, allowing you to edit it quickly from this dialog.

In the *Timing* tab be sure to set up the simulation start time and how long it runs. The wind coverage's time span should have the same start time and duration, or be longer so that it encompasses the simulation time span.

Also set up any other ADCIRC settings as needed. When finished, click **OK**, then go to *File* | **Save ADCIRC**, then *ADCIRC* | **Run ADCIRC**. The model check will alert you to any potential problems before ADCIRC runs.

ADH

The ADH Vessel Coverage is used to add vessels to the simulation and give them paths to follow. Generally, one coverage represents one vessel. You can drag one or more vessel coverages into the ADH Mesh to add those coverages to the simulation (by creating links). This allows you to have more vessels defined than you are using in your simulation, and you can swap them in and out of the simulation to run different tests with different vessels.

Boat Path

Each vessel coverage has at least one arc to define the boat path. The arc determines the vessel's starting position and starting speed, and where it will go from there. Additional segments can be added to the arc by adding vertices. Vertices add destinations but do not affect speed. To change the speed of the boat you can convert a vertice into a node, and enter the node's attributes.

In general, there will always be exactly one path per vessel coverage. The path can be made up of multiple arcs, but the arcs should not split into multiple paths. The one exception to the rule of "one path per coverage" is when you have multiple vessels that are identical except for their speeds & paths. In this case you can have multiple separate paths in the same coverage, and each will create its own separate copy of the boat defined in the coverage properties. These paths can cross each other as long as they do not connect to each other (at a node).

Another option is to simply duplicate the coverage after the boat properties are defined. This allows you to add and remove the boats from the simulation separately, and the boats can have vertices and nodes in the same place (same x and y coord) along their path without conflicting.

The boat path writes the FDEF and SDEF cards to the boat file.

Dialogs

Node Attributes

Nodes on arcs change the speed of the boat or, in the case of the first node of the path, define the boat's starting speed. To change the speed of the boat at a node, select it and right-click, then select **Node Attributes...**. Alternately, you can double-click the node.

Boat Parameters

The boat's parameters are defined in the coverage properties. Right-click on the coverage and choose **Properties...**. This dialog lets you set the boat's size and shape, and define propellers if desired. If the OP BTS card is included, each vessel in the simulation will need to have propellers defined. Without the OP BTS card, propellers supposedly do nothing.

Each field corresponds directly to a card in the boat file.

- Length (BLEN)
- Width (BWID)
- Bow to Length Ratio (PBOW)
- Stern to Length Ratio (PSTR)
- Draft (DRFT)
- Fraction of Draft Applied to PBOW (CBOW)

- Fraction of Draft Applied to PSTR (CSTR)

Propeller (PROP card):

- Propeller Type: Open wheel or Kort nozzle
- Propeller Diameter
- Distance between propellers
- Tow boat length: This length provides an offset distance of the propeller induced shear stresses from the vessel. Set to 0 if there is no tow boat.
- Distance from prop to tow boat stern

Related Topics

- ADH
- Coverages

EFDC

The EFDC coverage is used to create curvilinear grids for use with the EFDC model (EFDC grid format). A curvilinear grid has I and J coordinates but the cells do not have to follow straight lines like rectilinear grids see Curvilinear Grid Module.

The coverage doesn't contain any point or arc attributes.

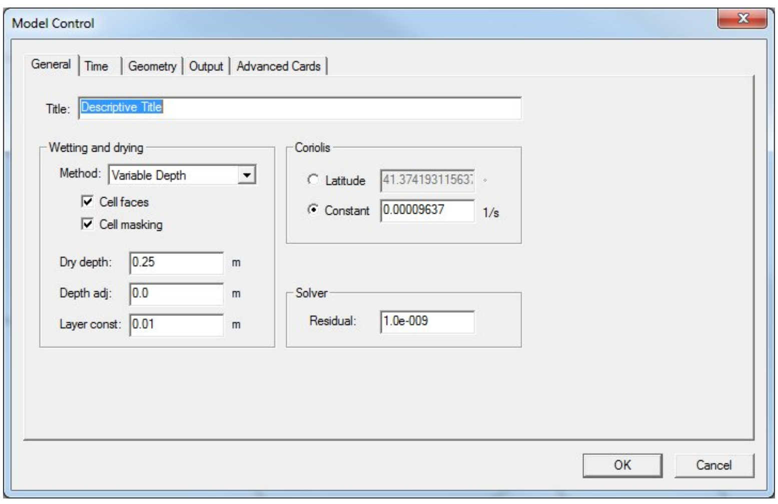
The EFDC coverages uses the same meshing tools as other SMS coverages but with limited options. The only supported meshing option is rectangular patches. Patches cannot have transitions (all opposing sides must have the same number of vertices). See Mesh Generation for more information about the meshing options available in SMS.

EFDC Model Control

The *EFDC Model Control* dialog is accessed using the *EFDC* | **Model Control...** menu item.

Dialog Description

The EFDC Model control contains the following pages:



EDFC *Model Control* dialog showing the *General* tab

General

The *General* tab of the EFDC *model control* is used to specify general simulation parameters. It contains the following controls:

Title

Text description up to 80 characters in length for the run. EFDC.inp card 1.1

Wetting and drying

- **Method** – Wetting and drying method. EFDC.inp card 5.5
  - **None**

- **Constant Depth**
  - **Nonlinear iteration**
  - **Cell masking**
- **Variable Depth**
  - **Cell masking**
  - **Cell faces**

- **Dry depth –** Depth at which cell or flow face becomes dry. EFDC.inp card 11.10
- **Depth adj –** Adjustment to bottom bed elevation field in meters. EFDC.inp card 11.12
- **Layer const –** Log bdry layer const or variable rough height adj in meters. EFDC.inp card 11.5

Coriolis

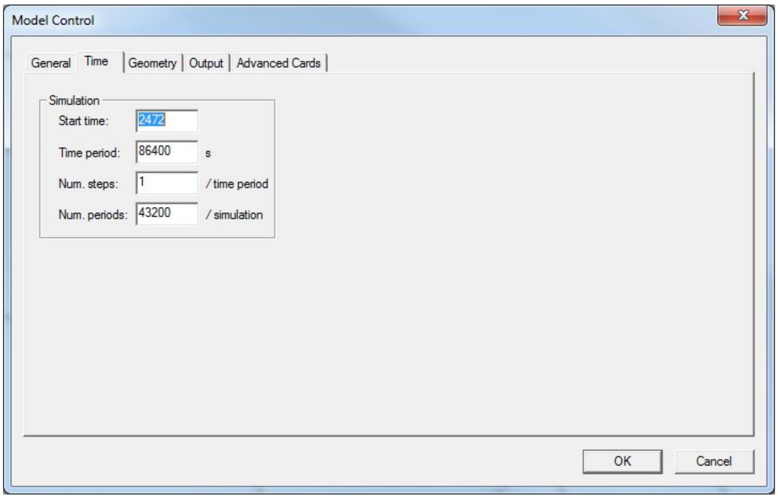
- **Latitude –** The coriolis parameter will be calculated by SMS from the specified latitude value. EFDC.inp card 8.4
- **Constant –** Constant coriolis parameter in 1/sec. EFDC.inp card 8.4

Solver

- **Residual –** Target square residual of iterative solution scheme. EFDC.inp card 3.2

Time

The *Time* tab of the EFDC *model control* is used to specify simulation time parameters. It contains the following controls:



EFDC *Model Control* dialog showing the *Time* tab

Simulation

- **Start time –** Time origin of the run. EFDC.inp card 8.2
- **Time period –** Reference time period in seconds. EFDC.inp card 8.3
- **Num. steps –** Number of time steps per reference time period. EFDC.inp card 7.2
- **Num. periods –** Number of reference time periods in run. EFDC.inp card 7.1

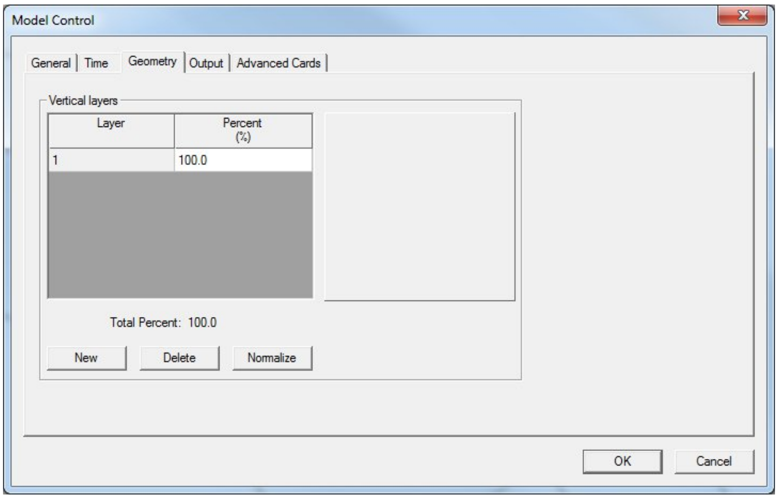
Geometry

The *Geometry* tab of the EFDC *model control* is used to specify geometry related parameters for the simulation. It contains the following controls:

Vertical Layers

The number of layers specified in the vertical layers spreadsheet is used for the value of EFDC.inp card 9.1.

- **Layer –** Layer number. Use the **New** and **Delete** buttons to create / remove layers. EFDC.inp card 10.1
- **Percent –** Dimensionless layer thickness. Must sum to 100%. EFDC.inp card 10.2



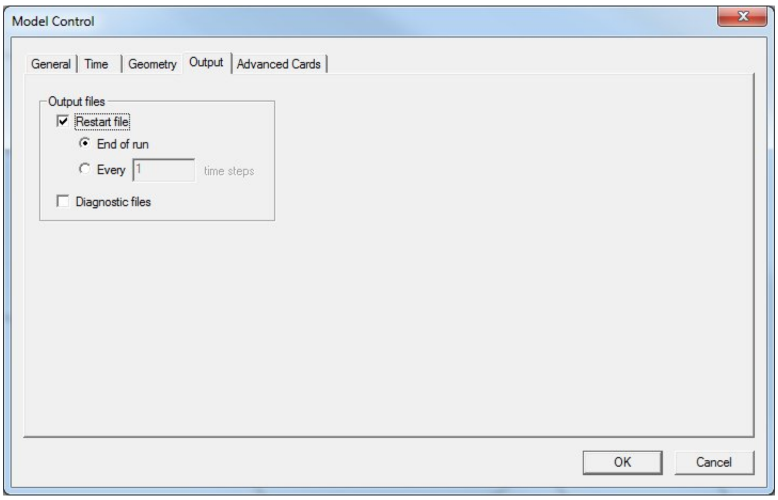
EFDC *Model Control* dialog showing the *Geometry* tab

Output

The *Output* tab of the EFDC *Model Control* is used to specify the output the model will create. It contains the following controls:

Output Files

- **Restart file**
  - **End of run –** Restart file written at the end of the run. EFDC.inp card 2.2
  - **Every <> time steps –** Restart file written every <> time periods. EFDC.inp card 2.2
  - **Diagnostic files –** Write diagnostics files for external mode solver. EFDC.inp card 3.10



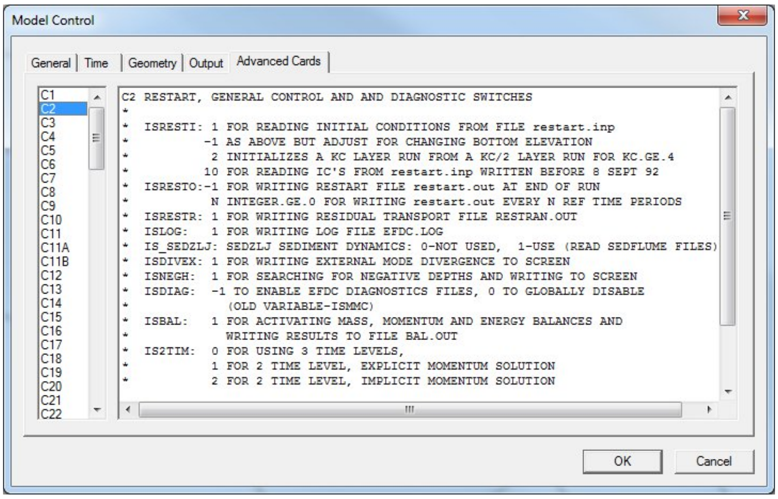
EFDC *Model Control* dialog showing the *Output* tab

Advanced Cards

The *Advanced Cards* tab of the EFDC *model control* is used to modify advanced simulation parameters not supported directly in the *model control*.

Dialog Description

The text blocks associated with a given card can be edited directly in the dialog. A list of the cards available to edit is shown on the left of the dialog. The fields corresponding to the card parameters directly supported by SMS will be overwritten when the dialog is exited.



EFDC *Model Control* dialog showing the *Advanced Cards* tab

ESMF – Earth System Modeling Framework

The Earth System Modeling Framework (ESMF) [1] is used to couple the following Models:

- ADCIRC – STWAVE

To create a coupled simulation using ESMF, perform the following steps:

- Right-click in the Project Explorer
- Choose the menu *New Simulation* | **Hurricane**
- Change the simulation name if desired
- Create the elements to be include in the simulation and drag tree pointers representing them into the simulation. The following items can be linked to an Hurricane simulation:
- ADCIRC – STWAVE
  - ADCIRC Mesh
  - STWAVE Grid
  - Hurricane Coverage
  - PBL Wind Coverage
  - WAM Simulation
  - Right-click on the Hurricane simulation in the Project Explorer
  - Choose the menu **Properties**. This action will create a new Hurricane coverage containing polygons that identify overlapping grid sections and areas of interest defined by the user. It will then bring up a *Hurricane Mode Project Summary* property sheet containing four tabbed dialogs.
    - Overview

- ESMF
- Spatial View
- Timeline
- Right-click on the ESMF simulation and choose **Export ESMF Files**. SMS will create a folder named ESMF in the same directory as the .sms file of the current project. Inside of the ESMF folder will be a folder and input files for the ESMF Simulation. The ESMF Simulation folder name will be the same as the name given to the ESMF Simulations in the *Project Explorer*.

ESMF Hurricane Overview

The dialog gives an overview of the components of the Hurricane simulation. It also allows the user to designate the number of processors to be use by each model in the simulation.

- Simulation Name – Name of the Hurricane simulation as set in the project explore
- Properties – Properties of the Hurricane simulation
  - Model Name – Name of models found in the simulation
    - ADCIRC <mesh name> (1 required) – Name of the adcirc mesh used in the hurricane simulation.
    - STWAVE <grid name> (1 required) – Name of the stwave cgrid used in the hurricane simulation.
    - PBL <pbl cov name> (optional) – Name of pbl wind coverage used in the hurricane simulation.
    - WAM <wam sim name> (optional) – Name of wam simulation used in the hurricane simulation.
    - Projection – Projection type of model.
      - Geo – Geographic (Latitude/Longitude)
      - STPL <#> – State Plane number
      - Other – Other type of projection
    - Start Time – Temporal starting time of model.
    - End Time – Temporal ending time of model.
    - Threads – Number of processor threads used by model for computation or I/O processing.
    - Function – Function used to adjust the processor threads used by each model.
      - Set Threads – Set the number of processor threads for:
        - Computational or I/O processing (ADCIRC)
        - Grid Partition I and J processing (STWAVE)
        - NONE – WAM and PBL use only one processor thread.

Hurricane ESMF

- Set the ESMF simulation options in the *Data Exchange* and *Area Mapping* sections of the dialog
  - Data Exchange
    - Model A – The meshed based model to be use (hard-coded to ADCIRC)
    - Data Exchange – Controls how data is exchanged between the two models (↔ bi-directional, → uni-direction A to B, ← uni-directional B to A)
    - Model B – The grid based model to be use (hard-coded to STWAVE)
    - Model A → B – How frequently the results of Model A are passed to Model B
    - Units – Frequency units for A to B exchange (days/hours/minutes/seconds)
    - Model A ← B – How frequently the results of Model B are passed to Model A
    - Units – Frequency units for B to A exchange (days/hours/minutes/seconds)
    - Area Mapping
      - ID – Polygon identifier correlated to the ids visible in the main graphics window.

- Mapping
  - Single – Model A exchanges data with a single instance of Model B
  - Combined – Model A exchanges data with multiple instances of Model B. Model A can receive either the average or maximum values from the instances of Model B.
- Option – If the "Mapping" field is set to "Single" and multiple grids overlap the identified polygon, this field allows the user to select which grid will be used. If the "Mapping" field is set to "Combined", this field allows the user to select how the data from the grids will be combined (Average or Maximum.)

Hurricane Spatial View

This dialog provides a view of any grid boundaries (WAM and STWAVE), grid frames (PBL), and the ADCIRC mesh boundary associated with the simulation.

Hurricane Timeline

This dialog displays the timelines for each simulation (WAM, STWAVE, PBL, and ADCIRC) that are part of the hurricane simulation. The start and end times need to be set for each simulation individually.

Related Topics

- Model Specific Coverages
- CSTORM-MS
- Steering

Generic Mesh Coverage

Generic 2D mesh coverage

Generic mesh coverage is for a Generic mesh model. The generic model interface spans both the map and the mesh modules. You can create a Generic Model coverage and assign arc boundary conditions based on the types defined in the generic model template. You can also assign attributes to feature points. More information about a meshing coverage can be found under the article Coverages.

Convert Map feature arc and point attributes (Generic 2D mesh coverage) to mesh nodestring and node attributes

Users in the map module with a Generic 2D mesh coverage can create feature points and arcs. These points and arcs can then be assigned attributes. This is done by double clicking on either the feature point or feature arc. For points a dialog will appear allowing you to assign node or element boundary conditions. For arcs you will be assign attributes on the arc. When doing a Map→2D Mesh, point node boundary conditions will be assigned to the nearest node. Point element conditions will be assigned to the nearest element. Mesh nodestrings will be created and assigned from the nodes nearest the feature arc.

Convert a Map to 2D Mesh

To convert a map to a mesh, right-click on the map default or active coverage. Then select *Convert* | **Map→2D Mesh**. A dialog is displayed. If a mesh already exists, you may choose to delete it, or map the attributes to the existing mesh.

Generic Cartesian Grid Coverage

SMS provides the option to work in a Generic Cartesian Grid coverage. Selecting this coverage allows the user to use the **Create 2D Grid Frame** tool. After creating a grid frame in the map module, a user can use the **Map → 2D Grid** command.

The Generic Cartesian Grid Coverage provides a limited number of tools and options in the Cartesian Grid Module.

Related Topics

- Model Specific Coverages

Synthetic Storm Coverage

Background

When simulating an actual storm (hindcast), the storm information such as the track locations, central pressures, radius information, speeds, and Holland B values can come from an analysis on data collected during the storm. However, if our simulation is intended for design analysis you may not want to choose a storm that has happened but a storm which may happen. Often several configurations for storms would be analyzed to see the results of each. The synthetic storm coverage and associated generator executable provide a mechanism for creating a PBL coverage (trop file) based upon user specified parameters.

The first step is to decide upon a track path or multiple paths to simulate. For multiple paths, one option is to create a single track and use the perturbation tools in SMS to generate similar tracks by using offsets and modifications of central pressures, etc.

Once your track locations have been defined, we need to define the associated data for the track (central pressures, Holland B, storm radius, etc.). The US Army Engineer Research and Development Center has developed a utility that creates a full PBL input trop file from a small set of user defined parameters.

Inputs

The following parameters are used to fill in the track data:

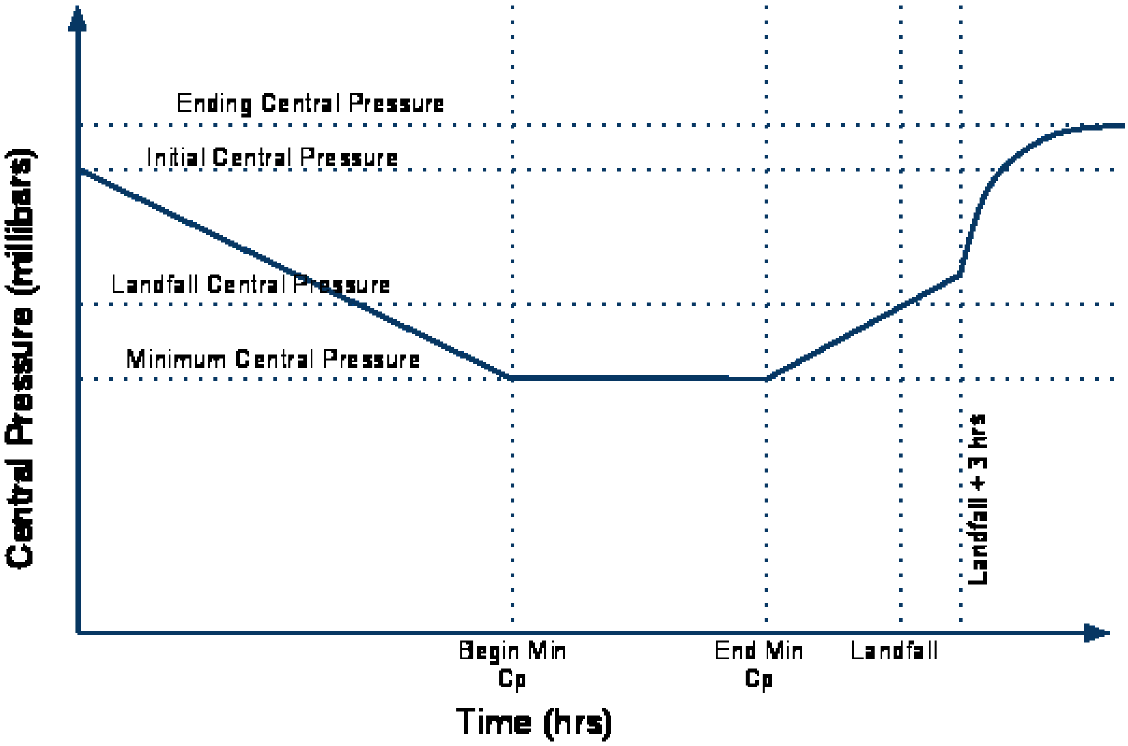
- Storm name
- Storm number
- Starting date/time (nearest hour)
- Forward speed in knots
- Far field atmospheric pressure in millibars
- Initial central pressure in millibars
- Minimum central pressure in millibars
- Landfall central pressure in millibars
- Initial storm radius in miles

- Landfall storm radius in miles
- Initial Holland B value

In addition to these parameters the locations where minimum central pressure exists along the track and the landfall location are specified.

Methodology

ERDC has developed curves to represent general behavior of storms. These curves are a simplistic approach and should be used with caution. When in doubt if you are providing appropriate wind fields consult your favorite meteorologist. Sample curves for central pressure, Holland B, and storm radius can be seen below. Some of the values shown are user supplied and others are determined by the synthetic storm generator (like ending radius).



User Interface

Synthetic storms are created in SMS using the "synthetic storm coverage." This coverage is in the folder "\Models\Wind" when accessing it from the new coverage dialog or changing the type of an existing coverage.

The coverage must contain a single track made up of multiple arcs. The locations of starting minimum central pressure, ending minimum central pressure, and landfall are specified on arc end nodes. You can change the type of an arc node by selecting the node, right clicking, and choosing the desired type from the "Type" submenu. When you change a node to any type except for generic and a node of that type already exists, the existing node will be changed to a generic node. There can only be one node of each "non-generic" type.

The parameters for the storm generator are stored with the coverage and can be accessed by right-clicking on the coverage and choosing "properties."

When the track has been defined, the node types specified, and the properties assigned to the coverage the PBL coverage can be created. To create the PBL coverage, right-click on the coverage and choose "Create PBL coverage." This will run the ERDC storm generator utility and read the resulting data into a new PBL coverage.

TUFLOW Coverages

TUFLOW models use several types of Feature coverages. Feature coverages are used in SMS to represent geometry and associated attributes as points, arcs, and polygons. Unlike most models used in SMS, TUFLOW uses coverage data as model inputs.

TUFLOW coverages can be associated with 2D Geometry Components and TUFLOW Simulations by creating links to the coverage in the component or simulation. The use of links allows multiple simulations or geometry components to share coverage data. Sharing data between simulations reduces required disk space for the simulation and makes it easier to update several simulations with the same changes.

In addition to the coverages listed below, TUFLOW models can use Area Property Coverages to define where to use specific material properties using polygons.

TUFLOW 1D/2D Connections Coverage

1D/2D connections are used with the 2D BC coverage to link 2D and 1D domains.

Two types of arcs can be created in a 1D/2D connection coverage: 1D/2D connection arcs (CN) and Flow vs. Head Connection arcs (SC).

1D/2D Connection arcs are written to a 2d_bc layer file for TUFLOW input (see 2d_bc_layers in the TUFLOW documentation). One end of the connection arcs must end at the same location as a 1D Flow/2D Water Level Connection (HX) in a TUFLOW BC coverage. The other end of the connection arc must end at the same location as a network node in a TUFLOW network coverage.

TUFLOW 1D Cross Section Coverage

Cross section coverages are used to define open channel cross section data for 1D networks. Cross section geometry is generally extracted from a TIN and may be edited by hand if desired.

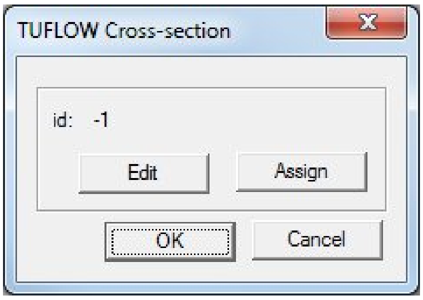
Cross sections are created by creating arcs in a cross section coverage. SMS can automatically create cross section arcs from a 1D network.

After cross section arcs are created, the elevations for the arc can be extracted from a scatter set (TIN) by right-clicking on the coverage and selecting **Extract from Scatter** from the menu. This will extract the cross-section data from the active dataset on the active scatter set. Values are extracted at each triangle edge in the scatter set and each node or vertex on the cross section arc.

After cross sections have been created, the materials from an area property coverage may be mapped to the cross-sections. This is done by right-clicking on the coverage and selecting, **Map Materials from Area Property Coverage**.

Individual Cross sections may be viewed and edited by double-clicking on an arc, or by selecting an arc and choosing *Feature Objects*|**Attributes**. This will bring up the *cross-section attributes* dialog.

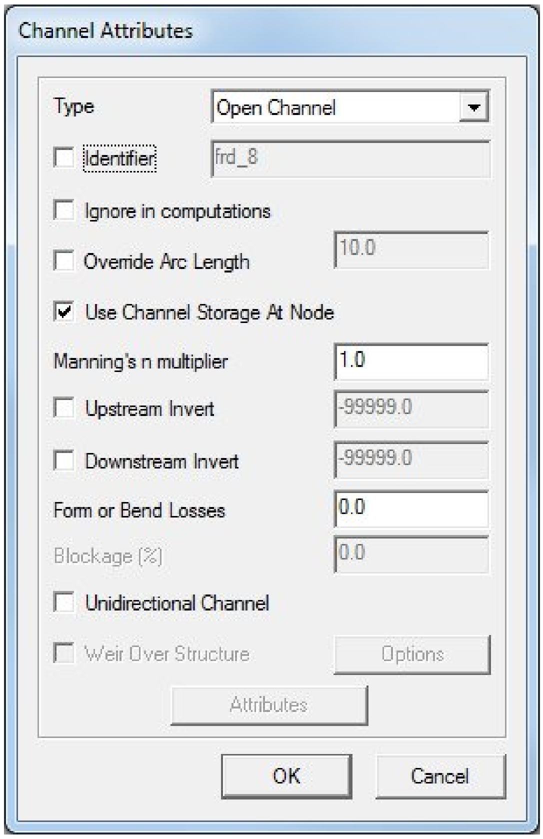
All the cross-sections in the coverage are stored in a cross section database. The cross section database may be opened by right-clicking on the coverage and selecting **Properties** (see Editing Cross Sections).



TUFLOW Cross Section dialog

TUFLOW 1D Network Coverage

1D domains are made up of a network of channels and nodes. Channels represent the conveyance of the flowpaths and nodes represent the storage of inundated areas (TUFLOW Users Manual). Channels are created using arcs and the arc endpoints are the nodes.



TUFLOW *Channel Attributes* dialog

There are a variety of channel types including open channels, weirs, and culverts (pipe networks). The channel also has a variety of attributes depending upon the channel type. The channel type and attributes are defined in the *Channel Attributes* dialog.

There are two types of nodes generic nodes, and inlets. Generic nodes can be used to specify storage and can be used to set channel invert elevations. Inlet nodes are used to get flows from a 2D domain into a 1D pipe network below the 2D domain. The node type and attributes are defined in the *Network Node Attributes* dialog.

The TUFLOW documentation sections 4.5.1.3: Connecting Pits and Nodes to 2D Domains and 4.5.3: 1d_nwk Attributes list some new TUFLOW features that SMS now supports. SMS's *Network Node Attributes* dialog has several new additions to the *Create Connection to 2D Domain (SX)* section. These new options allow you to control elevations at the connections, how many cells are connected, and the method for selection of additional cells (Grade or Sag). Each option correlates fairly directly to a TUFLOW field and some are labeled as such to make look-up easy.

TUFLOW 2D/2D Links Coverage

A 2D/2D Links Coverage is used to setup TUFLOW to use multiple 2D domains. See Linking 2D Domains.

TUFLOW 2D Flow Constriction Shape Coverage

A 2D Flow Constriction Shape coverage is used to define flow constrictions in TUFLOW. These are used to model hydraulic structures or other sources of additional losses in TUFLOW. Flow constrictions are of two categories: standard (non-layered) and layered flow constrictions. Layered flow constrictions can be used to model situations where flow has multiple pathways at different elevations. Examples would include flow under a bridge and over the bridge deck as well as a pipeline (typically large box culverts to model as 2D) crossing a waterway. The standard (non-layered) flow constrictions can model box culverts, floating bridge decks, bridges, or apply additional form losses to an area (due to submodel scale features). Flow constrictions can be created for arcs or polygons.

Note that this coverage is different from the older (and outdated) 2D Flow Constriction coverage, which is still supported.

Overview

Standard (non-layered) Flow Constrictions

Flow constrictions introduce additional losses and/or reduced flow areas through an area of the domain. Flow constrictions can be used for large box culverts, bridges, or floating bridge decks. Flow constriction properties include invert elevations (optional), low chord (obvert) elevations for structures, blockage information, manning n values, and form loss coefficients.

Layered Flow Constrictions

Here is the description from the TUFLOW manual:

> Four layers (not GIS layers!) are represented, with the bottom three layers each having their own attributes. The top, fourth, layer assumes the flow is unimpeded (eg. flow over the top of a bridge). Within the same shape, the invert of the bed, and thickness of each layer can vary in 3D. Each layer is assigned its own percentage blockage and form loss coefficient. For example, the layers of a bridge structure would be along the following lines.

- Layer 1: Beneath the bridge deck. Might be 5% blocked due to the bridge piers and have a small form loss for the energy losses associated with the piers.
- Layer 2: The bridge deck. This would be 100% blocked and the form loss coefficient would increase due to the additional energy losses associated with flow surcharging the deck.
- Layer 3: The bridge rails. These might be anything from 100% blocked (solid concrete rails) to 10% blocked (very open rails). Some form losses would be specified depending on the type of rails.
- Layer 4: Flow over the top of the rails – flow assumed to be unimpeded.

> Layered FCs function by adjusting the flow width of the 2D cell so as to represent the combination of blockages of the four layers, and by accumulating the form losses. When the flow is only within Layer 1, only the attributes of Layer 1 are applied. As the water level rises into Layer 2, the influence of the Layer 2 attributes increase as the water continues to rise. Similarly for Layer 3 and Layer 4.
>
> The cell side flow width is calculated by summing the flow areas of each layer (including the effects of layer blockages), and dividing by the water depth. The form losses are applied as follows:

- If the water is entirely within Layer 1, the Layer 1 FLC is applied.
- If the water level has reached Layer 2, the value applied is the Layer 1 FLC plus a fraction of the Layer 2 FLC based on the depth of water within Layer 2. For example, if Layer 1 FLC is 0.1 and Layer 2 is 0.5, and the

water is 40% of the way up Layer 2, the FLC applied is 0.1 + 0.4*0.5 = 0.3.

- Similarly, if the water level is into Layer 3, the FLC is the Layer 1 FLC plus the Layer 2 FLC plus a fraction of the Layer 3 FLC.
- Once the water level is above Layer 3, ie. is into Layer 4, the FLC is held constant at the sum of the FLCs for Layers 1 to 3.

--TUFLOW Manual 2008-08

Varying Elevations Within Flow Constriction

Flow constrictions can be specified as polygons or arcs (along a line). For a flow constriction defined by a polygon, some values may be varied within the polygon by using arc and point attributes. For standard flow constrictions, the invert and low chord (obvert) elevations can be modified within the polygon. For layered flow constrictions, the layer elevations, blockage, form losses, and invert elevations can vary within the polygon.

In order to distribute these vertices for the flow constriction, TUFLOW creates a TIN from the specified data points and interpolates these values through the flow constriction. The flow constriction dialog has several attributes which influence how this TIN is generated and applied. The final elevations for the grid can be verified by reading the "zsh_zpt_check.mif" file into the GIS coverage. The final information about standard flow constrictions can be verified using check files. The "fcsh_uvpt_check.mif" for standard flow constrictions and "lfcsh_uvpt_check.mif" for layered flow constrictions (see SMS:TUFLOW Check Files).

Arcs on the perimeter or within a polygon flow constriction can be used to spatially distribute the properties above. To use an arc for elevations, you need to specify it to be of type "Breakline Elevations" in the arc properties dialog. Elevations can be specified at nodes at the end of arcs but not on intermediate vertices. Therefore, in order to represent curved flow constrictions or changes in slope, you need to convert vertices to nodes to create multiple arc segments. These arcs will become breaklines in the TIN generated by TUFLOW. Points within a polygon can also be used to specify elevations and will become an individual point in the generated TIN.

It is important to keep in mind when using flow constrictions that elevation data comes from a combination of the cell elevations, elevations specified with the flow constriction itself, and elevations assigned to perimeter nodes or interior points depending upon the whether each kind of data exists and the options chosen for the flow constriction.

Creating Flow Constrictions

To create a flow constriction for an arc or polygon:

- Create a 2D Flow Constriction Shape coverage and an arc/polygon in the coverage. Note that the old 2D Flow Constriction coverage is also still available, and is different from the one we are creating now.
- Double-click on the arc/polygon and change the type to Flow Constriction or Layered Flow Constriction and set the options according to the guidelines below.

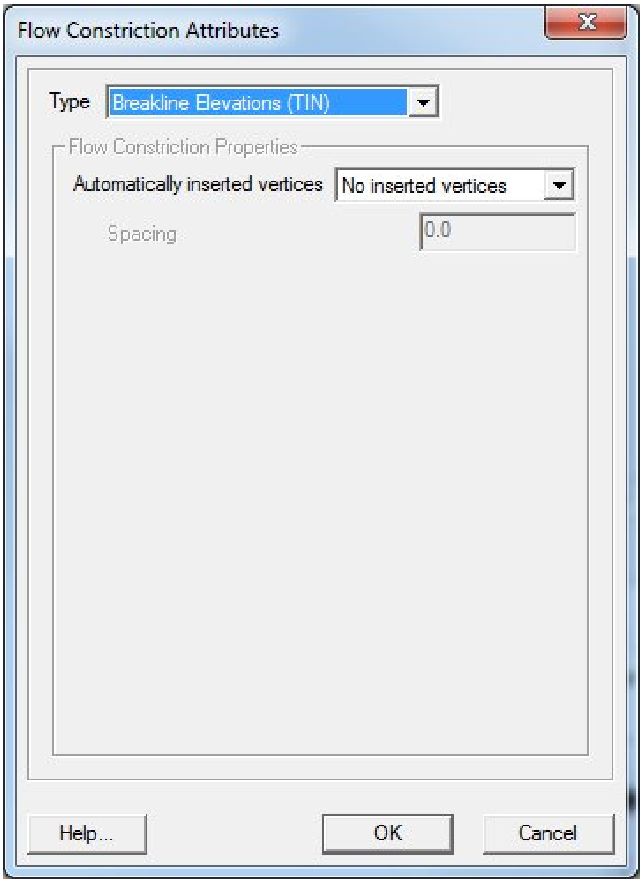
Flow Constriction Options

- **Width –**
- **Override Invert Elevation –** Unless overriden, TUFLOW will use the 2D cell elevations for the invert elevations of the flow constriction.
- **Option –** This controls when invert elevations will be applied. If the choice is *Minimum*, the new invert elevations will be used only if they are lower than the original cell elevations. Similarly, if the choice is *Maximum* the new elevations will only be used if they are higher than the original cell elevations. If *All* is chosen, the new elevations will be used regardless if the elevations are higher or lower.

TUFLOW has the option to merge elevations at the perimeter of the flow constriction with the existing elevations that exist in the perimeter cells to make the transition between the elevations smooth. The *Default* option (polygons only) will merge elevations of perimeter points that do not have a specified elevation value but apply specified elevations at points. The *All* option (arcs only) will ignore specified elevations at perimeter points and merge the values with the perimeter points. The *No Merge* option will ignore existing cell elevations and the elevations will come from specified node values if they exist otherwise they will come from the elevations specified for the flow constriction.

Breakline Elevations

- **Automatically Inserted Vertices** - Used to affect how TUFLOW generates the TIN to use to interpolate elevation data within the flow constriction. Additional vertices can be useful to provide a better triangulation and more smooth transitions between values. The default is for TUFLOW to generate additional vertices so that the final spacing is not more than half the cell size.



TUFLOW *Flow Constrictions Attributes* dialog showing *Breakline Elevation (TIN)* options

Layered Flow Constriction Options

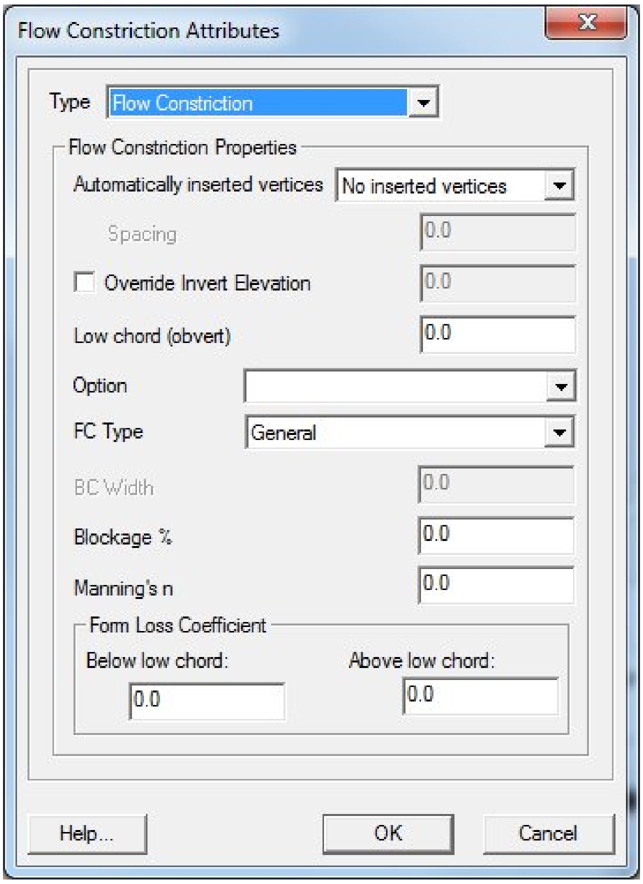
- **Invert Elevation Offset –** This will offset the invert elevations whether they are specified or are the original cell elevations.
- **Layer 1 Low chord –** The low chord (obvert) represents the top of layer 1. For a bridge this is the low chord of the bridge deck.
- **Layer 2 and 3 Depths –** This represent the depth of the layers after layer 1. The obvert of each layer is determined on a cell by cell level from the layer1 low chord/obvert elevation.
- **Blockage % –** This represents the percentage of the flow width lost due to large piers, railings, or other flow impediments. If half of the flow is blocked, the percentage lost is 50%.
- **Form Loss –** The form loss is the ratio of the dynamic head that is lost in the structure. This value can be larger than 1.0 since all of the energy does not have to come from the velocity component. This can be estimated from experience or by comparing with other numeric codes.

Non-Layered Flow Constriction Options

- **FC Type –** Flow Constriction Type. Options are:
  - **General –** Does not include allowances for any vertical walls or friction from underside of deck.

- **BC –** Box Culvert
- **BD –** Bridge Deck
- **FD –** Floating Deck

- **Low chord (obvert)/BC Height/Bridge Deck Depth –** The usage of this field changes depending on FC Type (above). Enter a sufficiently high value (eg. 99999) if there is no obvert constriction.
  - **General** or **BD –** Low chord/Obvert (soffit) of constriction in m above datum.
  - **BC –** The height of the box culvert.
  - **FD –** Floating depth (m) of the deck (ie. depth below the water line).
- **BC Width –** The width of one BC culvert barrel in metres. For example, if there are 10 by 1.8m wide culverts, enter a value of 1.8.



TUFLOW *Flow Constrictions Attributes* dialog showing *Flow Constriction* options

- **Blockage % –** The percentage blockage of the cells. For example, if 40 is entered (ie. 40%), the cell sides are reduced in flow width by 40%, ie. is set to 0.6 times the full flow width.
- **Manning's n –** According to TUFLOW's docs:

  For box culverts (BC), the Manning's n of the culverts (typically 0.011 to 0.015) should be specified. This value prevails over any other bed resistance values irrespective of where in the *.tgc file they occur (the exception is if another FC BC object overrides this one). If set to less than 0.001, a default value of 0.013 is used.

  For bridge decks (BD), can be used to introduce additional flow resistance once the upstream water level reaches the bridge deck low chord/obvert (or soffit). For floating decks (FD) this is always the case as the deck soffit is permanently submerged. The additional flow resistance is modelled as an increase in bed resistance by increasing the wetted perimeter at the cell mid-side by a factor equal to (2.*Bed_n)/FC_n. For example, if the FC Manning's n and the bed Manning's n values are the same, the wetted perimeter is doubled, thereby reducing the conveyance and increasing the resistance to flow. To be used as a calibration parameter to fine-tune the energy losses across a bridge or floating structure.

Ignored for General FC Types.

- **Form Loss Coefficient –** Form loss coefficient to be applied above and below the FC low chord/obvert. Used for modelling fine-scale "micro" contraction/expansion losses not picked up by the change in the 2D domain's velocity patterns (eg. bridge pier losses, vena-contracta losses, 3rd (vertical) dimension etc). The effect of these fields changes between arcs and polygons, thick and thin lines, as detailed in the TUFLOW docs (section 4.7.2).

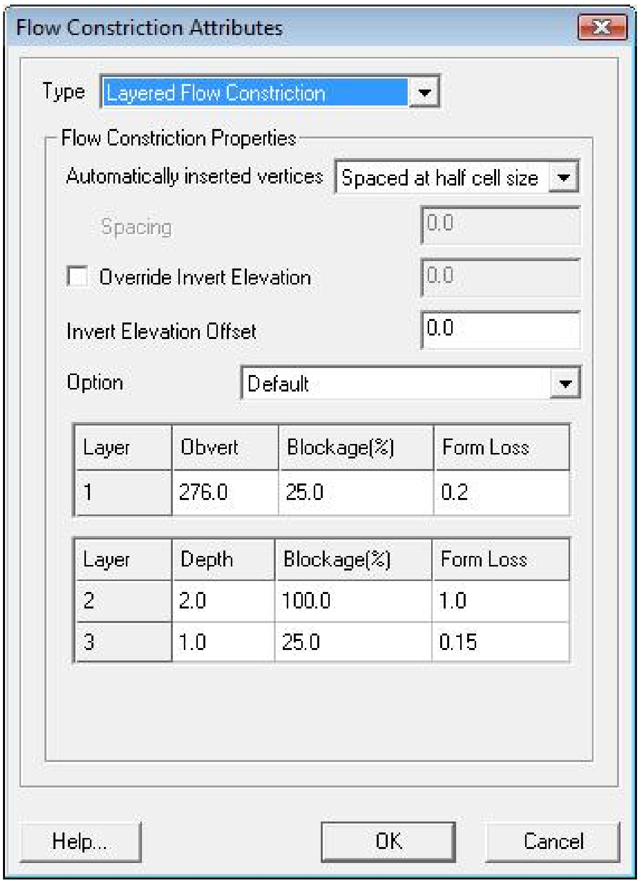
Node Options

All nodes have the option of overriding the invert height at their location. Beyond this, the fields available in the node attributes dialog change based on whether the node is inside (or on the edge) of a Layered Flow Constriction Polygon. If so, the node properties will resemble that of the layered polygon's with layers 1–3. Otherwise only the Low chord (obvert) can be modified.

As with polygons, the Low chord can actually mean different things depending on the settings of FC Type and other factors. This is determined by the polygon the node is on or inside of.

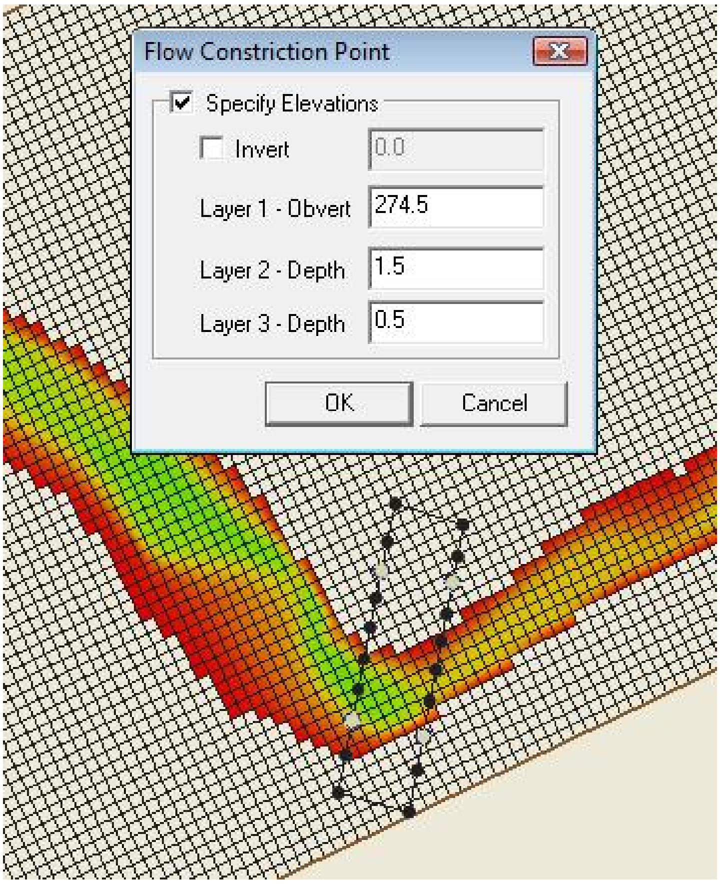
Example: Arched Bridge

Because it's hard to glean applicable knowledge from a dry description of dialogs, here's an example of how they might be used. We're going to create an arched bridge using a Flow Constriction polygon and Breakline Elevations.



Flow Constriction Attributes dialog

1. Start by creating your Flow Constriction Shape coverage.
2. Next create 4 connected arcs in a rectangle for the polygon for your bridge. Go to *Feature Objects*|**Build polygons** to create the polygon.
3. Select the polygon and enter its attributes dialog. Fill the values as follows:
4. Click the **Select Feature Arc** tool and select both arcs along the sides of the bridge. We want to turn both these arcs into breakline elevations. Right-click and select **Attributes...**.
    1. Set the Type to Breakline Elevations (TIN)
    2. Set Automatically insert vertices to "Spaced at half cell size".
5. Next we want to create some nodes along these arcs so we can set different low chord elevations at each one to create the arch. With the arcs still selected, right-click and select **Redistribute Vertices...**. In the dialog, choose *Number of Segments* for the Specify field and enter 9 as the number of segments. Click **OK**. This splits up the lines so that there are a total of 8 vertices on each, which we will convert to nodes.
6. Choose the *Select Feature Vertex* option and select all the vertices on the bridge sides (either with a drag-select or by holding shift). Right-click and choose **Convert to Nodes**.
7. Now it's time to set the node data. Choose the **Select Feature Point** tool. For each node along the sides of the bridge, enter its attributes dialog (right-click with it selected and choose **Node Attributes...**) and insert the data below. You can multiselect nodes with the *SHIFT* key and edit their attributes simultaneously. Since both sides of the bridge will be the same, and the data below is symmetrical on each end of the bridge, you can select 4 nodes at once to set their values to save time.

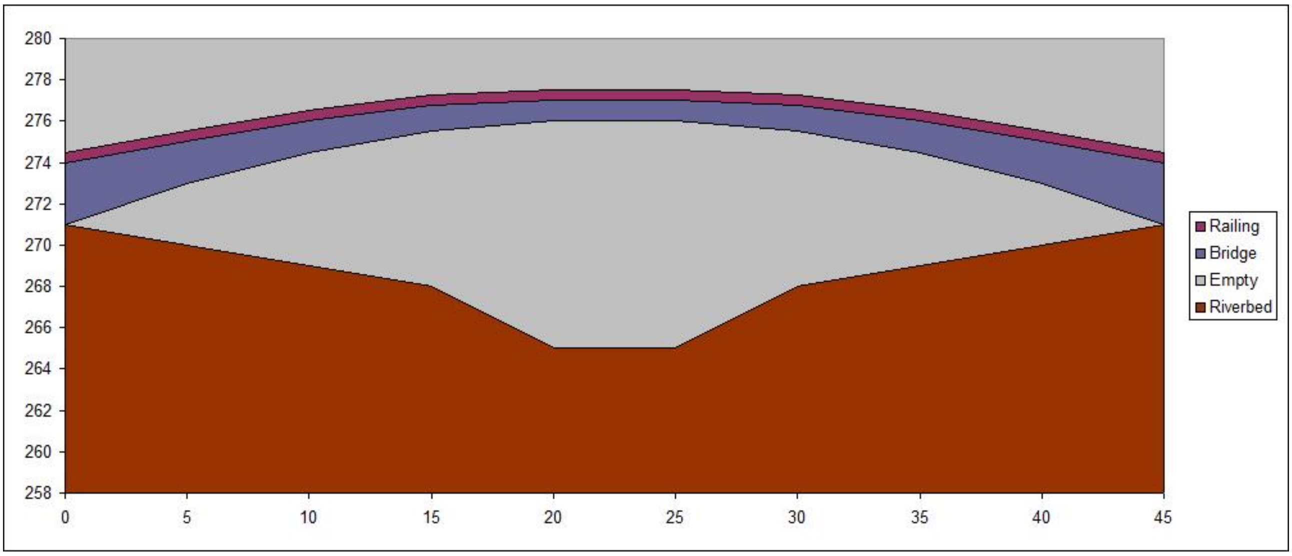


Flow Constriction Point dialog showing *Layered Flow Constriction* options

Node	L1 Low chord	L2 Depth	L3 Depth
Node1	271	3	0.5
Node2	273	2	0.5
Node3	274.5	1.5	0.5
Node4	275.5	1.25	0.5
Node5	276	1	0.5
Node6	276	1	0.5
Node7	275.5	1.25	0.5
Node8	274.5	1.5	0.5
Node9	273	2	0.5
Node10	271	3	0.5

We now have an arched bridge that will look something like the image below from the side. We added 25% blockage under the bridge to cover things like pillars holding the bridge up, vegetation, fairy-tale trolls hiding underneath, and so on. Layer 2 is the bridge itself, which provides 100% blockage because our bridge is solid concrete and does not allow any water to flow through that section. Above that, the railing gives 25% blockage but otherwise lets water through just fine. And above that? Nothing—any water that rises above the railing of the bridge is completely unrestricted.

The varying width of Layer 2 allows us to create a bridge that is thickest on the ends of the bridge and thin in the center. Our Layer 1 blockage takes support pillars into account, and TUFLOW will automatically determine how much of layers 1, 2 and 3 are affecting the water at any given time and calculate the flow constriction from there.



Bridge Layer Definitions

TUFLOW 2D Spatial Attributes Coverage

Spatially varied attributes can be defined using the 2D spatial attributes coverage. The values for the attributes are assigned to polygons. TUFLOW assigns the values from the polygons to the cells that exist within the polygon.

The two available spatial attributes are initial water levels and weir factors.

Assigning spatially varied initial water levels may be necessary to ensure that water bodies have water in them initially.

TUFLOW 2D Z Lines (Advanced) Coverage

The 2D Z Lines (Advanced) Coverage is similar to the geometry modification coverage which allows you to modify terrain along an arc or arcs, but has additional options. With the 2D Z Lines (Advanced) coverage, you can modify geometry through time to simulate levee failures or other changes to elevation data within the model run. These changes are set up start when a "trigger" is activated such as at a specific time during the simulation or when water depth exceeds a certain amount. An example application is a levee collapsing when flood water overtops it.

- Make sure the coverage is in the Geometry Component(s) you wish to apply to the modification.

TUFLOW also supports the features described above using polygon ZShapes. However, SMS does not support polygon ZShapes using these advanced features (such as triggers) at this time. If you need to create static modifications to terrain using polygons you can use the 2D Z Lines/Polygons (Simple) Coverage.

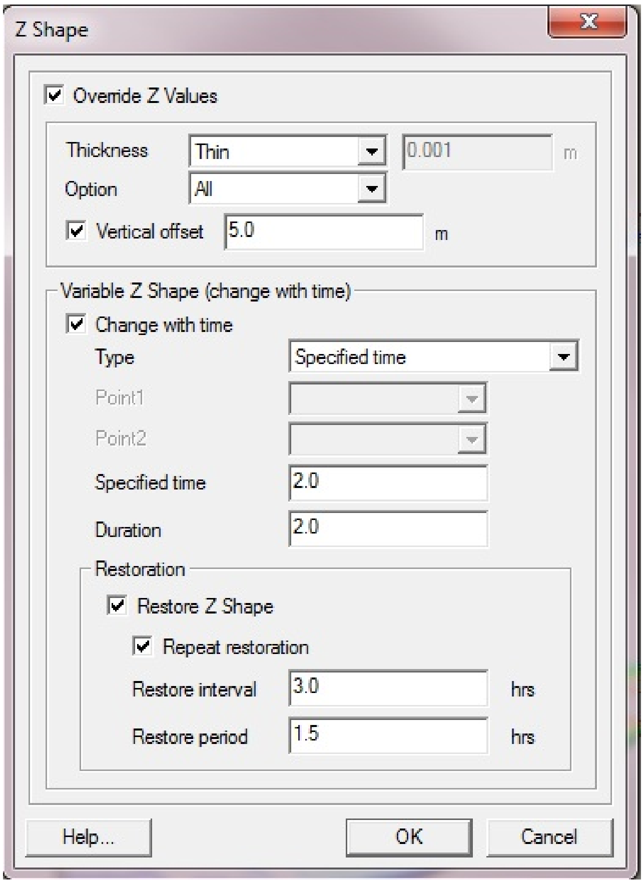
The 2D Z Lines (Advanced) Coverage

The 2D Z Lines (Advanced) coverage is available under the models/TUFLOW folder. ZShape data is stored in a feature arc's attributes. The Z values of the feature arc's points determine its Z, not the Z of the arc itself (exceptions to this are explained in the Arc Properties Dialog section below).

ZShapes are split into two main categories, Static ZShapes and Variable ZShapes. Static ZShapes are simple terrain modifications that do not change over time. They can be used to create a levee, pit, sand bar, dam, etc. Variable ZShapes have a trigger that causes them to raise or lower the terrain during the simulation. Often a variable ZShape is combined with a static one to provide raised terrain for the trigger to modify. This is necessary because a variable z shape cannot raise the same area it intends to lower when its trigger activates -- it needs the terrain to already be there and cannot create it itself, only modify once its trigger activates.

Arc Properties Dialog

Arcs in a 2D Z Lines/Polygons (Advanced) coverage store ZShape data only if they are told to "override Z values". Otherwise they are not exported for TUFLOW's use. To reach the properties dialog,right-click on an arc and choose "Attributes...". Below is an explanation of the zshape options.



Z Shape Properties dialog

- Override Z Values: ZShape data is only stored and exported if this checkbox is enabled. It tells TUFLOW to override the current terrain Z values with the new values from the arc.
- Thickness: Arcs can be Thin Lines, Thick Lines, or Wide Lines. Thin lines have a width of 0, thick lines have a thickness less than or equal to 1.5 times the 2D cell size, and wide lines are any width larger than that. Thin lines follow some special rules that affect other options in the dialog, such as Option and Trigger Type.
- Option: The ZShape Options are All, Add, Min, Max, and Offset. Offset is only available for variable thin line Z Shapes, which cannot use the Add, Min or Max options. Add is unavailable for any variable Z Shape.
  - Offset is not listed by the same name in the TUFLOW documentation and is an alias in SMS for not specifying "NO MERGE". This causes the points of the arc to not be written, and the Z of the arc is determined by the Offset field.
- Offset: Raises the entire arc by this amount. Negative values are appropriate.
  - When offset is specified as the Option for variable thin lines, this field specifies the height to adjust terrain to (instead of adjusting it up or down by this amount) when triggered.
  - For all other variable arcs, this the the amount to adjust the terrain by when the trigger activates.
- Use Trigger: Toggles the arc between being a static zshape and a variable zshape. Variable z shapes use triggers to change terrain when a specific event occurs. All variable Z Shapes have the Duration option to span the change over a certain period of time. Other options will change with the Trigger Type.
  - Trigger Type: The type of event that will trigger the Z modification.
    - Specified time: Triggers at a specific time during the simulation, measured in hours.
    - Water level at point: Uses a Trigger Point to measure water level, and triggers when the water level reaches or exceeds the value in the Water level field.
    - Water level difference: Uses two trigger points. Triggers when the difference between water levels measured at both points exceeds the value in Water level difference. A difference in either direction is treated the same in TUFLOW, and there is no need for negative numbers.
    - Thin line water level: Similar to Water level at point, but for thin lines. Does not use a trigger point, instead measuring the water levels on either side of the thin line.

- Thin line water level difference: Triggers when the difference between water levels on either side of the thin line reaches or exceeds the value in Water level difference.

- Point1 and Point2: Both point selection boxes list the trigger points available in the coverage. These fields are only enabled for trigger types that use them.
- Specified time, Water level, Water level difference: These fields become available based on the trigger type selected.
- Duration: If set to 0, changes are instantaneous once triggered. Otherwise the change will be interpolated over this many hours.
- Restore Z Shape: This option can be used to have TUFLOW restore the original elevations at a specified time.
- Repeat restoration: The z-shape will be restored indefinitely.
- Restore interval: The time (hrs) between when the variable z-shape is finished and when to start restoring the points back to their original values.
- Restore period: The time (hrs) needed to restore the points back to their original values.

Trigger Points

Trigger points are used by the Water level at point and Water level difference trigger types. Simply create a feature point and enter its attributes dialog to create a trigger point.

Trigger Point Dialog

The trigger point dialog is used to turn a feature point into a ZShape trigger point. Enabling the Specify Trigger checkbox and giving the point a name will cause it to appear in the Point1 and Point2 selection boxes in the arc attributes dialog for use as a trigger point. If you rename a trigger point that is being used by triggers, or delete it by unchecking Specify Trigger, SMS will notify you that there are one or more arcs/polygons being updated with the new change.

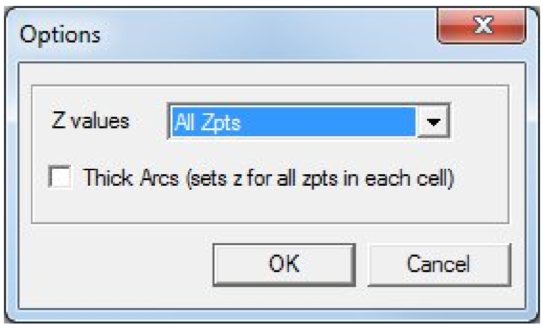
TUFLOW 2D Z Lines/Polygons (Simple) Coverage

These coverages are used as geometry modifications and force grid elevation values using arcs or polygons. This can be useful to ensure ridge or gully lines are represented in the model, simulate a proposed levy, or to simulate a proposed fill/excavation scenario. It is possible to only have the elevations assigned from a feature object if the feature elevation is higher or lower (user specified) than the existing elevations in the grid. This is controlled in the *2D Z Lines/Polygons (Simple) Coverage Properties* dialog (right-click on the coverage and choose **Properties**). The options specified in the coverage properties dialog apply to any geometry modification arcs and polygons within the coverage. If you want to use multiple settings within a simulation, you must have multiple 2D z line/polygon (simple) coverages.

Z Values

The z values determine how the elevations in the coverage are used to modify the existing model elevations. The options are:

1. All Zpts (default) – The z values from all of the arcs/polygons are used.
2. Min – The z values from the arcs/polygons will only overwrite the existing elevations if the elevations are lower.



TUFLOW 2D Z Lines/Polygons (Simple) Coverage Properties Z Values *Options* dialog

3. Max – The z values from the arcs/polygons will only overwrite the existing elevations if the elevations are higher.
4. Add – The z value becomes the existing elevation plus the z value from the arc/polygon (which may be negative).

Thick Arcs

If this option is not selected, the z values of the arc will be applied to the nearest elevation locations in each cell that the arc passes through. Otherwise, the arc elevations will be applied to every elevation location in each cell that the arc passes through.

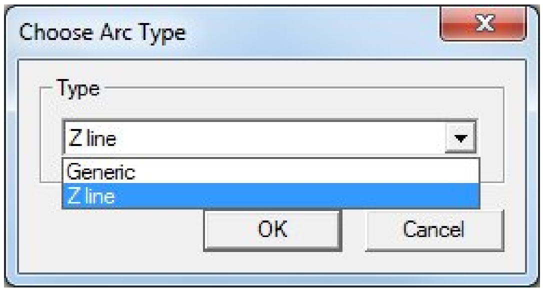
Make sure the coverage is in the Geometry Component(s) you wish to apply to the modification to.

Points

Points are ignored in the geometry modification coverage.

Arcs

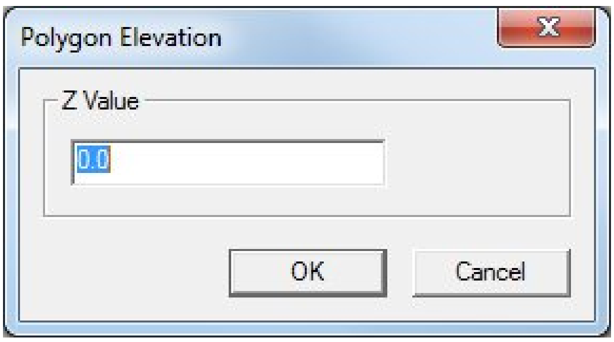
Only the elevations at the nodes (endpoints) are used to set the grid elevations. Vertex (intermediate) elevations are ignored. Cell elevations along the arc will be interpolated linearly based on distance from the endpoint elevation. If you would like to specify elevations at vertices on an arc, convert the arc vertices to nodes to create multiple arcs joined end to end.



TUFLOW *Choose Arc Type* dialog

Polygons

To set the elevation for polygons double-click on the polygon and enter the elevation in the *Polygon Elevation* dialog. This elevation will be applied to all cells within the polygon. This dialog can also be accessed by right-clicking on a polygon and selecting **Attributes** for the right-click menu.



TUFLOW *Polygon Elevation* dialog

TUFLOW Boundary Conditions Coverage

Boundary conditions are defined in TUFLOW with points, arcs, or polygons in a BC coverage. BC coverages can also be used to specify cell code (active/inactive) areas of the 2D model domain. The kinds of boundary conditions available depend upon whether it is applied to a point, arc or polygon.

Boundary conditions defined at points are 1D boundary conditions and must be placed at the same location as a 1D boundary node. (See Snapping Feature Objects) The boundary condition information is specified in the *BC Node Attributes* dialog (see BC Node Attributes Dialog).

Arcs can be used to define 2D Boundary conditions applied to the 2D domain. The BC attributes are specified in the *BC Arc Attributes* dialog (see BC Arc Attributes Dialog).

Polygons can be used to define rainfall applied to 2D domains or to specify active/inactive information for 2D domains. The *BC Polygon Attributes* dialog specifies the type of information stored with the polygons (see BC Polygon Attributes Dialog).

Clip Region Read TIN Z pts Coverage

Using a coverage to specify a clip region, makes it so only elevations within coverage polygons are changed by a TIN dragged under a 2D geometry component. This is particularly useful for clipping out a TIN due to unwanted or irregular triangulation around the periphery, especially for secondary TINs of proposed developments lying within the primary TIN.

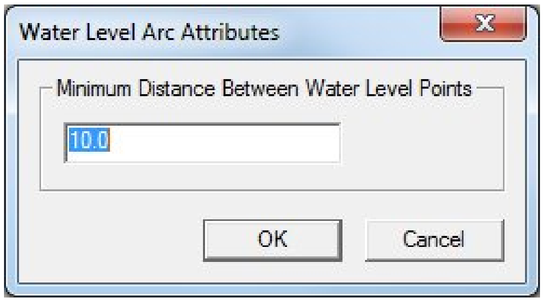
To use this coverage, create a coverage with one or more polygons. Only areas within polygons and within the TIN boundaries will have elevations assigned. You must have a scatter set in the 2D Geometry Components. Then drag and drop this coverage under the scatter tree item, under the 2D Geometry Components.

TUFLOW Grid Extents Coverage

A TUFLOW Grid Extents coverage is used to create TUFLOW grids. TUFLOW grids are created by creating and positioning a Cartesian Grid Frame and then right-clicking on the coverage and choosing **Map→2D Grid**.

TUFLOW Water Level Lines Coverage

Water level line coverages define the locations where 1D solutions will be written as 2D output. TUFLOW creates water level points along the water level lines. The water level lines in conjunction with the water level points guide TUFLOW on how to represent the 1D flow in the 2D domain.



TUFLOW *Water Lever Arc Attributes* dialog

Arcs are the only objects used in the water level lines coverage. The only attribute in the water level lines coverage is the minimum distance between water level points. This is used by TUFLOW to generate the water level points along the arcs. To change this attribute, double-click on an arc or right-click and select *'Attributes* from the right-click menu. This brings up the *Water Level Arc Attributes* dialog.

For more information about how TUFLOW uses water level lines to generate 2D flows see TUFLOW Water Level Points Coverage.

TUFLOW Water Level Points Coverage

Water level points are used in conjunction with water level lines to guide TUFLOW on creating 2D output for 1D networks (see TUFLOW Water Level Lines Coverage).

Water level points store a material value. This material value along with the z location of each point is used by TUFLOW to better approximate 2D flow along a water level line. TUFLOW performs a parallel channel analysis so that flow rates reported in the 2D output vary based upon the depth and roughness of a channel. Areas that are more rough and shallow than other areas will have a smaller flow rate than other areas along the same water level line.

The material value for each point can be mapped from an area property coverage by right-clicking on the coverage tree item and selecting *Materials for Area Coverage*. By default the elevation of each point is based upon the neighboring cross-section information.

Creating the Water Level Points Coverage

The initial water level points coverage data is created by TUFLOW during a run. Once a run has been completed, the data can be read from the check file that starts with the simulation name and ends with _WLLp_check.mif (see TUFLOW check files). Open this file from SMS and select TUFLOW WLL Points from the combo-box that comes up. This coverage now can be included in simulations to provide additional guidance to TUFLOW in distributing 1D flows.

3.7.b. Interface Components

Interface Components

The map module interface consists of the display options, menus, right-click menus, and tools associated with the map module.

- Menus
- Tools
- Project Explorer Items

Related Topics

- Display Options
- SMS Menus
- Dynamic Tools

Map Module Display Options

Map Module

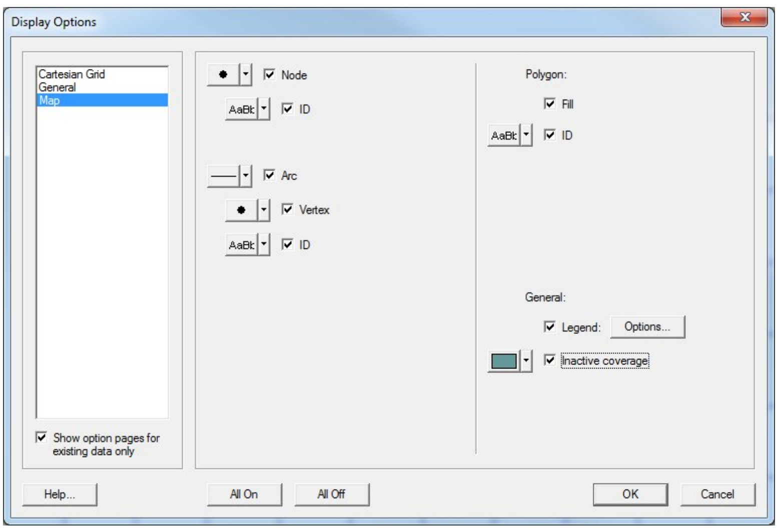
Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

The properties of all mapdata that SMS displays on the screen can be controlled through the map tab of the *Display Options* dialog. This dialog is opened by right clicking on the *Map Data* entry in the *Project Explorer* and selecting the **Display Options** command. (It can also be accessed from the from the *Display* menu or the **Display Options** Macro.)

The exact layout of the *Display Options* dialog for feature objects depends on the active Coverage type. Some options are available on all coverages, while other options are only available on certain coverage types. The following options are available in the *Display Options* dialog. The entities associated with the map module with display options are shown below. Some of these entities also show an **Options** button to the right. For these entities, additional display controls are available. The available map display options include the following:

Feature Objects

- Nodes
- Node IDs
- Refine Points
- Nodal Boundary Conditions
- Arcs
- Arc IDs
- Vertices
- Arc Boundary Conditions
- Polygon Fill
- Fill Colors
- Polygon IDs
- Legend
- Display of inactive Coverages



Display Options dialog showing *Map* options with Area Property coverage

Drawing Objects

Each drawing object stores its own attributes specified by selecting the individual object. This dialog allows the user to turn on/off all drawing objects of a specific type.

- Text
- Lines
- Rectangles
- Ovals

Arcs

Display the feature arcs.

- ID – Display the ID of the feature arcs.
- Vertices – Display the vertices along the arcs.
- Arc types – Arc attributes may be displayed depending on the coverage.
- BC Display Opts – Open a dialog with display options for the boundary condition specific to the active coverage.
- CL Flow Direction Arrow – Display an arrowhead showing the direction of center line arcs.

Polygons (fill)

Color fill the feature polygons. The display of the boundary of the polygons is controlled by displaying feature arcs.

- ID – Display the ID of the feature arcs.
- Polygon types – Polygon attributes may be displayed depending on the coverage.
- Show Materials / Mesh Type – Fill the polygons to display the materials assigned to the polygons or the mesh generation type assigned to each polygon.
- Material Opts – Set the display options of the materials.

Grid Frame

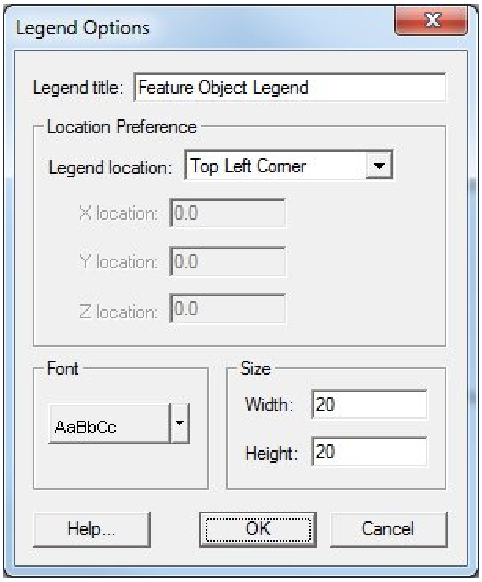
Set the display of the grid frame (Cartesian grid coverages only).

Inactive coverage color

Set the color for the display of all inactive coverages.

Legend

Turn the legend on or off for feature objects. The **Options** button opens a dialog that controls the title, font, location, and size of the legend. The **Active** button in this dialog signifies to show only the active coverage in the legend and the **All** button signifies to show all coverages in the legend.



Map Module Display *Legend Options* dialog

Background Color

Set the background color of the Graphics Window.

Observation Coverage Only

- **Calibration Target** – Turn on/off calibration targets drawn next to observation points. The Interval and Two std. dev. options tell SMS to set the size of the calibration targets based on the interval or the standard deviation assigned to each point. The Scale tells SMS to scale the targets larger (>1.0) or smaller (<1.0) than the default size.
- **Computed** – Tells SMS to Use the active data set or to Use the selected data set for displaying the calibration targets.

Related Topics

- Map Module
- Display Options

Map Module Menus

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

The following menus are available in the Map Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Feature Objects Menu
- Map Module Right-Click Menus

Map Feature Objects Menu

The *Feature Objects* Menu includes the following commands:

General Commands

- Delete – This command deletes all the feature objects in an SMS session and creates a new blank coverage (since SMS requires an active coverage at all times). SMS will ask for confirmation of this action.
- Attributes
- **Create Arc Group** – This command will create a new entity from a group of selected contiguous (end to end) arcs. If the selected arcs are not connected end to end, SMS will give an error message. These arc groups can be used in some models to assign boundary conditions. This command is used to create an arc group from a continuous string of selected arcs. Once the arc group is created, it can be selected using the **Select Arc Group** tool. Properties can be assigned to the arc group as a whole, and the arc group can be selected to display the computed flow through the arc group. An arc group is deleted by selecting the arc group and selecting the *DELETE* key. Deleting an arc group does not delete the underlying arcs.
- **Build Polygons** – While most feature objects can be constructed with tools in the *Tool Palette*, polygons are constructed with the **Build Polygons** command. Since polygons are defined by arcs, the first step in constructing a polygon is to create the arcs forming the boundary of the polygon. After forming loops with arcs, choose *Feature Objects* | **Build Polygons** from the menu. The build polygons command will form polygons from all closed loops in the coverage.
- Clean
- **Vertices ↔ Nodes** – In some cases, it is necessary to split an arc into two arcs. This can be accomplished using the **Vertex ↔ Node** command. Before selecting this command, a vertex on the arc at the location where the arc is to be split should be selected. The selected vertex is converted to a node and the arc is split in two. The **Vertex ↔ Node** command can also be used to combine two adjacent arcs into a single arc. This is accomplished by converting the node joining the two arcs into a vertex. Two arcs can only be merged if no other arcs are connected to the node separating the arcs. Otherwise, the node must be preserved to define the junction between the branching arcs.
- Reverse Arc Direction – Reverses the direction of all selected arcs.
- Redistribute Vertices – Automatically creates a new set of vertices along a selected set of arcs at either a higher or lower density.
- Transform Feature Objects
- Select/Delete Data...
- **Find** – The **Find...** feature object command allows the user to find a feature object node, arc or polygon by its specified ID number.
- Map → 2D Mesh
- Map → 2D Grid
- **Map → Scatter** – The **Map → Scatter** command allows scatter sets to be interpolated from map data. Scatter points can be created from a specified source. Scatter points data can be extrapolated from feature points and vertices on arc or on feature points only on arc, feature points not on arcs or from feature polygon meshing options. The elevation source can be obtained from arc elevation, arc node and vertex elevations or from arc spacing.

Coverage Type Specific Menus

Optional menu items appear according to the active coverage type.

Generic Coverage Types

Stamping

- Stamp Features

Model Coverage Types

ADCIRC

- Model Control
- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens. For more information, see Arcs: Create Contour Arcs. This command is available if the current coverage type is SHOALS, ADCIRC, or CGWAVE.
- Define Domain

BOUSS-2D

- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.
- Extract Elevations – This option is found in the *Feature Objects* menu, in the Map module, when SHOALS is the active coverage type. The elevation of each node and vertex along every profile arc is interpolated from the active scatter set.

CGWAVE

- Model Control
- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.
- Define Domain

CMS-Flow

- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.

CMS-Wave

- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.

GenCade

- New Grid Frame

Related Topics

- Map Module

Map Module Tools

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

The following tools are contained in the dynamic portion of the *Tool Palette* when the Map Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the *Graphics Window* depends on the current tool. The following table describes the tools in the map tool palette.

Tool	Tool Name	Description
	Select Feature Point or Node	The **Select Feature Point or Node** tool is used to select stand alone feature points or the ends of arcs. A single point is selected by left-clicking directly on it. Multiple points can be selected at once by dragging a box. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. Additional points can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new points without holding the *SHIFT* key will first clear the selection list and then add the newly selected points. A selected point can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature points are locked so they are not accidentally dragged. When a single point is selected, its location is shown in the Edit Window. The coordinates can be changed by typing in the edit field. The Graphics Window's status bar will display information on the selected items depending on the settings find through the *File* | **Info Options** command in the *File* Menu. Selected points can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* Menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Arcs attached to the deleted points are deleted. This tool is available when one or more feature points exist.
	Create Feature Point	The **Create Feature Point** tool is used to place new feature point in the current coverage. A single point is created at a time by left-clicking at the coordinate desired. The newly created point is selected to allow Z Coordinate changes in the Edit Window. This tool is always available, however, creating a feature point is only allowed while in plan view.
	Select Feature Vertex	The **Select Feature Vertex** tool is used to select one or more vertices on an arc. These vertices define the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.
	Create Feature Vertex	The **Create Feature Vertex** tool is used to create a new vertex on the interior of an arc. The vertex is created at the current arc location, but can be selected and moved to change the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.

	Select Feature Arc **Create Feature Arc**	The **Select Feature Arc** tool is used to select one or more existing feature arcs. This is typically done to assign attributes to an arc or delete the arc. A single arc is selected by left-clicking directly on it. Double-clicking on the arc will bring up the arc attributes dialog or that arc. Multiple arc can be selected at once by dragging a box. Additional arcs can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new arcs without holding the *SHIFT* key will first clear the selection list and then add the newly selected points. A selected arc can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature arcs may have elevations associated with the arc as an entity. This is independent of the node and vertex elevations. When a single arc is selected, its elevation is shown in the Edit Window. The coordinates can be changed by typing in the edit field. Multiple feature arcs may also be selected to create a feature arc group to associate attributes with a string of arcs rather than a single arc. These arcs must connect end to end. The arc group is created from the *Feature Objects* menu command. The Graphics Window's status bar will display information on the selected items depending on the settings find through the *File* | **Info Options** command in the *File* Menu. Selected arcs can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* Menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Nodes attached only to the deleted arcs are deleted. This tool is available when one or more feature arcs exist. The **Create Feature Arc** tool is used to create a new feature arc.
	Select Feature Arc Group	The **Select Feature Arc Group** tool is used to assign attributes to a string of arcs. The group must be created before it can be selected as a group. The attributes of the group then operate just as if the group was a single arc.
	Select Feature Polygon	**Build Polygons** menu command.
	**Create 1D Grid Frame**	The **Create 1D Grid Frame** tool only appears when the coverage is associated with a 1D grid model (GenCade). This tool is used to create a guide for the 1D grid that will be generated for a coastal morphology analysis.
	**Select 1D Grid Frame**	The **Select 1D Grid Frame** tool only appears when the coverage is associated with a 1D grid model (GenCade). This tool is used to select/edit the grid frame that is used to generate the 1D grid.
	Create 2D Grid Frame	The **Create 2D Grid Frame** tool is used to create a new grid frame for the creation of Cartesiona grids. This tool is only available for coverages related to Cartesian grid models. The grid frame is defined by clicking three times in the graphics window. The first click defines the origin. The second click defines the I axis of the grid frame (both extents and direction). The third click defines the extents of the J axis. The direction is set to be perpendicular to the I axis.
	Select 2D Grid Frame	The **Select 2D Grid Frame** tool allows the user to select a grid frame and alter its position, orientation and size. This tool is only available for coverages related to Cartesian grid models.

General Tool Right-Click Menus

The following is a list of options that appear in every tool's right-click menu:

- **Clear Selection –** The **Clear Selection** command undoes the selection of the object that was clicked on.
- **Invert Selection –** The **Invert Selection** command selects every object of the same type selected, and undoes the selection of the object that was originally selected.
- **Zoom to Selection –** The **Zoom to Selection** command zooms to a closer view of the object selected.

Tool Specific Right-Click Menus

Each tool in the Map module has its own right-click menu. When the object is selected, one can right-click and a menu will appear. The following is a table showing all of the different tools of the map module with their respective right-click menus.

Tool	Tool Name	Right-Click Menu
	Select Feature Point or Node	• **Convert to Vertex/Vertices –** Converts node(s) to Vertex/Vertices • **Delete –** Deletes node(s) • **Transform –** Moves the node(s) either by scaling, translation, or rotation. User specified. See Data Transform for more information.
	Select Feature Vertex	• **Convert to Node(s) –** Converts Vertex/Vertices to node(s). • **Delete –** Deletes vertex/Vertices • **Transform –** Moves the vertex/vertices either by scaling, translation, or rotation. User specified. See Data Transform for more information.
	Select Feature Arc	• **Create Arc Group –** Creates a group out of two or more arcs selected together. • **Delete –** Deletes arc(s) selected. • **Split Arc(s) –** Sub divides arc(s) at the vertices. See Split Feature Arcs Utility for more information. • **Offset Arc(s) –** Invokes the *offset arc* dialog which prompts the user for offset distances and options to create one or more arcs offset from the selected arc(s). See Offset Arcs for more information. • **Align Arc(s) with Contour –** Moves the nodes and vertices of the selected arc(s) to the closest points on a contour at that value. See Align Arc With Contour for more information. • **Redistribute Vertices –** User specified distribution of vertices. Vertices can be evenly distributed based on spacing or number of vertices desired. See Redistribute Vertices for more information. • **Reverse Arc Direction –** Reverses direction of arc. The arc direction only impacts the direction of extracted 2D plots and the direction for defining a domain. See Reverse Arc Direction for more information. • **Smooth Arc(s) –** Repositions each vertex on an arc to smooths the arc(s). See Smooth Arc for more information. • **Transform –** Moves the arc either by scaling, translation, or rotation. User specified. See Data Transform for more information. • **Attributes –** Type specific. Many types do not have any attributes in their arcs. See Arc Attributes Dialog for more information. • **Select Connected Arcs Turning Left –** Selects the arc to the left of the originally selected arc. See Select Connected Arcs Turning Left for more information.
	Select Feature Arc Group	• **Delete –** Deletes Feature Arc Group(s).
	Select Feature Polygon	• **Delete –** Deletes Feature Polygon (not the arcs that make up the polygon). • **Attributes –** Model specific.
	**Select 1D Grid Frame**	• **Properties' –** Invokes the **Grid Frame Properties** dialog that allows the user to specify the location, orientation, size and spacing of the grid frame using text fields; Specific to 1D grid coverages.
	Select 2D Grid Frame	Type Specific • **Properties –** Shows the properties of the 2D grid frame. Model specific.

Related Topics

- Map Module

Project Explorer Items

In the *Project Explorer*, the Map data folder houses all of the coverages that are controlled by the Map module. The Map data folder can hold as many coverages as the user desires, and can also generate sub folders. Coverages are considered 'active' when clicked on in the *Project Explorer*, and the name of the coverage becomes bold while the coverage icon becomes green. When a new project is created in SMS, a coverage will be automatically created. This coverage is named 'default coverage', and it will be set to a default type which can be specified in 'Preferences'.

Map Module Right-Click Menus

The following Project Explorer mouse right-click menus are available when the mouse right-click is performed on a Map Module item.

Map Module Root Folder Right-Click Menus

Right-clicking on the Map module root folder in the project explorer invokes an options menu with the following options:

- **New coverage –** Opens the *New Coverage* Dialog.
- **New Folder –** Creates a new folder under the Map module root folder.
- **Clear Coverages –** Deletes all coverages.
- **Display Options –** Opens the Display Options Dialog

Coverage Item Right-Click Menus

Right-clicking on a Map item in the Project Explorer invokes an options menu with the following module specific options:

- **Duplicate –** Adds another coverage exactly identical to the existing coverage clicked on.
- **Rename –** Allows the user to specify a new name for the coverage.
- **Convert –** Converts coverage to a 2D grid, mesh or scatter object.
- **Projection –** Allows the user to set the projection of the coverage.
- **Reprojection –** Allows the user to reproject the projection of the coverage.
- **Metadata –** Allows the user to annotate the coverage.
- **Zoom to Coverage –** Zooms to area where coverage is within the graphic window.
- **Type –** Change the coverage type.

New Folder Right-Click Menus

Right-clicking on a new folder item in the Project Explorer invokes an options menu with the following options:

- **New Folder** – Creates a new sub folder under the new folder.
- **Delete** – Deletes the new folder.
- **Rename** – Allows the user to rename the new folder.

Right-click options for the coverage may also include options applicable only to the specific coverage type.

3.7.c. Functionalities

Feature Objects Types

WARNING: Article could not be rendered - ouputting plain text.
Potential causes of the problem are: (a) a bug in the pdf-writer software (b) problematic Mediawiki markup (c) table is too wide

Map ModuleMapFeature ObjectsCoveragesMoreMap Display OptionsMap Module ToolsMap Module Menus Feature objects in SMS have been patterned after Geographic Information Systems (GIS) objects and include points, nodes, arcs, and polygons. Feature objects can be grouped together into coverages. Each coverage defines a particular set of attributes that store information for the objects. Since feature objects are patterned after GIS objects, it is possible to import data from GIS applications such as ESRI Shapefiles (Arc/Info or ArcView) and MIF/MID file pairs (Map Info). The primary use of feature objects is generate high level conceptual representations of a site. The area included by the polygons defines the domain of the mesh, grid, or limit the extents of cross sections. Each polygon represents a material zone or element type. Special points can be identified in the interior of the domain as areas of particular interest. Boundary parameters such as flow and head values can also be assigned to points or arcs. Depending on the numerical model to be used, SMS either passes this conceptual representation to the model, or constructs finite element meshes, finite difference grids, or one-dimensional cross sections that a numerical model will use. Thus, the user can focus on a simplified, high level representation of the model and little or not tedious cell-by-cell editing is required. The conceptual model approach can be used to build models for any of the numeric models supported by the SMS interface. Feature Object Types Feature Objects The definition of feature objects in SMS follows that used by typical GIS software that supports vector data. The basic object types are points, nodes, vertices, arcs, and polygons. The relationship between these objects is illustrated in the figure below. Points Points are XY locations that are not attached to an arc. Points have unique ids and can be assigned attributes such as a source or sink. Points are often used to refine a mesh in an area of interest. Points are also used when importing a set of XY locations for the purpose of creating arcs or polygons. ArcsArcs are sequences of line segments or edges, which are grouped together as a single "polyline" entity. Arcs have unique ids and can be assigned attributes such as specified head. Arcs are grouped together to form polygons or are used independently to represent geometrical features such as ridges or channels. The two end points of an arc are called "nodes" and the intermediate points are called "vertices". The vertices in an arc define the shape. As more vertices are added, the shape can be more complex. An arc is split into to arcs by selecting a vertex in the arc and converting that vertex to a node. Two adjacent arcs are merged into a single arc by selecting the node that joins them, and converting it to a vertex. Several tools or utilities are provided for working with arcs. These can be accessed while the select arc tool is active by right clicking in the graphics window. Some of the tools also require that at least one arc be selected. The tools include: Delete the selected (or all) arcs. Filter arc(s) Split arc(s) Offset arc(s) Redistribute vertices Reverse arc direction Smooth arc(s) Transform The standard selection utilities (clear, invert, zoom to selection) Select connected arcsNodes Nodes define the beginning and ending XY locations of an arc. Nodes have unique ids and can be assigned attributes. Vertices Vertices are XY locations along arcs in between the beginning and ending nodes. They are used solely to define the geometry of the arcs. Vertices do not have ids or attributes. PolygonsPolygon with Hole Polygons are a group of connected arcs that form a closed loop. A polygon consists of one or more arcs. If two polygons are adjacent, the arc(s) forming the boundary between the polygons is shared (not duplicated). Polygons may not overlap. However, a polygon can have a hole(s) defined by having a set of closed arcs defining interior polygon(s). An example of a hole is shown in the figure below. In this case, four arcs define two polygons. Polygon A is made up of arcs 1, 2, 3 and 4, whereas polygon B is defined by a single arc (arc 2). For polygon A, arcs 1, 3, and 4 define the exterior boundary whereas arc 2 defines a hole. Polygons have unique ids and can be assigned attributes. Polygons are used to represent material zones such as main channel, overbank flood plain, lakes, etc. Coverages Feature objects are grouped together into coverages. Each coverage represents a particular set of data. For example, one coverage can be used to define recharge zones, and another coverage can be used to define zones of hydraulic conductivity. Conceptual Models Coverages are grouped into conceptual models. Conceptual models may consist of multiple coverages. In simple cases like TABS (RMA2), an "RMA2" meshing coverage may be combined with an "Area property" coverage to define material zones. When converting an RMA2 coverage to a mesh, SMS allows the user to specify this option. For more complex conceptual models, such as those used for TUFLOW, a simulation entry in the tree includes links to the component coverages. A TUFLOW simulation may have coverages for HX links, cross section, boundary conditions and levies. Related LinksConverting Feature ObjectsFeature Objects MenuShapefiles

---

Attributes in Feature Objects Menu

Feature object attributes are dependent on the coverage type. If a feature object has attributes, the attributes are edited by selecting the feature object and then selecting the menu command *Feature Objects* | **Attributes** or select **Attributes** from the right-click mouse menu.

Feature Point Attributes

Generic Coverage Types

- Observation
- Stamping

Model Coverage Types

- ADCIRC
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- Generic 2D Mesh
- PTM
- TABS
- TUFLOW
  - 2D Flow Constriction Shape
  - BC
  - Network
  - Water Level Points

  - Z Shape

Feature Arc Attributes

Generic Coverage Types

- Observation
- Stamping

Model Coverage Types

- ADCIRC
- BOUSS-2D
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- GenCade
- Generic 2D Mesh
- PTM
- STWAVE
- TABS
- TUFLOW
  - 1D/2D Connections
  - 2D Flow Constriction Shape
  - 2D Z Lines/Polygons (Simple)
  - 2D/2D Linkage
  - BC
  - Cross Section
  - Grid Extents
  - Network
  - Water Level Lines
  - Z Shape

Polygon Attributes

Generic Coverage Types

- Area Property

Model Coverage Types

- ADCIRC
- BOUSS-2D
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- Generic 2D Mesh
- PTM

- STWAVE
- TABS
- TUFLOW
  - 2D Flow Constriction
  - 2D Z Lines/Polygons (Simple)
  - 2D Spatial Atts
  - 2D/2D Linkage
  - BC
  - Grid Extents

Related Topics

- Feature Objects Menu

Map Module Selection

Select By

The *Select By* option allows the user to select an object by its Material Type, Dataset Value, Area, Length, or Ambiguous Gradient.

Select Connected Arcs

As conceptual models become more complex, and include many arcs (possibly hundreds or thousands), detailed connectivity may not be visibly obvious. This means that you may create or import arcs defining model extents or other features, that appear to be connected, but in actuality are not.

The *Select Connected Arcs Turning Left* utility allows a user to easily determine if a conceptual model has gaps in connectivity. The utility is used by right-clicking on a feature arc and selecting a **"Select Connected Arcs Turning Left"** command. There are two options for selecting the arcs, Forward and Backward. The Forward option follows the direction of the arc and the Backward option follows the opposite direction. This utility selects a string of connected arcs. If more than two arcs connect at a node, the utility selects the left-most turn. In a completely defined polygon, this will select all the arcs in the polygon and traverse right back to the starting arc. If the select polygon tool fails to operate as expected, this utility may identify the gaps in connectivity causing the polygons to not be defined correctly by the build polygon command.

Select Connected Arcs Turning Left

This feature allows the user to select specific arcs attached to an arc already selected determined by the direction of the arc. The user can specify if they want to select the arc turning left from the perspective of going forward, or the arc turning left from the perspective of going backward. Both are opposing directions.

3.7.c.1. Feature Object Creation

Digitize

When you digitize in the map module, elevations are assigned as with other digitization in SMS. That means that when you create a node, point, or vertex, it is assigned the default elevation value for digitization. The default elevation is initialized to 0.0. The default changes any time you specify a Z-value. Therefore, if you create a map point or node, and specify an elevation for that selected point, the value you specify is now the default value for newly digitized points, nodes and vertices. (Note: when you create mesh nodes, there is an option to ask for an elevation each time you create a node, but this option is not available for scatter vertices or map module objects.)

If you would like to digitize aspects of an image (*.tif, *.jpg, ...), simply load the image file into SMS and select the desired tool from the Map module tools (**Create Feature Arc**, **Create Feature Point**, etc.) and click over the part of the image that you would like to digitize.

Related Topic

- Map Module Tools

Build Polygons

While most feature objects can be constructed with tools in the Tool Palette, polygons are constructed with the **Build Polygons** feature object command. Since polygons are defined by arcs, the first step in constructing a polygon is to create the arcs forming the boundary of the polygon. All closed loops will be formed into polygons. Before defining material zones or creating meshes from a coverage, the **Build Polygons** command must be used first.

Polygon Attributes

The *Polygon Attributes* dialog is used to set the attributes for feature polygons. Attributes that can be specified for each polygon include:

- Polygon Type – The polygon type can be set to either *Land* or *Ocean*.

For models that generate meshes, there are certain polygon attributes that need to be set before the mesh can be created. For more information, see 2D Mesh Polygon Properties.

3.7.c.2. Feature Object Modification

Feature Object Modification: All

Tansform

The **Transform** command is used to move scatter points. The user is asked which will be transformed, the active set or all sets. In the dialog that appears, the transformation type can be chosen and then appropriate parameters can be entered.

- Data can be scaled, translated, rotated
- Depths/Elevations can be converted back and forth

For more information, go to Data Transform.

Clean

The **Clean** command is used to fix errors in feature object data (Feature Objects menu, Map module).

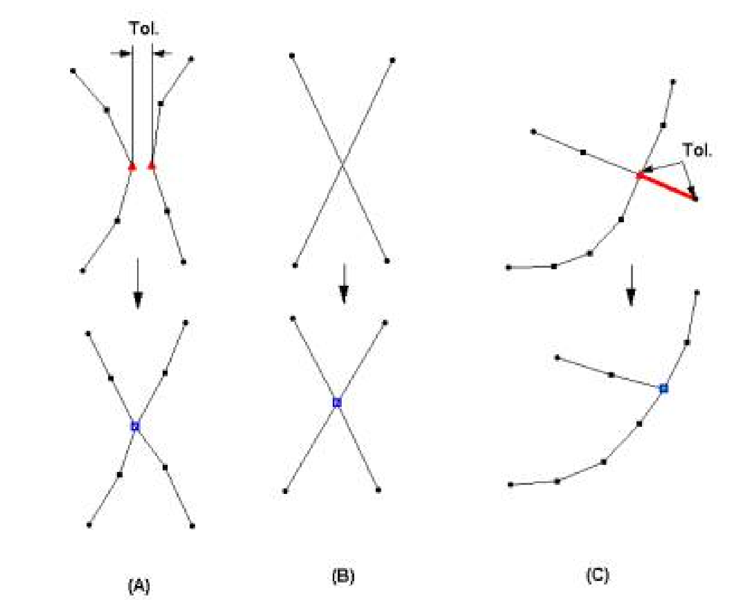


Figure C: (A) snapping (B) intersecting (C) dangling arcs

- **Snap nodes –** This command will merge any two nodes or vertices together if they are within the Tolerance of each other. The new node will be placed at the location of one of the old nodes or vertices (see Figure A).
- **Snap selected nodes –** This command merges two or more selected nodes or vertices. The user must click on one of the selected points which will be treated as the new location (see Figure A).
- **Intersect arcs –** This command places a node where two arcs intersect. All intersections are fixed with this command (see Figure B).
- **Intersect selected arcs –** This command again places a node where two arcs intersect, but only selected arcs are checked.
- **Remove dangling arcs –** The user specifies a Tolerance and all dangling arc segments (at least one end of the arc is not connected to another arc) are deleted if their length is less than the tolerance (see Figure C).

Delete

Delete all feature objects including all coverages and entities in the coverages. A new, empty coverage is created because there must always be a coverage in SMS. This does not delete Drawing Objects, DXF, or images.

Converting Coverages

Map coverages can be converted to other data types in SMS such as meshes, grids, scatter sets and cross sections. This can be accomplished by either right-clicking on a coverage in the project explorer and selecting a convert command or by selecting the following commands from the Feature Objects menu:

- Map → 2D Mesh
- Map → 2D Grid
- Map → Scatter

For more information, see Converting Feature Objects.

Converting Feature Objects

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

Feature objects can be converted to other data types in SMS such as meshes, grids, scatter sets and cross sections. This can be accomplished by either right-clicking on a coverage in the project explorer and selecting a convert command or by selecting the following commands from the Feature Objects menu:

Extract Cross Section

The **Extract Cross-sections** command uses the cross section arcs and a digital terrain model (TINs are the only source that can currently be used) to extract the elevations at vertices of the feature arc cross-sections, or at the intersection points with the triangles.

Cross-sections for individual arcs may be extracted by selecting the arc(s) before choosing the **Extract Cross-sections** command. If not cross-sections are selected then the *Use All Cross-sections* option is used.

Point properties (thalweg, left bank, right bank) can be defined from a 1D-Hydraulic Centerline coverage, or by AutoMark. The *AutoMark* option will examine the elevations of the extracted cross sections and try to infer the thalweg (low point) and the left and right bank points (change of slope) automatically.

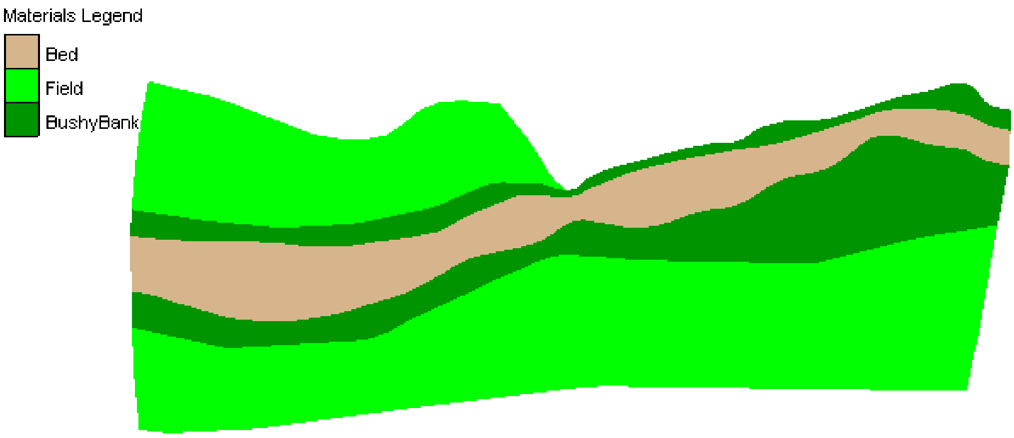
Line properties can be determined from an area property coverage by intersecting the cross-section arcs with the area property polygons and marking them in the cross section database.

Cross Section Database

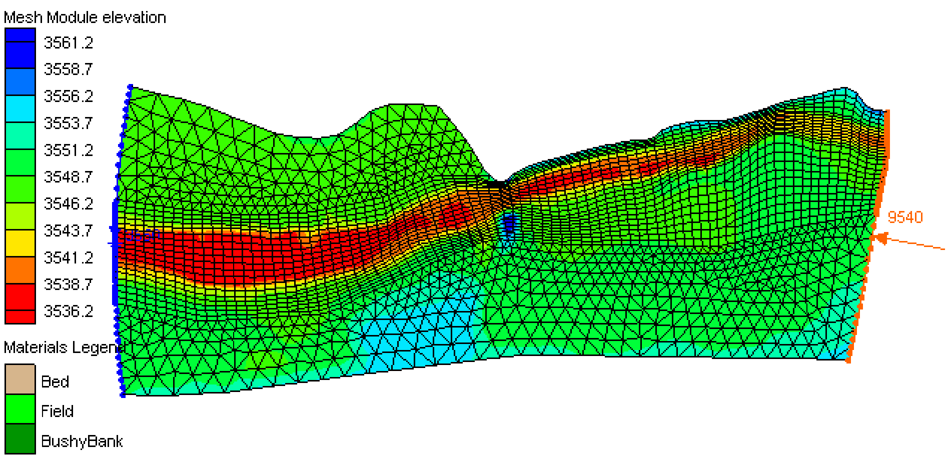
When extracting the cross sections you will be prompted for the name of a cross-section database file. SMS stores all of the cross-section information in a text database file. The cross section database can also be edited independently using the *Cross Section Editor* tools. Extracting cross sections with feature arcs is only way to generate cross-section information, they also can be imported from spreadsheet files (cut and paste), or entered manually.

Map to 2D Mesh

Once a set of feature objects has been created for a coverage (conceptual model) associated with a finite element based model such as RMA2, FESWMS, ADCIRC or CGWAVE , the **Map → 2D Mesh** command can be used to generate a 2D finite element mesh from the objects. The **Map → 2D Mesh** command creates a 2D Mesh on the interior of all of the polygons in the current coverage. The figure domain of a flood plain using the feature objects in the Map Module. The second figure shows a 2D Mesh created from the polygons.



Flood Plain Domain



*2D Mesh created from Map Objects using **Map → 2D Mesh** command*

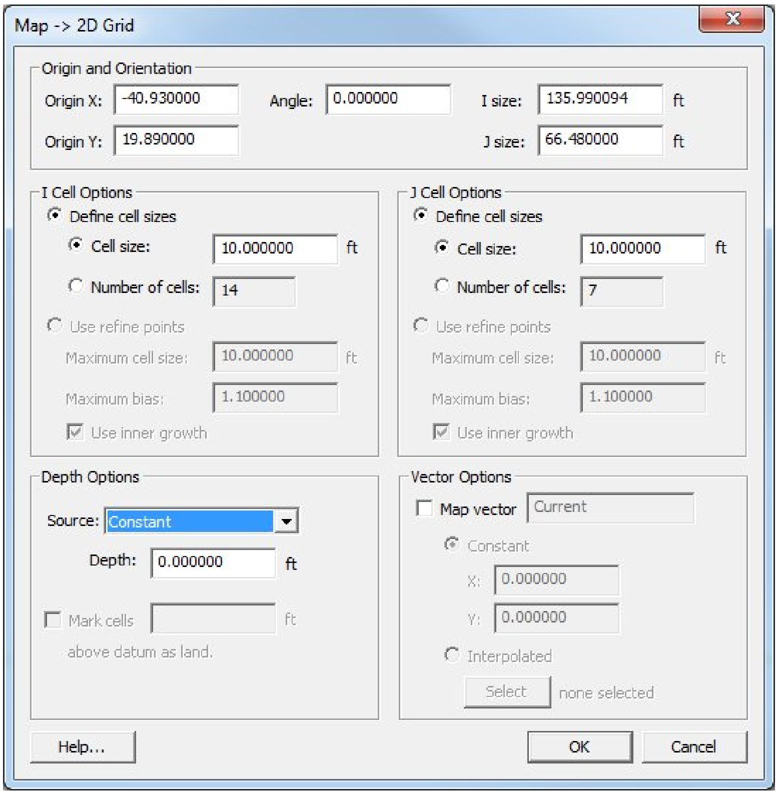
The recommended method for creating unstructured grids (meshes) in SMS for use with either finite element or finite volume engines is to use the conceptual modeling approach. This method includes the following general steps:

1. Define a bathymetric source (scatter set or raster/DEM).
2. Define a map module coverage consisting of polygons that cover the modeling domain. This is the region to be covered by the mesh.
3. Assign attributes to the points/arcs/polygons in the coverage to control the mesh characteristics.
   - Point meshing attributes:
     - Used to force the creation of a mesh node at a specific location.
     - Used to specify the element density in the area of the point location by assigning refine point attributes.
   - Arc meshing attributes:
     - Used to define linear features such as a river thalweg or an embankment toe/shoulder. Mesh nodes will be created along the arc.
     - Used to control element density if a size function (scalar paving) is not utilized. Vector spacing on the arc controls mesh node spacing for all mesh generation options except scalar paving.
   - Polygon meshing attributes:
     - Specify a bathymetry source for each polygon
     - Specify a meshing type for each polygon. Choose from:

  - Patching – create quad dominant elements conforming to a topographic rectangle.
  - Paving – create triangular elements layer by layer from the polygon boundary inward.
  - Scalar Paving – create triangular elements as with paving with the spacing controlled by a size function defined on an associated scatter set
4. Optionally, define an area property coverage to define the source of material attributes.
5. Issue the **Map → 2D Mesh** command is used to create a 2D mesh using the feature objects in a 2D Mesh Coverage. When the **Map → 2D Mesh** command is selected, the 2D Mesh Options dialog opens.

Map to 2D Grid

The **Map → 2D Grid** command is used to create a 2D grid using the feature objects in a 2D Grid Coverage. When the **Map → 2D Grid** command is selected, the *Create Grid* dialog appears. A grid frame must have been defined. The size and location of the grid frame are used to initialize the fields in the *Create Grid* dialog. In most cases, these values will not need to be changed and the user can simply select the **OK** button to create the grid. If a grid frame has not been defined, the size and location of the grid are initialized so that the grid just surrounds the currently defined feature objects. If desired, the grid dimensions can be edited prior to selecting the **OK** button to create the grid.



Map → 2D Grid dialog

Grid Frame Properties

The grid frame properties dialog allows the user to specify the attributes applied to the grid frame when performing a **Map → 2D Grid** operation. These properties are as follows:

- Origin
- Orientation
- Directional properties (u and v direction)
  - Define cell sizes – specified uniform cell sizes
    - Cell size – the cell size in the specified direction
    - Number of cells – number of cells in the specified direction
  - Use refine points – refine points will be used to generate the grid
    - Maximum cell size – the max size the should exists when growing
    - Maximum bias – the max growth ratio to be used when growing
    - Use inner growth – specifies whether the cell sizes should grow between two refine points
  - Grid size – the grid dimension in the specified direction

When the user specifies "Define cell sizes", there are a few options available. These options are:

1. Specify cell size – specify the cell size and the number of cells will be computed.
2. Specify number of cells – specify the number of cells and the cell size will be computed.

If the grid is to have square cells, the v direction cell size will always be linked to the u direction cell size.

Refine Points

Refine points for a Cartesian Grid allow a user to change the cell dimensions when generating the grid. They are not available for all models, since some Cartesian Grid models require uniform cell sizes. The user can specify whether to refine in the I and/or J direction and the base cell size for each direction.

When the refining is performed, the base size may be changed in order to fit the other restrictions applied to the refining process. If two refine points are too close to each other to allow the cell size to transition, one will be ignored when generating the grid. See *Refine Point Dialog* for more information.

Depth and Vector Options

In addition to the options specified on the grid frame, depth and vector interpolation options can be specified during the mapping process for some models. The depth mapping is required for all models, while the vector mapping is optional even for the models it can be performed on. Depth and vector datasets can be constant or interpolated from a scatter set.

Cells a user specified tolerance above the datum can be marked as land (inactive) cells. This option is on by default for BOUSS-2D, but defaults to off for other models.

When specifying a constant vector, the X and Y components are oriented based on global space, not grid space.

The name of the vector dataset can be specified, but the name of the depth dataset is always set to "Depth".

Map → 2D Scatter Points

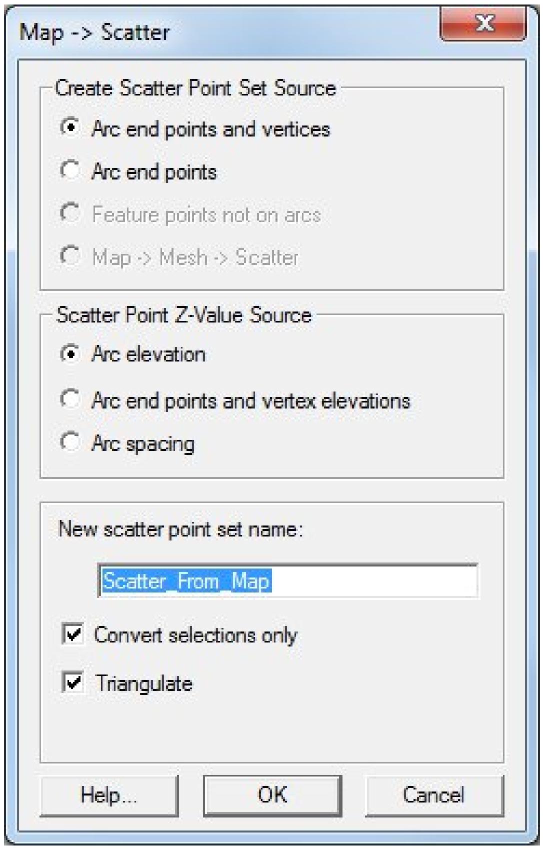
The **Map → 2D Scatter Points** command creates a scatter point set from the points and nodes and vertices of the current coverage. The process is different for observation coverages and non-observation coverages.

- **Non-observation Coverages**

  With non-observation coverages, a single elevation dataset is created for the 2D scatter points representing the Z location of all the points, nodes and vertices.

- **Observation Coverages**

  With observation coverages, the *Observation Points → Scatter Points* dialog appears. This dialog allows you to create a dataset for the 2D scatter points from one of the measurements associated with the observation points.



Map → Scatter dialog

Measurement

A dataset is created for the 2D scatter points from the measurement selected in the dialog. The model associated with the selected measurement (if any) is shown, along with whether the measurement is steady state or transient.

Time Step Times

This section of the dialog is only available if the selected measurement is transient. It allows you to define the number of timesteps, and the timestep times to be created for the scatter point dataset.

- **Match all unique times**

  The Match all unique times option gets the set of unique times from the XY series of all the observation points. This is the union of all the times. If some XY series use dates/times and others don't, this option won't be available. Otherwise, the times in the spreadsheet will be displayed as either dates/times or relative times depending on the XY series. The spreadsheet will not be editable. The Use dates/times toggle will be unavailable but set according to whether the observation point XY series use dates/times or not. The Reference time section will be unavailable, but if the XY series use dates/times, the minimum time will be used as the reference time for the scatter point dataset.

- **Match time steps from model**

  The Match time steps from model option will only be available if the measurement is associated with a model, and the model is transient. If so, this will be the default choice and GMS will get the times to display in the spreadsheet from the stress period and time step info for the model. The spreadsheet will not be editable. The Use dates/times toggle will be unavailable but set according to whether the model uses dates/times or not. The Reference time section will be unavailable, but if the model uses dates/times, the model reference time will be used as the reference time.

- **Specify times**

The spreadsheet of times will be editable with this option and you can copy and paste times from another program such as a spreasheet. Also, the **Initialize Times** button becomes available allowing you to bring up a dialog you can use to create times at a specified interval. If you select the Use dates/times toggle, the Reference time section will become available and the times in the spreadsheets will be displayed as dates/times.

Related Topics

- Map Feature Objects Menu

Unstructured Grid Generation from a Conceptual Model

Traditionally, the most time consuming component of using a multi-dimensional hydrodynamic numerical model has been the generation of unstructured grids (also called meshes). This effort has given models based on Cartesian grids (structured grids) a decided simplifying advantage. Digitizing node points and connecting them into elements, while seemingly not a complicated process, becomes overwhelming when you consider the number of nodes and elements that compose a numeric simulation (thousands to even millions and the number is still growing).

The SMS interface includes the capability to define a 2D mesh using the feature objects in a 2D Mesh Coverage. When the **Map → 2D Mesh** command is selected, the *2D Mesh Options* dialog opens. A meshing polygon must have been defined prior to issuing this command. The attributes of the meshing polygon(s) are used to generate the 2D mesh.

The meshing options are used with coverages that generate meshes for specific numeric engines. Some of the options may not be available for all coverage types since some models have specific requirements such as a limited number of supported element types.

Feature Polygon attributes

The process of generating a mesh involves filling the polygons in the coverage with elements. These elements can be triangular or quadrilateral depending on the numeric engine they will be used with. The user specifies how the polygons will be filled choosing from the following "Mesh Type" options:

- None – there will be no elements in this polygon. This will represent an island in the domain.
- Patch – the mesh is topologically a triangle or rectangle that will be filled with elements that conform to its sides.
- Paving – the mesh will be filled with elements by offsetting from the boundaries. The distribution of the vertices on the arcs comprising the polygon control the mesh density.
- Scalar Paving Density – the mesh will be filled with elements using the paving approach, but the distribution of vertices along the arcs, and throughout the interior, will be controlled by a scalar dataset specifying a target edge length.
- Existing Nodes – the polygon will be used as a stencil to keep the mesh nodes and elements already in that region.

In addition to specifying the method to fill the polygon, the user also specifies the source of bathymetric or topographic elevation data for the newly constructed mesh using the "Bathymetry Type" controls. The user can choose from:

- Constant – all newly created nodes will be assigned a single specified value.
- Scatter Set – all newly created nodes will be assigned a value based on interpolation from a scatter set or TIN.
- Raster – all newly created nodes will be assigned a value based on interpolation from a Raster object.

- Exiting Mesh – all newly created nodes will be assigned a value based on interpolation from the previously existing mesh.

The user also has control to specify the material type that will be assigned to newly created elements.

Feature Arc attributes

The feature arcs in the conceptual model serve three purposes.

1. They carry boundary condition attributes for the specific model or engine.
2. They control final mesh density for the paving or patch options.
3. They control detailed feature maintenance when they lie inside of paved polygons. An arc representing a thalweg (channel), crest, ridge, or shoulder will be incorporated into the meshing pattern to ensure that these features are maintained.

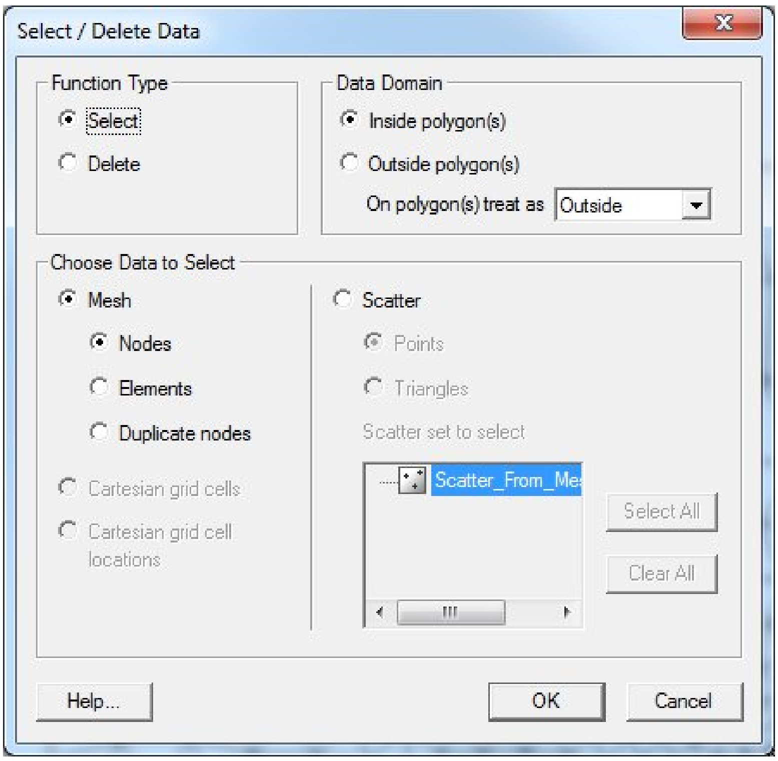
Feature Point attributes

Feature points can be included in polygons to carry a boundary condition such as a source or sink, or they can be used to control resolution in a specific area. The feature point can be assigned an attribute to be a refine point. In this case, the user specifies the size of element around that location. The mesh generation process will generate an element or cluster of elements at that location matching the specified size. These are then incorporated into the surrounding mesh using the advancing front paving method.

Feature points can also be assigned an attribute to control whether a mesh node will be incorporated into the mesh at the exact feature point location.

Select/Delete Data...

The **Select/Delete Data...** command (*Feature Objects* menu, Map module) is available when one or more polygons are selected. The user may select or delete data that is located inside or outside of selected polygons. Options are provided to select or delete mesh node, elements or duplicate nodes, scatter points or triangle, Cartesian grid cells or cell location that are partially (triangles/elements that cross a boundary) or completely inside/outside the polygons.



Select/Delete Data dialog

Function Type

- **Select** – Select objects
- **Delete** – Delete objects

Data Domain

- **Inside polygons(s) –** Trim data inside selected polygons.
- **Outside polygon(s) –** Trim data outside selected polygons.
- **Treat boundary as [Outside | Inside] –** Treat data that lies on polygon boundaries as if it were outside/inside of the polygon(s).

Select Data Type

Choose the data type to select or delete

- **Mesh**
  - **Nodes** (Select) – Select mesh nodes only.
  - **Nodes and Elements** (Delete) – Trim mesh nodes and elements.
  - **Elements –** Select or delete mesh elements only.
- **Scatter**
  - **Points** (Select) – Select scatter points only
  - **Points and Triangles** (Delete) – Trim scatter points and triangles.
  - **Triangles –** Trim triangles only.
  - **Scatter Sets to Trim –** Click on a scatter set in the window to toggle it on/off for trimming. Push the **Active Set** button to only select the active scatter set and the **All Sets** button to turn on/off all sets. Points and/or triangles are deleted only from the selected sets.
- **Cartesian grid cells**
- **Cartesian grid cell locations –** this can be useful in working with elevation values in TUFLOW grids.

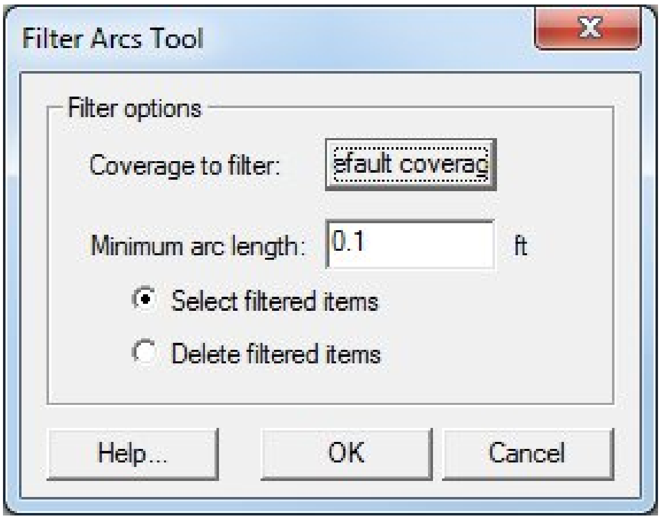
Related Topics

- Map Feature Objects Menu

Arcs

Filter Arc

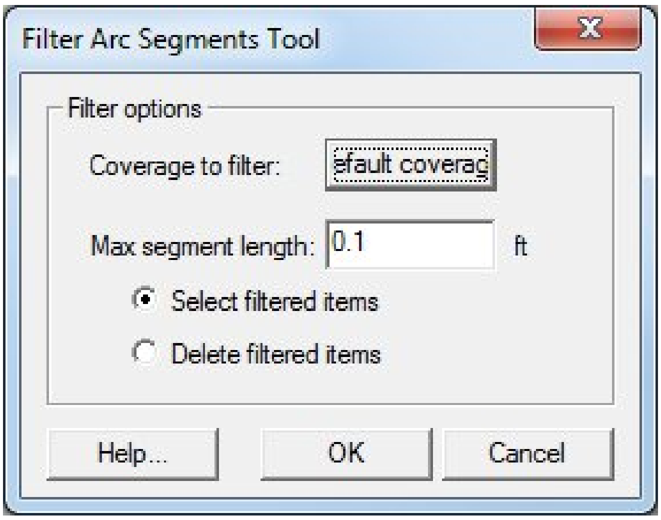
The *Filter Arcs Tool* dialog is accessed through the right-click menu of a selected arc.



Filter Arcs Tool dialog

Filter Arc Segments

The *Filter Arcs Segments Tool* dialog is accessed through the right-click menu of a selected arc.



Filter Arc Segment Tool dialog

Split Feature Arcs Utility

Arcs are used to represent features in a conceptual model. A feature arc may represent a very small (short) individual feature, or it may represent a convoluted complex feature such as an entire shoreline. Two sample applications include arcs representing a shoreline, or arcs representing the thalweg of a river. There are advantages to having a single arc represent a long feature. These include the ability to select the entire feature easily, it is less cumbersome to manage the feature because there are less features, and you can ensure that there are no gaps in the feature because it is a single arc. However, there are also applications and purposes for splitting the arc into pieces. One such example is illustrated by the change of shape an arc undergoes when it is redistributed. The vertices move, but the feature nodes do not. In this situation, making key points nodes prevents the arcs shape from loosing its ability to represent these key locations. In the case of a thalweg, it could be advantageous to have a separate arc for each section of the river. Each arc would then span from one river station to another (perhaps every 100 yards or meters, or each river mile).

This utility is in the right-click menu when you select one or more arcs. The command brings up the *Split Arcs Tool* dialog. In this dialog the user provides criterion for splitting the selected arc(s) into multiple arcs. If the user desires to process all the arcs in a coverage, the user can **select all** first.

There dialog includes three toggle boxes associated with criteria including:

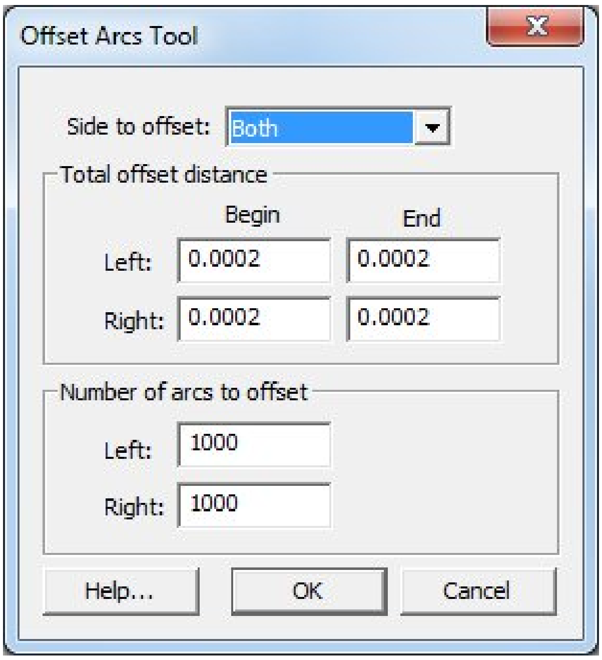
- **Split long arcs** – if this is selected, the user specifies a length (labeled as ft or m) based on the current projection. Processed arcs will be split into arcs of this specified length starting at the first of the arc. The last arc in the group will generally not be the specified length.
- **Split sharp corners** – if this is selected, the user specifies an angle (labeled as degrees). This is the angle of deviation from one arc segment to the next. A straight line has an angle of deviation of 0.0. If the arc doubled back on itself completely, the angle of deviation is 180 degrees. This deviation is a magnitude. A left turn is processed identically to a right turn. Processed arcs will be split at vertices where the arc direction changes more than the specified angle.
- **Split long segments** – if this is selected, the user specifies a threshold length. If a segment is longer than this threshold, the vertices at either end will be converted to nodes making it a separate arc.

If multiple toggles are selected, the arc will first be split based on maximum arc length, the resulting arcs will be split based on angle, and finally the resulting arcs will be processed for long segments. This will retain the feature of creating feature nodes at the key length locations, and then add additional feature nodes at bends.

Offset Arc

The **offset arc** command (arc right-click) provides a mechanism to create additional arcs in a coverage by offsetting from the selected arc(s). Multiple offsets can be created in a single command. This command brings up a dialog that asks the user for a number of parameters including:

- The direction of the offset (to the left of the arc, to the right of the arc or both).
- The number of arcs to create in the specified direction.
- The maximum offset in the specified direction. If more than one offset is created on a side, the offset provided is for the last offset and the others are interpolated between the original arc and the specified offset. (If to arcs are created on the left side with an offset of 10.0, the first is offset by 5 and the second is offset by 10.)



Offset Arcs Tool dialog

This command uses the orientation of the selected arc as a basis meaning it knows one end of the arc as the "start" and the other as the "end". This convention is determined based on the creation of the arc. This direction is not always obvious and the user may need to investigate using trial and error which end is the start and which end is the end (as with bias in the redistribute command).

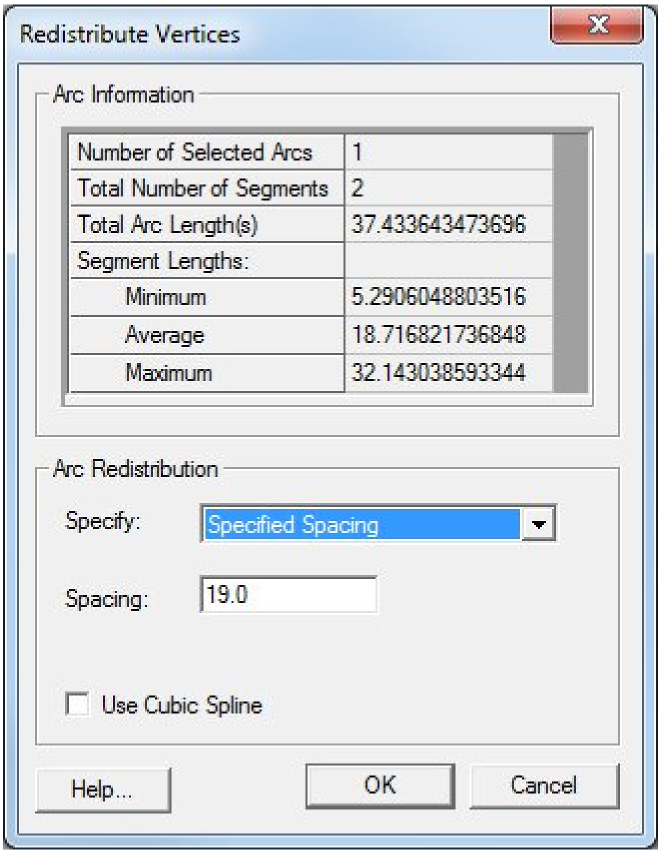
This command has several useful applications. These include:

- When defining a conceptual model, the base line data may include a feature such as a thalweg or a toe of a bank, or a center-line of a road or levee. The **offset arc** command can be used to approximate the opposing bank, a shoulder from a toe or similar, nearly parallel feature.
- When creating arcs that define a tropical storm path, variations of that path may be desired. The **offset arc** command creates arcs to represent these paths.

Redistribute Vertices

The primary function of the vertices of an arc is to define the geometry of the arc. If the arcs are to be used for automatic mesh generation, the spacing of the vertices is important. The spacing of the vertices defines the density of the elements in the resulting mesh. Each edge defined by a pair of vertices becomes the edge of an element. The mesh gradation is controlled by defining closely spaced vertices in regions where the mesh is to be dense and widely spaced vertices in regions where the mesh is to be coarse.

When spacing vertices along arcs, the **Redistribute** command in the *Feature Objects* menu (or using the right-click menu) can be used to automatically create a new set of vertices along a selected set of arcs at either a higher or lower density. The desired arc(s) should be selected prior to selecting the **Redistribute** command.



Redistribute Vertices dialog

The current status of the selected arc(s) is given at the top of the dialog. This includes the number of segments and spacing of those segments. When multiple arcs are selected, the current status is a combination of all selected arcs. However, the parameters set in this dialog apply to each arc individually. Therefore if multiple arcs were selected, each arc would reflect the options selected in this dialog.

The following options are available for redistributing vertices:

Linear Interpolation

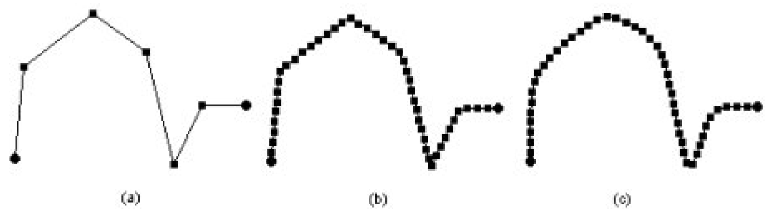
If the *Linear interpolation* options is specified, then either a number of intervals or a target spacing can be given to determine how points are redistributed along the selected arcs. In either case, the new vertices are positioned along a linear interpolation of the original arc. The arc may change shape due to the fact that original vertices are removed as the new vertices are created. This may round corners from the arc.

Spline Interpolation

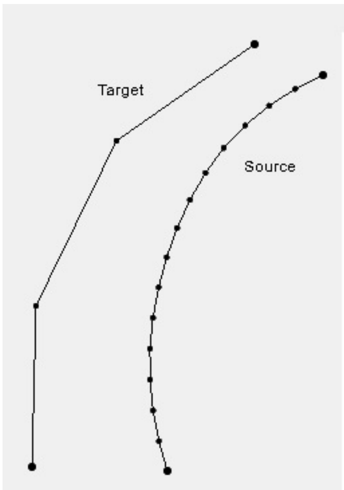
If the *Spline interpolation* option is specified, vertices are redistributed along a series of cubic splines defined by the original vertices of the selected arcs. The difference between the linear and spline interpolation methods is illustrated in the figure below.

Source Arc

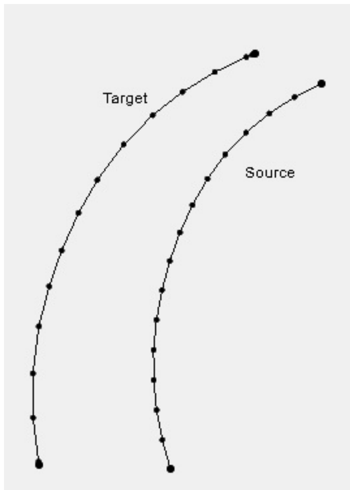
The Source arc option is only available when two arcs are selected. One arc is specified to be the source arc while the other is the target arc. SMS redistributes the vertices on the target arc to be as close as possible to the vertices on the source arc.



Redistributing Vertices: (a) Original Arc (b) Linear (c) Spline



Before Redistribution



After Redistribution

Reverse Arc Direction

This command (*Feature Objects* in the Map module or right-clicking on a selected arc) reverses the direction of all selected arcs. Arc orientation is only important for applications that use coastline or centerline arcs that have an arrow drawn on the arc.

Each arc has a direction. One node is the "from" node, the other node is the "to" node. For most applications, the direction of the arc does not matter. However, when the arc is used to define a observation plots, and in various other situations the direction of the arc becomes significant. The **Reverse Arc Direction** command can be used to change the direction (upstream to downstream) for an arc.

Smooth Arc

This command reduces the variability or roughness of an arc. When an arc is created by digitization, it includes the manual variations and noise from the digitization process. Similarly, when an arc is created to follow a contour line, it can have numerically created corners and bends. Smoothing an arc results in more gradual bends in the shape of the arc. Applying the smooth command repetitively will eventually result in a straight line.

An arc should not be smoothed, if the meanders and bends accurately depict a physical feature. However, smoothing may result in more gradual variations that can enhance numerical stability.

Smooth Arc Tool dialog

The dialog that appears when the **Smooth Arc** command is invoked asks the user to specify the number of neighbors to be included in the smoothing window and a self weight. As the number of neighbors increases and the self weight decreases, the level of smoothing becomes more dramatic (moves to a straight line more quickly).

The algorithm computes a new position for each vertex in the arc from the existing vertex positions. The number of neighbors must be the same both before the vertex being smoothed and after. Since there are no vertices before the first node, and none after the last node, these two locations are not impacted by the smoothing process. The first vertex can be smoothed using the first node from before its position, and the first vertex after its position.

If the self weight is set to 1.0, no influence is assigned to the neighbors, so the arc does not smooth at all. If the self weight is set to 1/3, and the number of neighbors is set to 1, then each of the points has an even 1/3 weight in determining the new vertex location. If the self weight is set to 1/5, and the number of neighbors is set to 2, then five vertices impact the new location (Two vertices before the vertex in question and two vertices after) and the resulting point is the average of those five points (20% self weight, 80% neighbor weight with four neighbors).

Transform

See the article Data Transform

Select Connected Arcs

As conceptual models become more complex, and include many arcs (possibly hundreds or thousands), detailed connectivity may not be visibly obvious. This means that you may create or import arcs defining model extents or other features, that appear to be connected, but in actuality are not.

The *Select Connected Arcs Turning Left* utility allows a user to easily determine if a conceptual model has gaps in connectivity. The utility is used by right-clicking on a feature arc and selecting a **Select Connected Arcs Turning Left** command. There are two options for selecting the arcs, Forward and Backward. The Forward option follows the direction of the arc and the Backward option follows the opposite direction. This utility selects a string of connected arcs. If more than two arcs connect at a node, the utility selects the left-most turn. In a completely defined polygon, this will select all the arcs in the polygon and traverse right back to the starting arc.

If the select polygon tool fails to operate as expected, this utility may identify the gaps in connectivity causing the polygons to not be defined correctly by the build polygon command.

Select Connected Arcs Turning Left

This feature allows the user to select specific arcs attached to an arc already selected determined by the direction of the arc. The user can specify if they want to select the arc turning left from the perspective of going forward, or the arc turning left from the perspective of going backward. Both are opposing directions.

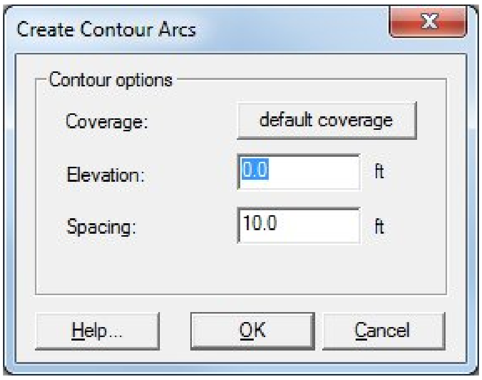
Create Arc Group

The **Create Arc Group** command is found in the *Feature Object* menu of the Map module. By selecting multiple arcs and using this command, the arcs will be made into a group. When selecting any one of the arcs in the group, all arcs in the group will be selected.

Create Contour Arcs

The *Create Contour Arcs* dialog is used to create a feature arc along a specified elevation of the active scatter dataset or cartesian grid elevation. To access the dialog do the following:

- Right-click on a Cartesian Grid or Scatter set in the *Project Explorer*.
- Select **Convert** then **2D Grid Contours→Map** for Cartesian Grid, otherwise select **Convert** then **Scatter Contours→Map**. This brings up the *Create Contour Arcs* dialog. By default the active coverage is selected. The user is prompted for the elevation and vertex spacing along contour.



Create Contour Arcs dialog

The dialog allows the user to specify the following attributes:

- Coverage
- Elevation
- Spacing

Align Arc With Contour

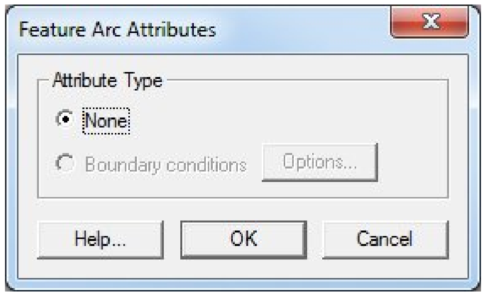
This command (arc right-click) can only be used when the project has scatter data. A dialog is brought up from which a scatter dataset, dataset value, and maximum distance are set. SMS will then move the vertices along the selected arc to locations where the scatter dataset value matches the dataset value specified in the dialog, as long as it is not moved by a distance greater than the maximum distance (also specified in the dialog). This is done by processing each location individually. For each point, SMS extracts the contour value from the TIN/dataset and the gradient of the TIN at the (x,y) location. It then tracks up or down gradient to intersect with the contour at the specified dataset value. If a local extrema is encountered before finding the contour, the operation fails.

Arc Attributes Dialog

The *Arc Attributes* dialog is used to set the attributes for feature arcs. Attributes that can be specified for each arc include:

- Arc Type
  - Generic
  - Percent preference coastline
  - Land preference coastline
  - Ocean preference coastline

The *Arc Attributes* dialog is a dynamic dialog that will provide different options depending on the active model.



Feature Arc Attributes dialog

Recompute All Stations

This function (found in the *Feature Objects* menu) can be used if an arc's "Computational Length" value on a river changes. It sets the Start and End station values for each of the arcs with the same river name based on the computational lengths of each arc. So, for example, if you had 3 arcs in a river, the most downstream arc being 30 feet, the next arc being 40 feet, and the next arc being 50 feet, the starting and ending stations would be as follows for each of the arcs (Arc 1 is downstream):

Arc	Start Station	End Station
1	0.0	30.0
2	30.0	70
3	70.0	120.0

These start and end stations are set when you assign a river name to an arc, so there there should not be a need to use the **recompute all stations** command unless there is a change in the computational length, start station, or end station values for a centerline arc.

Related Topics

- Feature Objects Types
- Map Module
- Map Feature Objects Menu
- 1D River Module Tools

Feature Object Commands

General Feature Object Commands

Define Domain

This option is found in the *Feature Objects* menu, in the Map module, when CGWAVE or ADCIRC is the active coverage type. A domain is the region to be filled by a finite element mesh. An ocean arc is created that connects to the coastline arc. There are several options for defining the domain:

1. User selects one or more coastline arcs. The ocean arc starts and ends at the extreme ends of the chain of arcs.
2. User selects two vertices or two nodes. The ocean arc starts and ends at the nodes.
3. User selects a single node or vertex along the coastline arc. The user defines the radius of the ocean arc and the ocean arc starts and ends where it intersects the coastline arc.
4. User selects a single disjoint point. If the point is inside a closed coastline arc (an island), the user specifies the radius of the ocean arc. The ocean arc is a circle that encloses the island.

After the user selects an arc(s) or point(s), the *Domain Options* dialog appears with the following options:

- **Rectangular –** If the user selects two points or an arc, the user only sets the Offshore Length. If the user specifies a single point, the user sets both Offshore and Along Shore Lengths.
- **Semi-circular –** If the user selects two points or an arc, the user does not specify the radius, the radius is the distance between the points. If the user specifies a single point, the user sets the Radius.
- **Circular –** This option only works if the user selects one point. The user then sets the Radius.

Right-Click Menu

- **Clear Selection (Right-Click Menu) –** The **Clear Selection** command undoes the selection of the object that was clicked on.
- **Invert Selection (Right-Click Menu) –** The **Invert Selection** command selects every object of the same type selected, and undoes the selection of the object that was originally selected.
- **Zoom to Selection (Right-Click Menu) –** The **Zoom to Selection** command zooms to a closer view of the object selected.

Coverage Type Specific Menus

Optional menu items appear according to the active coverage type.

Generic Coverage Types

- **Create Coastline**

  When this command is invoked, the *Create Contour Arcs* dialog opens. For more information, see Arcs: Create Contour Arcs. This command is available if the current coverage type is ADCIRC, or CGWAVE.

- **Stamping**

  See the article Feature Stamping for more information.

Model Coverage Types

- **ADCIRC**
  - Model Control
  - Create Coastline – see above for more information.
  - Define Domain – see above for more information.
- **BOUSS-2D**
  - Create Coastline – see above for more information.
- **CGWAVE**
  - Model Control
  - Create Coastline – see above for more information.
  - Define Domain – see above for more information.
- **CMS-Flow**
  - Create Coastline – see above for more information.
- **CMS-Wave**
  - Create Coastline – see above for more information.
- **GenCade**
  - New Grid Frame – see the article Grid Frame Dialog for more information.

Related Topics

- Map Module

3.8. Mesh Module

Mesh Module

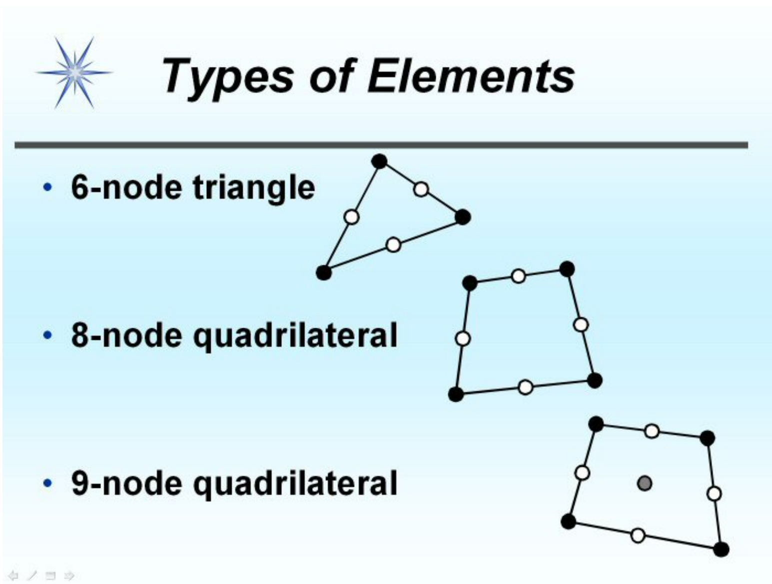
The 2D Mesh Module is used to manipulate 2D unstructured grids (referred to as a mesh inside of SMS). A mesh consists of nodes that are grouped together to form elements. These nodes and elements define the computational domain of the numerical model. In addition to nodes and elements, a mesh may assign additional information to the nodes and elements, such as material values assigned to elements and boundary conditions assigned to nodes. In general, this additional information is used as input data for the numerical model.

Nodes

Nodes are the basic building blocks of elements in a mesh. Nodes store elevation and other dataset values. Nodes can also be used for building nodestrings and assigning boundary conditions. The density of mesh nodes helps determine the quality of solution data and can be important to model stability. See 2D Mesh Nodes Menu for more information.

Elements

Elements are used to describe the area to be modeled. Elements are formed by joining nodes. The element types supported vary from model to model. Element types include:



Mesh Element Types

1D elements

- Three-node line

Triangular Elements

- Three-node linear triangule
- Six-node quadratic triangle

Quadratic (order of solution) Elements

- Eight-node “serendipity” quadrilateral
- Nine-node “Lagrangian” quadrilateral

Water surface and ground elevations are interpolated linearly within each element based on values at the corner nodes. Velocity is interpolated using a quadratic approximation based on values at all the nodes of the element. The quadrilateral elements use identical linear interpolation functions, but their quadratic functions differ because of the presence of an additional node at the center of the nine-node quadrilateral element.

Mesh Datasets

Datasets in a mesh store scalar or vector values at each node. Dataset Active/Inactive areas (generally indicating wet vs dry areas) can be defined on elements or be determined from specific NULL values (often -999) on the nodes themselves. Datasets are used for input data primarily for bathymetry values. Datasets are generally the primary output from a numeric model. Datasets are used with visualization options such as contours and vectors as well as data extraction tools like observation profile plots.

Node/Element properties

In addition to datasets, nodes and elements sometimes need to store additional information generally for model setup. The most common example of this is material types which are assigned to elements.

Nodestrings

A collection of nodes can be formed into a nodestring. Nodestrings are most commonly used to assign boundary conditions such as a flowrate or water-surface elevation. Nodestrings can also be used for mesh renumbering, forcing break lines, and boundary smoothing. Finally, a nodestring can store attributes pertinent to a location such as the total flow nodestring.

Mesh Generation

See Mesh Generation for more information.

Editing a Mesh

Whenever practical a mesh should be reconstructed from a conceptual model rather than edited in the mesh module. Often this isn't an option and a mesh must be edited by hand. The 2D Mesh Module Tools are used to create and edit meshes within the mesh module.

Mesh Visualization

After an analysis, output data at each node of the mesh can be used to generate linear or color filled contours as well as display vector arrows to visualize model solutions. Animations can be generated that shows changes through time for a time-varying solution. Meshes can also be used with flowtrace and multiple view animations.

Advantages of a Mesh

Meshes and other types of unstructured grids (such as TINs) have the advantage they can include a wide range of element sizes and transition gradually between them. Coastal models are often extreme examples of this sometimes with elements as small as a few meters and as large as many kilometers in the same domain. This allows a very large domain while keeping model computation time to a reasonable level.

Mesh Models

SMS has interfaces to several models that use meshes for computations and boundary conditions. These include:

- CGWAVE
- FESWMS-2DH
- ADCIRC
- TABS – (RMA2, RMA4)

In addition to the models with complete interfaces, the mesh module contains the Generic Model interface which allows users to customize SMS to generate data for a user defined model.

The finite difference model TUFLOW uses meshes for post-processing but not for building model domains.

Current Numerical Model

The mesh module is set up to be used with a single numerical model analysis engine at any given time. The current numerical model is changed using the *Data* Menu, *Switch Current Model* menu command. SMS shows only those tools in the tool palette and those menus in the *Menu Bar* which are relevant to the current numerical model. After a finite element mesh has been read, boundary conditions and material properties can be assigned using the commands in the menus associated with the current numerical model. The current model on startup can be changed in the *Preferences* dialog.

Mesh Module Tools

See Mesh Module Tools for more information.

Mesh Module Menus

See 2D Mesh Module Menus for more information.

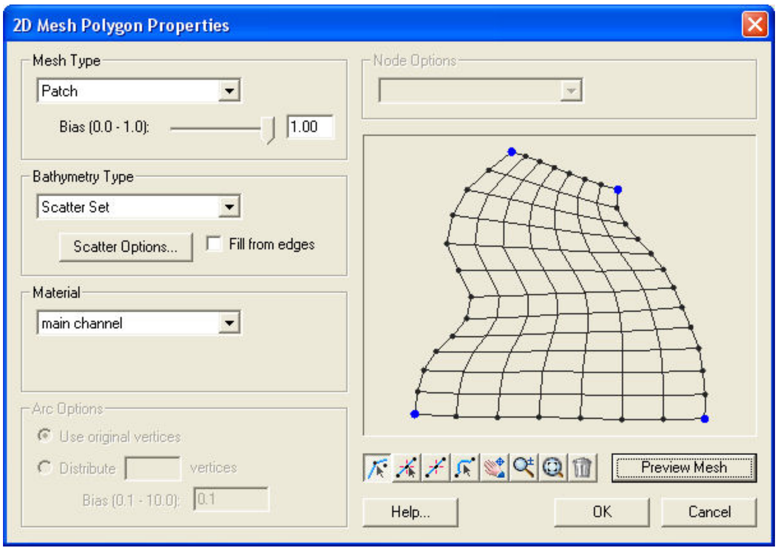
Related Topics

- Creating 2D Meshes
- Editing 2D Meshes
- 2D Mesh Generation
- Mesh Data Menu
- Mesh Module Display Options
- SMS Modules

3.8.a. Mesh Generation

Mesh Generation

At a glance



2D Mesh Polygon Properties dialog

- Generating a quality finite element mesh is central to using many SMS models
- Conceptual models make generating meshes easier
- Polygons can use a variety of meshing options to generate triangular or quadrilateral elements
- Polygons can be assigned bathymetry and material information that will be transferred with the mesh
- Scalar paving density generates elements with sizes based upon a size dataset allowing for smooth transitions and a large range of element sizes and is particularly useful for coastal and wave models.
- Datasets for scalar paving density can be user defined or generated using the data calculator, the create datasets command, or LTEA (linear truncation error analysis) (ADCIRC)

2D Meshes can be created in the following different ways in SMS:

Using a Conceptual Model

This method converts a conceptual model to a mesh using the **Map → 2D Mesh** command. This is the preferred method for mesh generation in SMS.

The mesh generation capability is based on feature objects.

Mesh From Coverage

Each polygon in a meshing coverage (such as TABS, FESWMS, ADCIRC or CGWAVE) includes a mesh type attribute. This attribute defines how SMS should generate nodes inside the polygon and connect them into elements. Individual polygons in the coverage may each utilize their own meshing type. SMS supports the following principal mesh type attributes:

* Patch

* Paving

* Scalar Paving Density

Mesh Generation Toolbox

The mesh generation toolbox is the first pass at a new approach to further automating the mesh generation process. Currently the toolbox includes a single option to generate ADCIRC meshes with automatic density variation. This tool is based on Local Truncation Error Analysis.

Manually Creating a 2D Mesh

In order to create a 2D Mesh in SMS you must have a set of 2D Mesh nodes. Elements can be created by using one of the create mesh element tools and then selecting the mesh nodes to create elements. A 2D Mesh can also be created by triangulating the nodes. The triangulation algorithm assumes that each of the vertices being triangulated is unique in the xy plane, i.e. no two points have the same xy location. Duplicate points can be removed by selecting **Select/Delete Duplicate Nodes** command from the Node menu.

A 2D Mesh can be created manually from the following steps:

1. Select the **Create Nodes** tool from the *Tool Palette*.
2. Create the nodes by clicking inside the *Graphics Window* at the xy coordinates where you want the vertex located. (To change the node location see Editing Node Coordinates)
3. Select a create element tool from the *Tool Palette* ***OR*** Select the *Elements* | **Triangulate** command from the *Mesh* menu to form triangular elements using a Delaunay triangulation [1].

Creating a 2D Mesh from Existing Geometry

TINs, 2D grids, 2D scatter points, and 3D meshes (in GMS) can all be converted to a 2D Mesh. This is accomplished by using the following commands:

- TIN → 2D Mesh
- 2D Grid → 2D Mesh
- 2D Scatter Points → 2D Mesh Nodes
- 3D Mesh → 2D Mesh

After using the **Scatter Points → Mesh Nodes** command you must triangulate the nodes to create the 2D Mesh.

A finite element mesh is defined as a network of triangular and quadrilateral elements constructed from nodes. SMS includes advanced tools to create finite element meshes from underlying bathymetry, meshing parameters and mesh domain limits.

Digitized or survey points can be imported to provide the bathymetry. This type of data is generally not appropriate for use as mesh nodes due to random location and distribution. In this case the data should be converted to scatter points. If the bathymetric points are to be directly used as nodes, the triangulate command can generate elements from the points.

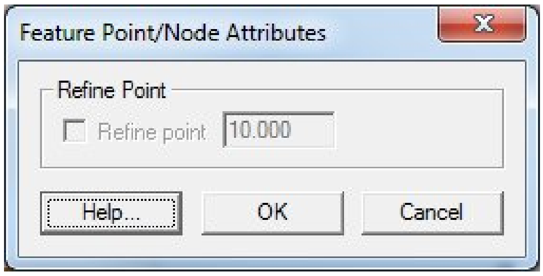
The Map Module provides tools for defining the study area boundaries and features from which a finite element mesh can be created. SMS then interpolates the bathymetry data onto the mesh. This process is also described in Lesson 2 of the tutorials. The Mesh Module provides various Tools for manually editing the finite element mesh.

Related Links

- Mesh Module
- Scatter Data Interpolation
- Adaptive Tesselation
- Advancing Front Triangulation
- Patches
- Refine Points

Refine Attributes Dialog

The *Refine Attributes* dialog is used to set the attributes for a refine point represented by a feature point in a 2D Mesh model coverage. Attributes that can be specified for each refine point include:



Feature Point/Node Attributes dialog

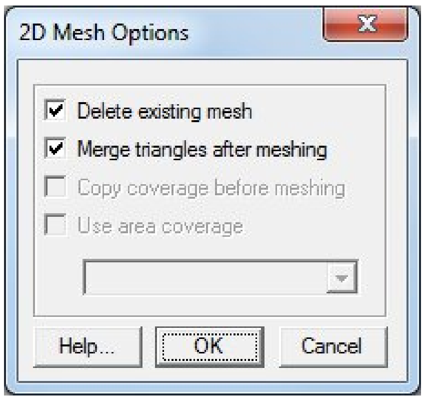
- Refine point (checked = on)
    - Element size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.

Related Topics

- Feature Objects Menu
- Mesh Generation

2D Mesh Options Dialog

The **Map → 2D Mesh** command is used to create a 2D mesh using the feature objects in a 2D Mesh Coverage. When the **Map → 2D Mesh** command is selected, the *2D Mesh Options* dialog opens. A meshing polygon must have been defined prior to issuing this command. The attributes of the meshing polygon(s) are used to generate the 2D mesh.



2D Mesh Options dialog

The *2D Mesh Options* dialog is used to set options for the 2D mesh generation process. The options include:

- **Delete existing mesh** – If checked, the existing mesh will be deleted. If not checked, the new mesh will merge with the existing mesh
- **Merge triangles after meshing** – If checked, triangular elements created during the mesh generation process will be merged into quadrilateral elements where possible.
- **Copy coverage before meshing** – Create a copy of the coverage before the mesh generation algorithm redistributes vertices along the feature arcs defining meshing polygons. Feature arcs are only redistributed when using the Scalar Paving Density mesh generation method.
- **Use area coverage**– This option is only available is at least one area property coverage exists. These coverages consist of polygons with a material type assigned to each polygon. If this option is checked, the material type specified for the polygon in the source coverage is ignored. Instead, each element is assigned a material type based on the polygon the element centroid lies inside of in the area property coverage.

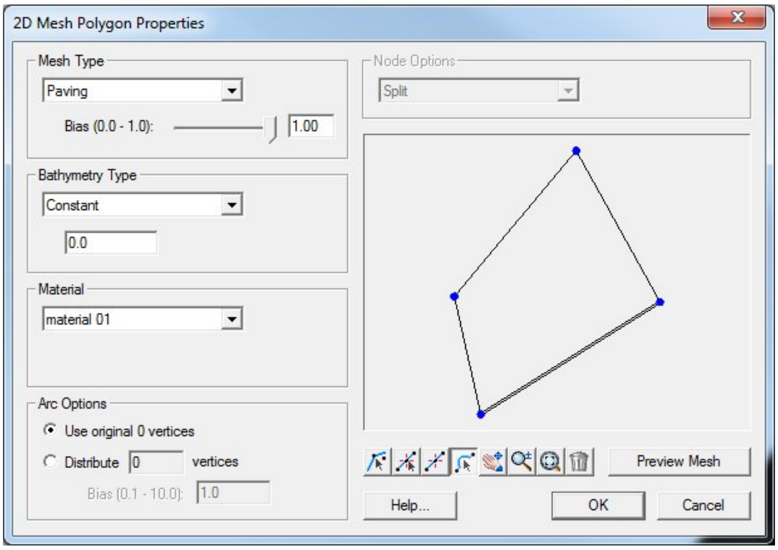
Related Topics

- Mesh Generation
- Map Feature Objects Menu

2D Mesh Polygon Properties

The *2D Mesh Polygon Properties* dialog is used to set meshing options for the conceptual model. See Mesh Generation for a discussion of the conceptual modeling approach.

To access the *2D Mesh Polygon Properties* go to *Feature Objects* | **Attributes**. The *2D Mesh Polygon Properties* dialog will open. Double-clicking inside a polygon will also bring up the *2D Mesh Polygon Properties*.



2D Mesh Polygon Properties dialog

The following polygon attributes can be set:

- **Mesh Type** – Specify the mesh generation algorithm. The following options are available:
  - Patch
  - Paving
  - Scalar Paving Density.
- **Bathymetry Type** – Specify the bathymetry source for assignment to the mesh. The following options are available:
  - Constant – assignes a constant elevation value to all nodes in the mesh
  - Scatter Set – interpolates elevation values from the specified scatter set
  - Existing Mesh – interpolates elevation values from an existing 2D mesh
- **Material** – material type to assign to elements created within the polygon
- **Arc Options** – Used to modify the feature vertices along the arc. The arc must be selected in the polygon preview window. The following options are available:
  - Use Original Vertices
  - Distribute Vertices – Change the number of vertices along the arc. A bias can be specified rather than distributing the vertices with a constant spacing.
- **Node Options** – When the mesh type is set to Patch, the node options are used to define the 3 or 4 sides for the patch mesh generation. If a feature point is selected in the polygon preview window, the following options are available:
  - Split – The mesh generation algorithm will treat the two arcs meeting at the feature point as separate sides.
  - Merge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a single side.
  - Degenerate Edge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a "degenerate edge." When using a degenerate edge, the Patching algorithm will require only 3 sides to be

defined. This option is only valid for meshes which allow triangular elements. Only one degenerate edge can be specified per feature polygon.

Related Topics

- Feature Objects
- Map Module Menus

Advancing Front Triangulation

With advancing front triangulation the polygon is filled in layer by layer. In previous versions of SMS, this has been referred to as Paving. That term is associated with a specific algorithm, so the terminology is being changed here. The process includes offsetting the polygon boundary to the inside of the polygon (or outside of an island), performing intersections on this new offset layer and redistributing the vertices along the offset arc. The process is performed repeatedly until the area is filled with triangles.

Boundary Spaced Advancing Front

In the advancing front methodology utilized by SMS, the new layer position comes from the spacing on the vertices along the current polygon. The advanced front is created by forming equilateral triangles. The vertices on the new arc are redistributed based on the spatial interpolation of the original boundary spacings. This option requires no further input from the users.

Scalar Advancing Front

SMS supports the option to control the spacing between layers of elements using a Size Dataset. This is the scalar advancing front method and requires the user to select a spatial dataset that is everywhere positive to define the local spacing of the desired mesh. This may come from a variety of sources. See the tutorials on mesh generation for CGWAVE and ADCirc for more information.

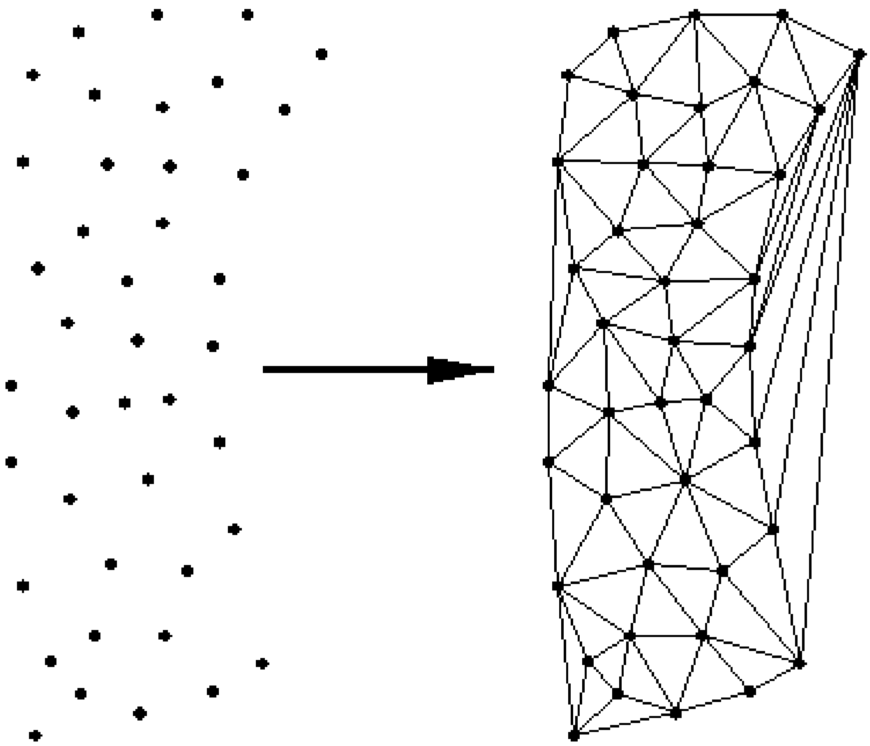
Related Topics

- Mesh Generation
- Adaptive Tesselation
- Patches

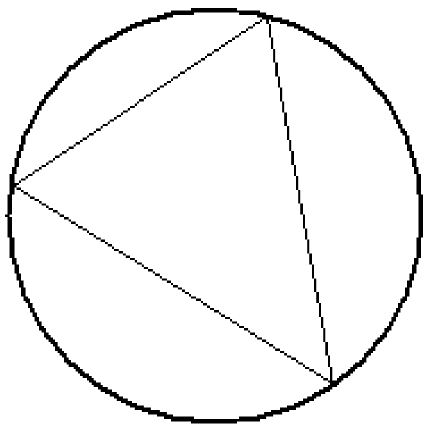
Mesh Node Triangulation

A simple means of creating a large number of elements is to triangulate a set of nodes into a network. This provides a surface that simulates the region being modeled, but normally does not result in quality elements.

This option is available through the **Triangulate** command from the *Elements* menu is executed. The selected nodes are connected with a series of triangles. If nodes are not selected, then all nodes will be triangulated. If linear elements exist, or a linear element creation tool has been selected, then this command creates linear triangles. Otherwise, quadratic triangles are created.



The triangulation algorithm ensures that the Delauney criterion is satisfied. The Delauney criterion is such that the circumcircle of a triangle does not enclose a node on any other element. The circumcircle of a triangle is the circle that passes through its vertices.



Optimize Triangulation

At times, the user will perform manual mesh editing using the **Swap Edge** tool. This makes the Delauney criterion no longer hold. Selected elements can be returned to the Delauney state by choosing the **Optimize Triangulation** command from the *Elements* menu.

Related Topics

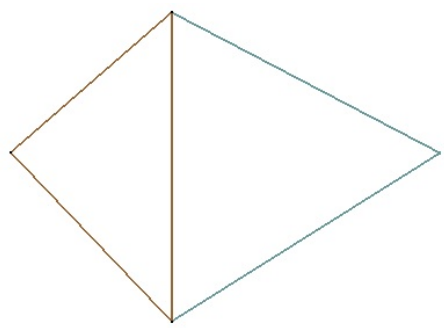
- Boundary Triangles
- 2D Mesh Elements Menu

Merge 2D Meshes

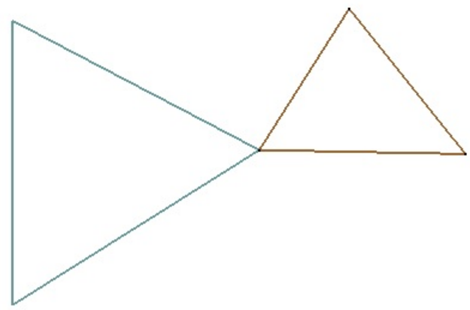
Two meshes can be merged to form a new mesh which will be called "Merged" (or something similar if that name is already used)in the *Project Explorer*. No model data will be transferred, and the type of the new mesh will be the same as the default mesh model set in the user Preferences regardless of the model type of the meshes used in the merge. (To check your user preferences go to *Edit*|**Preferences...** and check the *Startup* tab). This option is accessed by selecting two meshes and right-clicking and then selecting **Merge 2D Meshes**.

A merged mesh will be generated if and only if the meshes do not overlap except on nodes and element edges that are on the border of the other mesh. Overlapping element edges can only be on the border of the mesh and must have nodes at the same location. Disjoint nodes of one mesh that are inside the area of the other mesh will cause the merge to be prevented. A pair of meshes are also invalid for merging if the centroid of an element in one mesh is within the area of an element in the other mesh.

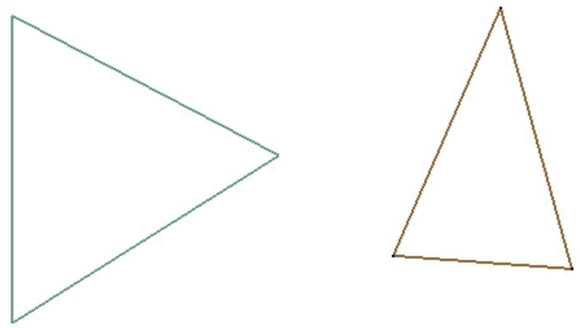
Examples of Meshes that will merge



Example 1: Mesh cells that can merge because they share the same element edge and nodes.

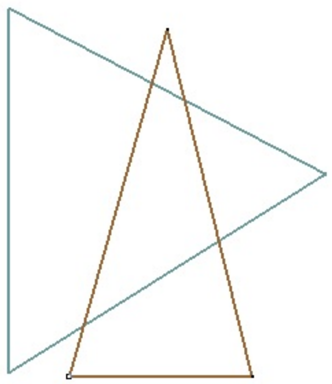


Example 2: Mesh cells that share the same node and do not overlap can be merged.

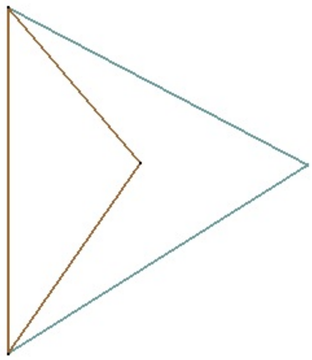


Example 3: Mesh cells that can merge because they do not overlap.

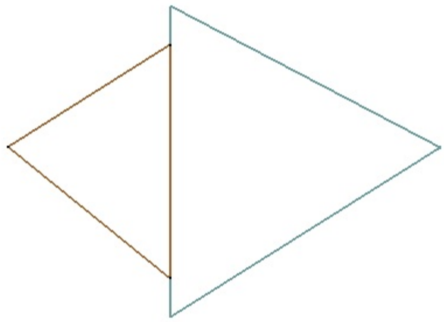
Examples of Meshes that will not merge



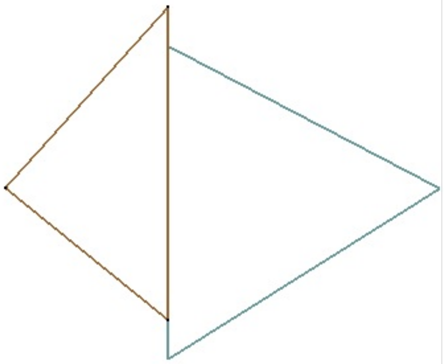
Example 1: Mesh cells that will not merge because they are overlapping.



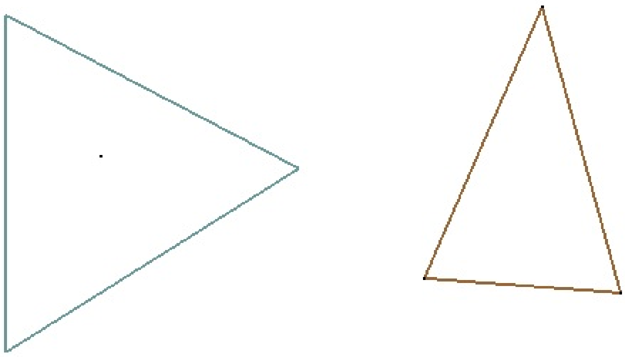
Example 2: Mesh cells that will not merge because they overlap and do not share the same element edges and nodes.



Example 3: Mesh cells that will not merge because they do not share the same nodes in the same location.



Example 4: Mesh cells that will not merge because they do not share the same nodes.



Example 5: Mesh cells that will not merge because a disjointed node of one mesh is with in the area of the second mesh

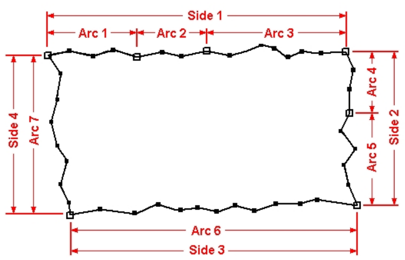
Related Topics

- 2D Mesh Module Menus

Patch

The patch method fills polygons that can be defined as topologic rectangles. The method may combine multiple arcs to define a single side of the patch. A bidirectional coons patching method is used to interpolate from the boundary arcs to interior nodes. Typical applications of this method include river channels and regions aligned to channels. The patching method supports transitioning the number of elements across the channel or from one side to another. Transitions in both directions often result in poorer quality elements and should be avoided if possible.

The Rectangular Patch mesh generation method requires a polygon made of exactly four arcs, ***forming four sides***. However, very rarely do exactly four arcs make up a polygon. SMS provides a way to define a rectangular patch from a polygon that has more than four arcs by allowing multiple arcs to ***act as a single patch side***. An example of a rectangular patch made up of four sides is shown below. Note that Side 1 and Side 2 are both made from multiple arcs. Hollow squares represent the beginning and ending points of an arc (Feature Points). Filled squares represent intermediate points along the arc (Feature Vertices).



Definition of four sides for a rectangular patch

The basic process to define the meshing attributes for a polygon using the Patch method is to:

- Switch to the Map Module
- Select the menu *Feature Objects* | **Build Polygons**
- Switch to the **Select Polygon** tool
- Select the polygon you want to set meshing attributes for
- Select the menu *Feature Objects* | **Attributes**
- Set Mesh Type to *Patch*
- Set Bathymetry Type
- Set Material Type
- Set Node Options if the polygon consists of more than four arcs
  - Switch to the **Select Feature Point** tool in the dialog
  - Select the node you would like to "merge"
  - In the *Node Options* combo box, change the selection to "merge"
  - The two arcs meeting at the "merge" node will now be treated as a single arc for the mesh generation within the current polygon.
- Press **OK** to exit the dialog and save the polygon attributes

Related Topics

- Paving Meshing Algorithm
- Scalar Paving Density Meshing Algorithm
- Mesh Generation
- 2D Mesh Polygon Properties dialog

External Links

- Gonzales, Darren S. (2000). An Automatic Finite Element Mesh Generation Method: the Adaptive Rectangular Coons Patch. Thesis, Brigham Young University. TA 4.02 .G6476 2000

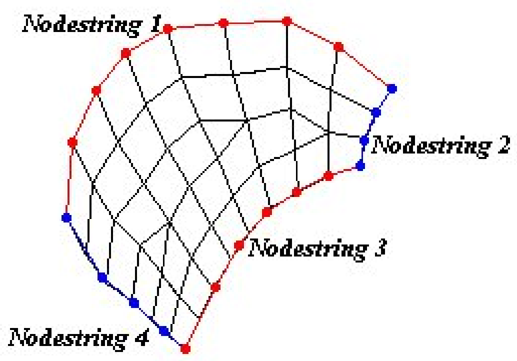
Patches

Patching is a mesh generation technique used to fill the interior of a polygon. A polygon is assigned to be a patch in the *Polygon Attributes* dialog and is filled with the **Map → 2D Mesh** command.

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the polygon as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. Patches are applicable when the data points are gathered along parallel lines, such as cross sections in a river.

Rectangular Patch

Elements can be made to fill a rectangular area by choosing the **Rectangular Patch** command from the *Elements* menu in the Mesh Module. To define a rectangular patch, four nodestrings must be selected. The nodestrings must connect at the ends.



Rectangular Patch

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the nodestrings as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. Patches are applicable when the data points are gathered along parallel lines, such as cross sections in a river. The following options are available for each edge of the rectangular patch:

- **Use original nodes** – This option causes the original nodes from the nodestring to be used as corner nodes of elements along the boundary.
- **Distribute nodes** – This option distributes the specified number of nodes as corner nodes of elements along the boundary. If elements already exist on the boundary, then this option is unavailable.
- **Bias** – This is used with the *Distribute nodes* option. It causes the spacing of nodes along the nodestring to be weighted more to one of the corners.

After the spacing on each side is defined, click the **Preview** button to see how the patch will look. If changes are desired, they can be made. When the patch looks good, click the **OK** button to accept it. The patch can be canceled by clicking the **Cancel** button. Be careful to use the preview button because **THERE IS NO UNDO FOR THIS OPERATION**.

The elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

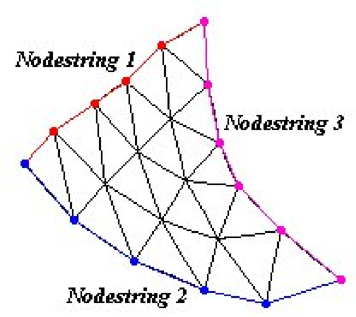
Rectagualar Patch Hints

The following are some hints when using rectangular patches:

- The curvature of the patch can change somewhat, but it should not switch directions. If it does, then the patch should be split at the inflection point of the curve.
- Although opposite sides in the rectangular patch are not required to have the same number of nodes, the best patches occur when this is close. In the example shown above, the two ends have the same number of nodes and the two sides only differ by three nodes.

Triangular Patch

Elements can be made to fill a triangular area by choosing the **Triangular Patch** command from the *Elements* Menu. To define a triangular patch, three nodestrings must be selected. The nodestrings must connect at the ends.



Triangular Patch

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the nodestrings as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. The following options are available for each edge of the triangular patch:

- *Use original nodes* – This option causes the original nodes from the nodestring to be used as corner nodes of elements along the boundary.
- *Distribute nodes* – This option distributes the specified number of nodes as corner nodes of elements along the boundary. If elements already exist on the boundary, then this option is unavailable.
- *Bias* – This is used with the *Distribute nodes* option. It causes the spacing of nodes along the nodestring to be weighted more to one of the corners.

All three sides of a triangular patch must have the same number of nodes. After the spacing on each side is defined, click the *Preview* button to see how the patch will look. If changes are desired, they can be made. When the patch looks good, click the **OK** button to accept it. The patch can be canceled by clicking the **CANCEL** button. Be careful to use the preview button because **THERE IS NO UNDO FOR THIS OPERATION**.

The elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

Triangular Patch Hints

All three sides of a triangular patch must have the same number of nodes.

Errors

When the patch is previewed in the *Polygon Attributes* dialog, the elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

If a polygon cannot be patched, a help string under the preview window in the *Polygon Attributes* dialog explains what needs to be changed.

Related Topics

- Mesh Generation
- 2D Mesh Elements Menu
- Adaptive Tesselation
- Adaptive Front Triangulation

Paving

The paving method uses an advancing front technique to fill the polygon with elements. Based on the vertex distribution on the boundaries, equilateral triangles are created on the interior to define a smaller interior polygon. Overlapping regions are removed and the process is repeated until the region is filled. Interior nodal locations are relaxed to create better quality elements.

Both the Paving method and the Scalar Paving Density can be selected in the *2D Mesh Polygon Properties* dialog. The dialog is accessed by right-clicking on the a polygon and selecting the **Attributes** command.

Scalar Paving Density

Scalar paving density utilizes the same approach as paving with the added component of a size dataset. A size dataset defines the desired spacing of nodes in a spatial fashion. A scattered dataset provides the geometric basis for the size dataset, and a dataset on the scatter set provides the values for the size dataset. SMS redistributes the vertices on the boundaries of the polygon to match the underlying size dataset.

Related Topics

- Patch Meshing Algorithm
- Mesh Generation

Adaptive Tesselation

Adaptive tessellation is a mesh generation technique used to fill the interior of a polygon. The method is based on overlaying a quad tree on the polygon, and recursively splitting the quads until the size approaches the desired spacing. SMS derives the desired spacing based on either the spacing of the original polygon, or based on a user specified spatially varying scalar dataset (for a scattered dataset). A polygon is assigned to be adaptive tessellation in the polygon attributes dialog and is filled with the *Map to 2D Mesh* command.

The adaptive tessellation technique is robust and relatively quick, however, it often results in discrete increments in resolution as the overlying quad tree grid transitions from one resolution level to another. For this reason, the Advancing Front Triangulation method is preferred.

Boundary Spaced Adaptive Tessellation

Adaptive tessellation uses the existing spacing on the polygons to determine the element sizes on the interior. Any interior arcs and refine points are forced into the new mesh. If the input polygon has varying node densities along its perimeter, SMS attempts to create a smooth element size transition between these areas of differing densities. By altering the size bias, the user can indicate whether SMS should favor the creation of large or small elements. Decreasing the bias will result in smaller elements; increasing the bias will result in larger elements. In either case, the elements in the interior of the mesh will honor the arc edges and the element sizes specified at nodes. The bias simply controls the element sizes in the transition region.

Scalar Adaptive Tessellation

SMS supports the option to control the local target size of elements using a Size Dataset. This is the scalar adaptive tessellation method and requires the user to select a spatial dataset that is everywhere positive to define the local spacing of the desired mesh. This may come from a variety of sources. See the tutorials on mesh generation for CGWAVE and ADCirc for more information.

Related Topics

- Mesh Generation
- Advancing Front Triangulation
- Patches

Size Function

A size function is a multiple that guides the size of elements to be created in SMS.

A size function determines the element size based off of a dataset that will be created by SMS. Each point is assigned a size value. This size value is the approximate size of the elements to be created in the region where the point is located. The mesh will be denser where the size values are smaller. Size functions can be based off of different criteria. For example, they may be based on either depth, slope, or curvature of the model.

Size Function Based on Depth

Many coastal models utilize a size function based on depth. As the depth gets shallower, the elements should get smaller. The model will become finer near areas of interest and coarser at deep water areas that are less significant.

Size Function Based on Slope

Size functions based on slope are helpful when analyzing slope data because as the rate of change of the gradient increases, the smaller the mesh element becomes. Size functions based on slope are mostly applied to riverine models.

Related Topics

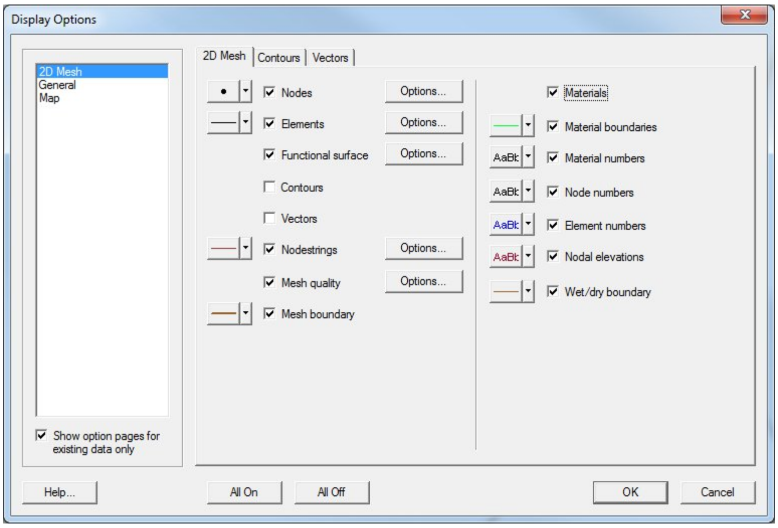
- Mesh Generation

3.8.b. Interface Components

3.8.b.1. Mesh Module Display Options

Mesh Module Display Options

The properties of the mesh data SMS displays on the screen can be controlled through the *Display Options* dialog. The entities associated with the mesh module with display options are shown below. Some of these entities also have an associated Options button. For these entities, additional display options are available. The available mesh display options include the following:



Display Options dialog showing *2D Mesh* options

- **Nodes –** A circle is filled around each node. The user can specify the radius and color of these circles. The **Options** button is used to set the display of nodal boundary condition data. The dialog that opens when this button is clicked depends on the current numerical model.
- **Nodal BC –** Some models support nodal boundary conditions. For those that do, each type of boundary condition can be displayed by highlighting the node with a symbol associated with that boundary condition. The **Options** button next to the *Nodal BC* entry of the display options allows the user to select/modify the symbols associated with each boundary condition.
- **Elements –** Element edges.
- **Functional Surface**
- **Contours –** The mesh contours are drawn for the active scalar dataset. Use the contours tab to change contour options.
- **Vectors –** The mesh vectors are drawn for the active vector dataset. Use the vectors tab to change vector options.
- **Nodestrings –** The color in which a nodestring is drawn depends upon its type. Unassigned nodestrings are drawn in the color/thickness/style shown at the left of the toggle box. For the display of boundary condition nodestrings, click the Options button. The dialog that opens when this button is clicked depends on the current numerical model.
- **Mesh Quality –** The mesh quality shows potential problems with the finite element mesh layout. An element is highlighted in a color corresponding to the criterion which it violates. The **Options** button opens the *Mesh Quality Options* dialog to specify display options for the mesh quality criteria.
- **Mesh Boundary –** A line is drawn around the perimeter of the mesh.
- **Inactive Mesh –** Determines the color of all inactive meshes when using multiple meshes.
- **Materials –** Elements can be filled with the color and pattern which define their materials. Materials and their display properties can be edited by choosing *Edit*|**Materials** from the menu.
- **Material Boundary –** The boundary between zones of elements with a common material type is drawn using specified line attributes.

- **Material Numbers –** The material id number can be displayed in the center of each element. User selects font and color.
- **Node Numbers –** The node id number can be displayed next to each node. User selects font and color.
- **Element Numbers –** The id of each element.
- **Nodal Elevations –** Displays the z elevation at each node.
- **Wet/Dry Boundary –** After a simulation has been opened, the interface between wet and dry nodes can be displayed.

Model specific options

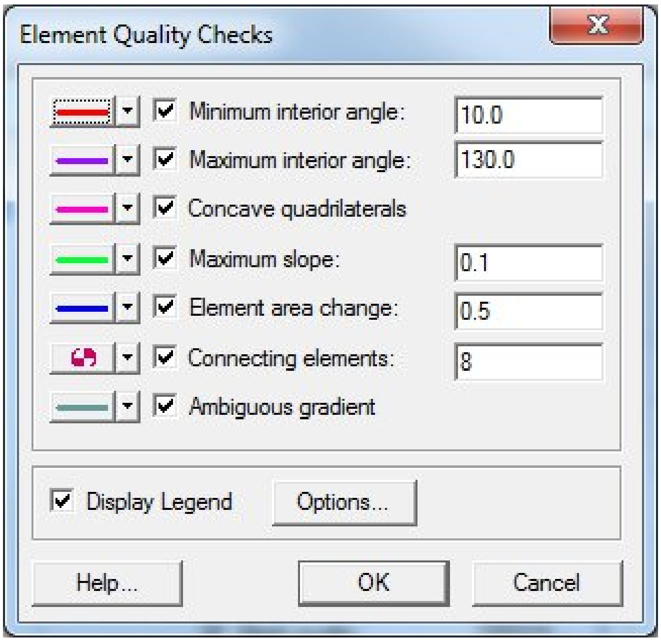
Each model may include model specific display options. These appear at the bottom of the *display options* dialog and include such things as 1D contour options for RMA2 and tidal ellipses for ADCIRC.

Related Topics

- Mesh Module
- Display Options

Mesh Quality

Several rules of mesh element construction, if adhered to, will help in creation of a well-behaved finite element network. Violations of the following mesh quality checks should be avoided. Violations of these mesh quality checks can be displayed in SMS (see Mesh Display Options):



Mesh Module *Element Quality Checks* dialog

- **Minimum / Maximum interior angle –** For triangular elements, if the angle is between 10 and 150 degrees, computation problems will usually be avoided. Care must also be taken when curved edges are defined (non-linear midside nodes) to prevent overlap of element sides.
- **Concave quadrilaterals –** For quadrilateral elements, if the angle is between 30 and 150 degrees, computation problems will usually be avoided. Care must also be taken when curved edges are defined (non-linear midside nodes) to prevent overlap of element sides.
- **Maximum slope –** Rapid changes in slope can cause computational instabilities.
- **Element area change –** Nodes need to be more plentiful and elements smaller in areas where the solution variables (u,v, and h) change rapidly. Such areas may be located near channel or floodplain constrictions, in channel bends, or at sudden changes in bed slope. The network should be dense in the critical areas of interest. The density of a network can vary through the solution domain. Areas that are of little interest and have stable flow characteristics should not be as dense as critical areas. The size of elements needs to change gradually when moving from an area described by small elements to an area modeled with large elements, or vice versa. A rule of

thumb is to keep the areas of neighboring elements within a factor of two, meaning an element is twice as big or half as big as its adjacent elements.

- **Connecting elements** – Avoid creating "pinwheels" by limiting the number of elements connecting at a node to fewer than eight.
- **Ambiguous gradient** – All triangular elements are planar by their definition. However, quadrilateral elements may vary significantly from a plane. It is a good idea to construct elements as close to a plane as possible. This precludes the existence of elements whose slope, or direction of drainage is ambiguous.
- **Display Legend** – checking this box will display the legend in the *Graphics Window*.
  - **Options...** – this button will bring up the *Legend Options* dialog.

Related Topics

- Mesh Display Options
- Mesh Module
- Display Options

3.8.b.2. 2D Mesh Module Tools

2D Mesh Module Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Mesh Module is active. Tools specific to a model interface are described with the corresponding model. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the 2D Grid tool palette.

Tool

Tool Name **Create Mesh NodesSelect Mesh NodesCreate NodestringsSelect NodestringsCreate ElementsSelect ElementsSwap EdgesMerge/Split ElementsLabel Contours** Description The **Create Mesh Nodes** tool is used to manually create a node using the mouse. A node will be created at the location where the mouse button is clicked inside the *Graphics Window*. If the node is created inside the triangulated area of a current mesh, and the Insert nodes into triangulated mesh option is turned on, then the new node will be added as part of the mesh. If the new node is not added as part of the current mesh, then the z-value assigned depends on the Nodal z-value option.

The **Select Mesh Nodes** tool is used to select nodes. A single node is selected by clicking on it. A second node can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple nodes can be selected at once by dragging a box around them. A selected node can be deselected by holding the *SHIFT* key as it is clicked.

If the nodes are not locked (see the menu command Nodes | Locked), then a single node can be clicked and dragged to a new location. As the node is being dragged, its new location is shown in the *Edit Window*. If a single node selected, the *X, Y, and Z Coordinate* fields in the *Edit Window* become available to set the node location exactly. If multiple nodes are selected, the *Z Coordinate* field in the *Edit Window* becomes available. The value shown is the average elevation value of all selected nodes. If this value is changed, the new value will be assigned to all selected nodes.

With one node selected, the *Edit Window* shows the node id number and the number of elements to which it is attached. With two nodes selected, the *Edit Window* shows both node id numbers and the distance between the nodes. With multiple nodes selected, the *Edit Window* shows the number of selected nodes.

The *Create Nodestrings* tool is used to create node string. Nodestrings are used for operations such as assigning boundary conditions, forcing breaklines into the mesh, and renumbering the mesh. To create a nodestring:

1. Click on a node. The node will be highlighted in red and a prompt will be shown in the *Help Window*.
2. Click on any node to add it to the nodestring. The selected node is also highlighted in red and a solid red line is drawn between the two nodes. Continue adding nodes to the nodestring in this manner.

   1. Note: For most operations, nodes in the nodestring should be adjacent, but this is not required. A breakline, for example, will usually be made of nodes which are not adjacent.
   2. Press the *BACKSPACE* key to backup one node. Press the *ESC* key to abort the nodestring creation.
   3. Double-click a node or press the *ENTER* key to end the nodestring creation.
3. The *SHIFT* and *CTRL* keys assist in creating large nodestrings which are made up of adjacent nodes. These can be used after at least one node has been selected and function as follows:
   1. *SHIFT*. Holding down the *SHIFT* key and selecting another node will add to the nodestring all nodes between the two. The path chosen is the shortest distance between the two nodes. This is useful for creating continuity strings which run along a cross section of the mesh.
   2. *CTRL*. Holding down the *CTRL* key and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going counter clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.
   3. *CTRL + SHIFT*. Holding down both the *CTRL* and *SHIFT* keys and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.

The **Select Nodestrings** tool is used to select nodestrings. When this tool is chosen, a small icon appears near the center of each nodestring. A nodestring is selected by clicking inside this icon. A second nodestring can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple nodestrings can be selected by dragging a box around their icons. A selected nodestring can be deselected by holding the *SHIFT* key as its icon is clicked.

When nodestrings are selected, the Z Coordinate field in the *Edit Window* becomes available. The value shown is the average elevation value of all nodes in the selected nodestrings. If this value is changed, the new value will be assigned to all nodes in the selected nodestrings.

With one nodestring selected, the *Edit Window* shows the number of nodes in the nodestring, its type, and its length. With multiple nodestrings selected, the *Edit Window* shows the number of selected nodestrings and their total length.

Most elements in SMS will be created using automatic mesh generation techniques. At times, however, it is necessary to manually create a single element or a small group of elements.

Although SMS supports various types of elements, only those element types supported by the current numerical model will be available in the tool palette. Some of these element types are linear while others are quadratic. A linear element has only corner nodes, while a quadratic element has midside nodes between the corner nodes.

The following linear elements are supported:

2-node lines 3-node triangles 4-node quadrilaterals.

The following quadratic elements are supported:

3-node lines 6-node triangles 8-node quadrilaterals 9-node quadrilaterals.

Linear and quadratic elements cannot coexist in a single mesh. If linear elements exist in a mesh, then the quadratic element creation tools are dimmed out. Similarly, if quadratic elements exist in a mesh, then the linear element creation tools are dimmed out. To create a single linear or quadratic element:

1. Select the tool which corresponds with the type of element to be created.
2. Click on the corner nodes which will make the element, one-by-one. Do not click midside nodes. As each node is clicked, it becomes highlighted in red.
3. Alternatively, a box can be dragged around the corner nodes which will make the element. A beep will sound if the box does not surround the exact number of corner nodes required by the selected tool.

The midside nodes in quadratic elements are created automatically, as is the center node of a nine-node quadrilateral. Before a new element is created, SMS performs the following quality checks:

- The new element cannot overlap other elements.

- A quadrilateral element cannot twist or overlap itself.
- A quadrilateral element cannot be concave.

If any of the these fails, the new element is not created.

The **Select Elements** tool is used to select elements. A single element is selected by clicking inside it. A second element can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple elements can be selected at once by dragging a box around them. Holding the *CTRL* key and dragging the mouse selects any elements through which the line is drawn. A selected element can be deselected by holding the *SHIFT* key as it is clicked.

When elements are selected, the*Z Coordinate* field in the *Edit Window* becomes available. The value shown is the average elevation value of all nodes in the selected elements. If this value is changed, the new value will be assigned to all nodes attached to the selected elements. Caution must be used when changing node elevations in this manner. Do not create large flat areas where surrounding elements may become dry because this can cause ponds to form when the finite element analysis is performed.

With one element selected, the *Edit Window* shows the element id number, its type, and its area. With multiple elements selected, the *Edit Window* shows the number of selected elements and their combined area.

The **Swap Edges** tool is used to manually swap the edges of two adjacent triangles. This is useful in such cases as preserving a geometrical feature in the mesh or avoiding a artificial dam in a channel.

Two adjacent triangles form a quadrilateral with an element edge down one diagonal. When the diagonal is clicked, it gets swapped to the other diagonal, as long as the quadrilateral is not concave.

The **Merge/Split Elements** tool is used to either merge two triangles into a quadrilateral or split a quadrilateral into two triangles. This is a useful tool to use when trying to avoid certain mesh drying problems.

To split a quadrilateral element, click inside it. An element edge appears on the diagonal which will make the triangles uphold the Delaney criteria. To merge two adjacent triangles into a quadrilateral, click the common edge. A quadrilateral will form as long as it is not concave.

The **Label Contours** tool is used to manually create a contour label using the mouse. To add a label, click on the point where the label should be created. The label will remain on the screen until either it is manually removed or the automatic contour label options are changed. To manually remove a contour label, hold the *SHIFT* key and clicking on it. There are also available automatic contour label options. Right Click Menu N/A See the 2D Mesh Nodes Menu and 2D Mesh Elements Menu articles. N/A See the 2D Mesh Nodestrings Menu article. N/A See the 2D Mesh Elements Menu article. N/A N/A N/A

Related Topics

- Editing 2D Meshes
- Mesh Module

Editing 2D Meshes

2D Mesh nodes can be inserted, deleted, or moved.

2D Mesh elements can be edited in the following ways:

- Elements can be converted between linear and quadratic.
- The type of element can be changed from a 3 node element to a 4 node element by merging triangles.
- A 4 node element can be converted to a 3 node element by splitting the 4 node element.
- Elements can be refined automatically.
- The material assigned to an element can be changed.
- Poorly shaped boundary triangles can be automatically selected for deletion.
- Breaklines can be inserted into the mesh

Deleting Nodes

A set of selected nodes can be deleted by hitting the *DELETE* key or selecting the **Delete** command from the *Edit* menu. Elements attached to the nodes are also deleted.

If the Retriangulate voids when deleting option is turned on, the void created when a node and the elements surrounding the node are deleted is re-triangulated or filled in with triangles. This feature makes it possible to selectively "unrefine" a region of the mesh or reduce the density of the nodes in a region of the mesh without having to completely recreate all of the elements in the region.

If the Retriangulate voids when deleting item in the *Node Options* dialog is not set, the selected node and the elements surrounding the node are simply deleted and the resulting void is not filled in with triangles.

If the *Confirm Deletions* option in the *Edit* menu is active, SMS will prompt the user to confirm each deletion. This feature is helpful in preventing accidental deletions. The *Confirm Deletions* item is toggled by selecting it from the menu.

Editing Node Coordinates

The coordinates of a 2D Mesh node can be edited by selecting the mesh node and entering the new coordinates in the edit boxes in the *Edit Window*. It is also possible to drag an existing node to a new location by clicking on the node and moving the mouse with the button held down until the node is in the desired position.

If the snap to grid option in the *Drawing Grid Options* dialog is set, the node will move in increments corresponding to the drawing grid. If the node being dragged is connected to one or more elements, SMS will not allow the node to be dragged to a position where one of the surrounding elements would become ill-formed.

Since it is possible to accidentally drag points, nodes can be "locked" to prevent them from being dragged by selecting the **Lock All Nodes** item from the *Nodes* menu. The nodes can be unlocked by unselecting **Lock All Nodes** from the *Nodes* menu.

Merging Triangles

The triangulate operation creates a mesh composed entirely of triangles. In some cases it is desirable to have the mesh composed primarily of quadrilateral elements. Quadrilateral elements result in less elements which leads to faster solutions, and quadrilateral elements are often more stable numerically. To address this need, two options are provided for converting triangular elements to quadrilateral elements:

The Merge Triangles Command

The **Merge Triangles** command in the *Elements* menu can be used to automatically merge pairs of adjacent triangular elements into quadrilateral elements. Upon selecting the **Merge Triangles** command, the user is prompted to input a minimum interior angle. This angle should be between 0 and 90 degrees. If no elements are selected, all of the triangular elements in the mesh are then processed. If some elements have been selected, only the selected elements are processed.

The conversion process works as follows:

1. The set of elements to be processed is traversed one element at a time. Each triangular element that is found is compared with each of its three adjacent elements. If the adjacent element is a triangle, the trapezoid formed by the triangle and the adjacent triangle is checked.
2. Each of the four interior angles of the trapezoid is computed and compared to a minimum interior angle. If all of the angles are greater than the user-specified minimum interior angle, then the two triangles are merged into a single quadrilateral element.

This process is repeated for all of the elements. The merging scheme will not always result in a mesh composed entirely of quadrilateral elements. Some triangular elements are often necessary in highly irregular meshes to provide transitions from one region to the next.

The Merge/Split Tool

The other option for merging triangles involves the use of the **Merge/Split** tool in the *Tool Palette*. This tool can be used to manually merge triangles one pair at a time rather than using the automatic scheme described above.

The manual method is also useful to edit or override the results of the automatic merging scheme in selected areas. The Merge/Split tool can also be used to undo a merge. A quadrilateral element can be split into two triangles by clicking anywhere in the interior of the element. This tool is useful if a pair of triangles is inadvertently merged.

Splitting Quadrilaterals

Occasionally it is necessary to split quadrilateral elements into triangular elements. For example, in order for new nodes to be automatically inserted into a mesh, the elements in the region where the node is inserted must be triangular. Also, in order to process a breakline, the elements in the region of the breakline must be triangular. In such situations, it may be necessary to split a group of quadrilateral elements into triangular elements. Two options are provided for splitting quadrilateral elements:

The Split Quads Command

The **Split Quads** command in the *Mesh* menu can be used to split a group of quadrilateral elements into triangular elements. If no elements are selected, all of the quadrilateral elements in the mesh are split. If some elements have been selected, only the selected quadrilateral elements are split.

The Merge/Split Tool

The other option for splitting quadrilateral elements involves the use of the **Merge/Split** tool in the *Tool Palette*. If the **Merge/Split** tool is selected, clicking anywhere in the interior of a quadrilateral element with the mouse cursor will cause the element to be split into two triangles. The shortest diagonal through the quadrilateral is chosen as the common edge of the two new triangular elements.

Mesh Subset Edit Mode

When working with large meshes, it can be useful to work on only a small portion of the mesh. The user can select elements on which to work on and enter subset edit mode either through the select element tool's right-click menu or the right-click menu on the mesh tree item. The geometry of the mesh inside the selected area can then be changed. Datasets other than the depth/elevation dataset are deleted upon entering mesh subset edit.

Prohibited Actions While in Subset Edit Mode

- Making changes on the border of the mesh subset
- Making changes outside of the mesh subset
- Reading in meshes from file
- Creating meshes from feature map or scatter data
- Running models
- Changing attributes such as boundary conditions on nodes, nodestrings, and elements
- Renumbering node/nodestring/element ids

Commit Mesh

The user can merge the changes made to the subset of the mesh with the rest of the mesh by selecting **Commit mesh** from the right-click menu of either the mesh tree item or the **select element** tool. **Commit mesh** will exit subset edit mode.

Revert Mesh

The user can revert to how the mesh was upon entering mesh subset edit mode by selecting **Revert mesh** from the right-click menu of either the mesh tree item or the **select element** tool. **Revert mesh** will exit subset edit mode.

Related Topics

- 2D Mesh Module Tools

2D Mesh Module Tools Right-Click Menus

The following tools are contained in the Dynamic Tools portion of the tool palette when the Mesh Module is active. These are tools with corresponding right-click menus.

Select Mesh Nodes

Right-clicking on a mesh node while using the **Select Mesh Nodes** will bring up the following menu options:

- **Delete** – Delete selected mesh Nodes.
- **Assign BC** – Assign a boundary condition to selected mesh nodes.
- **Transform** – Transform the selected nodes either by scaling, translation, or rotation.
- **Triangulate** – Triangulate the selected nodes to form triangle elements.

Select Nodestrings

Right-clicking on a selected nodestring while using the **Select Nodestring** tool will bring up the following options:

- **Delete Selected** – Delete selected nodestrings.
- **Assign BC** – Assign a boundary condition to selected nodestrings.
- **Reverse Direction** – Reverse direction of selected nodestrings.
- **Force Breaklines** – Force element edges to follow the selected nodestrings.
- **Renumber Nodes** – Renumber the nodes starting with an ID of 1.
- **Renumber Nodestrings** – renumber the nodestrings starting with an ID of 1.
- **Smooth** – Smooth the mesh boundary along the path of a selected nodestring by moving midside nodes. Only used for quadratic elements.

Select Elements

Right-clicking on a selected element while using the **Select Elements** tool will bring up the following options:

- **Delete** – Delete selected element.
- **Refine** – Split all selected elements into four elements.
- **Relax** – Moves all the nodes on the interior of the selected elements to the centroid of their contributing area. Iterates based on option set in the options command.
- **Assign Material Type** – Assign material type to selected elements.

Related Topics

- Editing 2D Meshes
- Mesh Module

3.8.b.3. 2D Mesh Module Menus

2D Mesh Module Menus

The following menus are available in the 2D Mesh Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Data
- Nodes
- Nodestrings
- Elements

Right-Click Menu

Right-clicking on the Mesh Data folder in the *Project Explorer* will bring up the *Display Options* Dialog. Right-clicking on a Mesh Item in the *Project Explorer* will bring up the following menu options:

- **New Folder –** Creates a new folder under the *Mesh* item
- **Delete –** Deletes the active mesh.
- **Duplicate –** this command will create a duplicate of the active mesh, along with its model data. The name of the new mesh will be the same as the original mesh, but with an appended number.
- **Rename –** Allows the user to rename the active mesh. Names need to be unique, hence if the user enters a non-unique name, a warning message will appear and the name will be reverted.
- **Merge 2D Meshes –** merges two meshes together. See the article Merge 2D Meshes.
- **Convert –** Converts mesh to Map or 2D Scatter.
- **Projection –** Allows the user to set the projection of the mesh.
- **Reproject –** Allows the user to reproject the projection of the mesh.
- **Metadata –** Allows the user to make annotations.
- **Create Quality Mesh Scatter Set –** This command creates a scatter set consisting of one vertex at the center of each element in the mesh. Six datasets are created for this scatter set including the six quality measures defined in the ARR quality plot. These datasets range from 0.0 to 1.0. The higher the value, the higher the quality of the element. (If SMS supported element centered datasets, these quantities could be displayed directly on the mesh.) The creation of this scatter set gives the user a spatial feeling for quality of the mesh.
- **Zoom to Mesh –** Zooms to where the mesh is located within the *Graphic Window*.

Model Specific Menus

- ADCIRC
- ADH
- CGWAVE
- FESWMS
- Generic Model
- TABS
  - RMA2
  - RMA4

2D Mesh Nodestrings Menu

The *Nodestrings* Menu includes the following commands:

General Commands

- **Options**

  Breakline options
- **Force Breaklines**

  Force element edges to follow the selected nodestring
- **Smooth**

  Smooth the mesh boundary along the path of a selected nodestring by moving midside nodes. Only used for quadratic elements.
- **Renumber Nodestrings**

  Renumbers the nodestrings starting with an id of 1.
- **Merge**

  Merge selected nodestrings to form a single nodestring. Available if more than one nodestring is selected.
- **Split**

  Split a single nodestring into multiple nodestrings at the selected mesh node. Available if a mesh node is selected.
- **Reverse Direction**

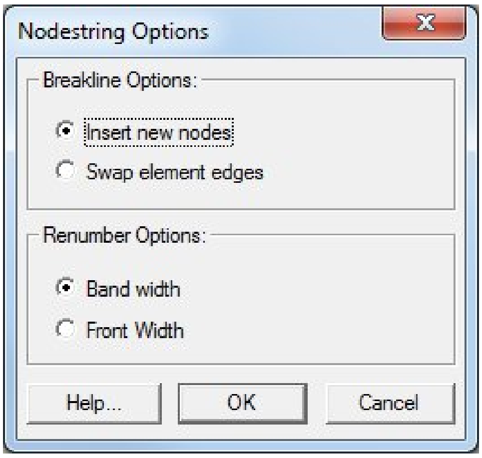
  Reverse the direction of selected nodestrings. Selecting a nodestring causes the direction arrows to be displayed and can be used to verify the nodestring direction.

Nodestring Options

Breakline Options

Controls how breaklines are processed.

- *Insert new nodes* – Triangles intersected by the breakline are modified by adding new nodes at necessary locations to ensure that the edges of the triangles will conform to the breakline. The elevations of the new nodes are based on a linear interpolation of the breakline segments. The locations of the new nodes are determined in such a way that the Delauney criterion is satisfied.
- *Swap element edges* – Triangles intersected by the breakline are modified by swapping element edges to ensure that the edges of the triangles will conform to the breakline.



Mesh *Nodestring Options* dialog

Renumber Options

- *Band Width* – See Front Width and Band Width for more information.
- *Front Width* – See Front Width and Band Width for more information.

Related Topics

- Mesh Module

Mesh Data Menu

Most of the SMS modules have a Data menu, but the items in this menu are different for each module. The Mesh Module commands include:

Command	Description
Steering Module	Model command – launches the *steering* dialog to connect multiple model runs
Switch Current Model	Model command – allows the user to change the current numerical model associated with the mesh
Data Calculator	Dataset command – invokes the *data calculator* for the mesh
Create Datasets	Dataset command – creates specific datasets for the mesh based on user specified options
Map Elevation	Dataset command – assigns a new "elevation" or "depth" dataset to the current mesh. See below.
Zonal Classification	Dataset command – creates polygons matching user specified criterion from the current mesh and its datasets
Contour Options	Visualization command – invoke the *contour options* dialog (display options)
Vector Options	Visualization command – invoke the *vector options* dialog (display options)
Film loop	Visualization command – launches the film loop generation wizard
Mesh → Scatterpoint	Data conversion command – converts the current mesh to a scatter point set. Also available as a right-click command. See Below.
Mesh → Map	Data conversion command – converts the current mesh to a map module coverage. Also available as a right-click command. See Below.

Map Elevation

SMS requires that a mesh or grid have bathymetry, or bottom elevation data, associated with the nodes or cells. By default, SMS creates a dataset named "elevation" to store the elevation values. The dataset being used to store elevations is referred to as the mapped dataset.

The *Data* menus in several modules include a command to use another functional dataset as this mapped elevation function. When this command is performed the *Select Dataset* dialog opens to allow any existing scalar dataset to be chosen. Any time step of any scalar dataset can be used as the mapped dataset and override the previous nodal elevation values. This is used mainly when interpolating new elevation data from scatter points.

Mesh to Map

The **Mesh → Map** command in the *data* menu (mesh module) is used to convert mesh data into feature data (map module). This can be useful for creating a conceptual model from an existing numeric model. The Mesh → Map command converts the mesh data and places it in the active coverage. If you want the new data in a new coverage, click on the **Create New Coverage** button to create a new coverage and make it active.

Material Regions → Polygons

This option converts the materials in the current mesh into polygons in the map module. If the coverage supports materials (area property or most mesh model coverages support this), the polygons will have attributes reflecting the correct material values.

Mesh Boundaries → Polygons

This option converts the current mesh boundaries into polygons in the map module.

Mesh Contours → Arcs

This option creates an arc at a specific contour value (based upon the active dataset). It is not necessary for the value to be associated with a displayed contour line as SMS will determine where the line would be if it existed.

Mesh Nodestrings → Arcs

This option creates an arc for each nodestring in the mesh.

Mesh to Scatterpoint

The **Mesh → Scatterpoint** command in the *Mesh Data* menu is used to create scatter point data from existing mesh nodes. When this command is performed, a prompt appears to request a name for the new scatter point set.

When the scatter point set is created, it contains one scatter point for each mesh node, including midside nodes and center nodes. Any mesh datasets that have been read into SMS are copied into the new scatter point set. The scatter point data can then be used for interpolation.

Related Topics

- Mesh Module

3.8.b.3.1. 2D Mesh Elements Menu

2D Mesh Elements Menu

The *Elements* Menu includes the following commands:

Command	Description
Options	General command – set up the default element options.
Select Thin Triangles	General command – selects all thin triangular element that meet the definition specified in the options command. During the process of triangulation, a mesh of triangular elements is created around existing nodes. This usually creates triangular elements outside the desired mesh boundary. Many of these exterior triangles are very skinny, and some are virtually invisible. The **Select Thin Triangles** command from the *Elements* menu finds and selects skinny triangular elements which are on the mesh boundary. Thin triangles interior to the mesh will not be selected when this command is performed, since deletion of interior triangles would result in gaps in the mesh. After the thin triangles have been selected, they can be removed by selecting the Delete macro .
Find	General command – finds the element with a specified ID or location. The **Find Element** command from the *Elements* Menu is used to locate an element either with a specific ID, or surrounding a specific location. When this command is executed the *Find Element* dialog opens. When the *Find by ID* option is selected, then the element with the specified ID is highlighted in red. If there is no element with the specified ID, then an error message is given. Conversely, when the Find by nearest (x,y) coordinate option is selected, the element which surrounds to the specified (x, y) location is highlighted in red. With either of these methods, if the current tool is the **Select Elements** tool, then the found element becomes selected in addition to being highlighted.
Assign Material Type	General command – requires a selected element. Sets the material type of the selected elements based on the option defined in the options command. Each element in the mesh is assigned a material type. The default material ID can be set in the *Element Options* dialog. A selected element is assigned a new material type by choosing the **Assign Material Type** command from the *Elements* menu. If the *Assign default material* option is selected in the *Element Options* dialog, then the default material is automatically assigned to the selected element. If the *Prompt for material when assigning* option is selected in the *Element Options* dialog, then the *Materials Data* dialog opens from which a material type can be chosen.
Merge Triangles	Conversion command – merges triangle pairs that meet the standard for rectangles defined in the options command. Can operate on selected elements.
Split Quadrilaterals	Conversion command – splits quadrilateral elements into two triangular elements. Can operate on selected elements.
QUAD8 ↔ QUAD9	Conversion command – converts all QUAD8 to QUAD9 elements and vice versa. Only applies to the FESWMS model. Can operate on selected elements.
Linear ↔ Quadratic	Conversion command – converts all elements from linear to quad or vice versa. Only applies to TABS and FESWMS models.
Triangulate	Generation command – triangulates the selected nodes to form triangle elements.
Optimize Triangulation	Generation command – swaps edges of triangular elements to meet the Dulanay criterion.
Refine	Generation command – splits all selected elements into four elements. At times, there is not enough definition in a finite element mesh. The **Refine** command from the *Elements* menu splits each of the selected elements into smaller elements. After the selected elements have been refined, SMS automatically creates transitions, from the refined area of higher density to the unrefined area of lower density, using triangular elements. Refine options are set in the *Element Options* Dialog.

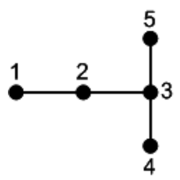
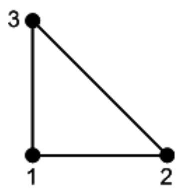
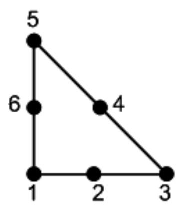
Relax	Generation command – moves all the nodes on the interior of the selected elements to the centroid of their contributing area. Iterates based on option set in the options command. The process of creating and editing a finite element mesh can result in poor quality elements. These elements may have poor interior angles or may violate the area change guideline for adjacent elements. The **Relax** command from the *Elements* menu can improve adjacent element areas and interior angles by moving nodes. This command moves nodes to improve the elements shape. Several options are available. Relaxation is an iterative process. The number of iterations performed and other options are specified in the *Element Options* dialog. If no elements are selected, then the relaxation is performed on all elements in the mesh.
Fix Bad Area Transitions	Generation command – removes nodes that cause bad area transitions as defined in the element quality control. The process of creating and editing a finite element mesh can result in poor quality elements. These elements may violate the area change guideline for adjacent elements, specifically, the area of the smaller of the adjacent elements divided by the area of the large element may be less than a recommended ratio. This ratio is set in the Mesh Quality entry of the 2D Mesh Display Options. The Fix Bad Area Transitions command from the Elements menu can improve adjacent element areas by removing nodes. The adjacent elements of each node in the mesh are examined for guideline violations. If more that one are found, a calculation is made to determine if the removal of the node and the retriangulation this would cause would maintain compliance for the newly formed elements and their neighbors. If this is the case the node is removed.
Rectangular Patch	Patch command – creates elements from four selected node strings. See the article Patches for more information.
Triangular Patch	Patch command – creates elements from three selected node strings. See the article Patches for more information.

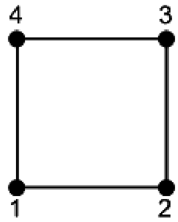
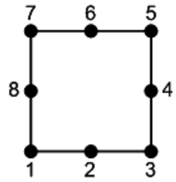
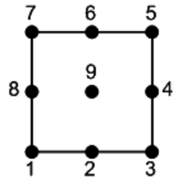
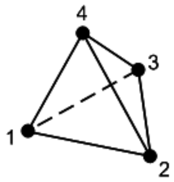
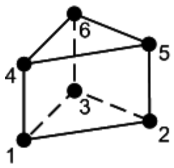
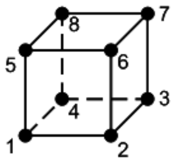
Related Topics

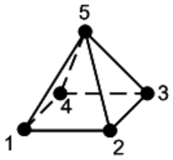
- Mesh Module
- Element Options Dialog

Element types

Element types used in XMS software. See also XMDF elements [1].

Element Type	Image	Faces
1D linear element with 2 nodes		
1D linear element with 3 nodes		
transition element		
2D linear triangle		
2D quadratic triangle		

<table>
<tr><td>2D linear quadrilateral</td><td>
</td><td></td></tr>
<tr><td>2D quadratic quadrilateral</td><td>
</td><td></td></tr>
<tr><td>2D quadratic quadrilateral with center node</td><td>
</td><td></td></tr>
<tr><td>3D linear tetrahedron</td><td>
</td><td><table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>2,3,4</td></tr>
<tr><td>2</td><td>1,4,3</td></tr>
<tr><td>3</td><td>1,2,4</td></tr>
<tr><td>4</td><td>1,3,2</td></tr>
</table></td></tr>
<tr><td>3D linear prism</td><td>
</td><td><table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>1,3,2</td></tr>
<tr><td>2</td><td>4,5,6</td></tr>
<tr><td>3</td><td>1,2,5,4</td></tr>
<tr><td>4</td><td>2,3,6,5</td></tr>
<tr><td>5</td><td>3,1,4,6</td></tr>
</table></td></tr>
<tr><td>3D linear hexahedron</td><td>
</td><td><table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>1,4,3,1</td></tr>
<tr><td>2</td><td>5,6,7,8</td></tr>
<tr><td>3</td><td>1,2,6,5</td></tr>
<tr><td>4</td><td>2,3,7,6</td></tr>
<tr><td>5</td><td>3,4,8,7</td></tr>
<tr><td>6</td><td>4,1,5,8</td></tr>
</table></td></tr>
</table>

<table>
<tr>
<td>3D linear pyramid</td>
<td>
</td>
<td>
<table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>1,4,3,2</td></tr>
<tr><td>2</td><td>1,2,5</td></tr>
<tr><td>3</td><td>2,3,5</td></tr>
<tr><td>4</td><td>3,4,5</td></tr>
<tr><td>5</td><td>4,1,5</td></tr>
</table>
</td>
</tr>
</table>

Back to XMS

Boundary Triangles

The perimeter of the mesh resulting from the triangulation process corresponds to the convex hull of the data points. This may result in some long thin triangles or "slivers" on the perimeter of the triangulated region.

Select Thin Triangles

There are several ways to select and delete long thin triangles.

Long thin triangles on the perimeter of the mesh can be automatically selected using the **Select Thin Triangles** item from the *Elements* menu. The triangles on the outer boundary are checked first and if the aspect ratio of a triangle is less than a critical value, the triangle is selected and the triangles adjacent to the triangle are then checked. The process continues inward until none of the adjacent triangles violate the minimum aspect ratio.

The "drag line" method for selecting elements was designed specifically for this purpose. Elements can be selected with a line by selecting the *Select Elements* tool, holding down the *CTRL* key, and dragging a line through all of the elements to be selected. The selected elements can then be deleted.

Related Links

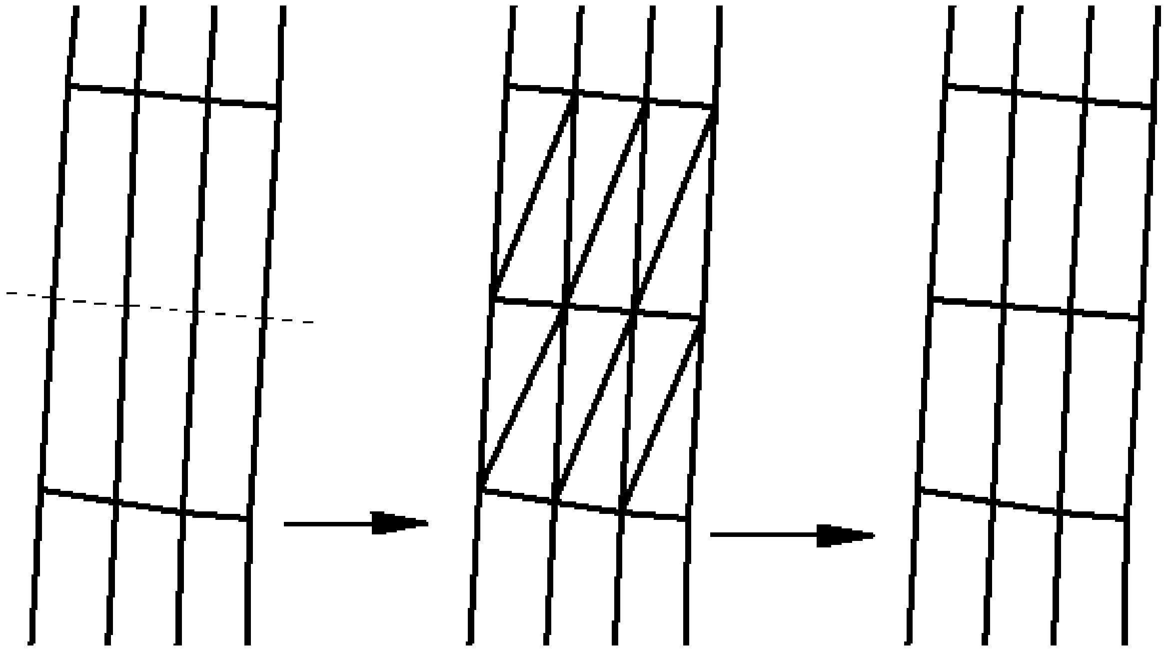
- Editing 2D Meshes
- Mesh Node Triangulation

Convert Elements

Several commands are available to modify the current elements these include:

Add Breakline

The **Add Breaklines** command from the *Elements* menu can be executed when at least one nodestring has been selected. This forces element edges along the nodestring line. When this command is performed, elements are sliced along the nodestring to ensure that the edges will conform to the breakline. The elevations of any new nodes are interpolated from the original mesh. All new triangles satisfy the Delauney criterion.



A breakline example is shown. This example has some long, skinny quadrilaterals which will be split across the width. The dotted line in the left part of the figure represents the location of the breakline. When the elements are split, triangles are formed. These can be merged together using the **Split/Merge** tool , as shown in the right part of the figure.

Merge Triangles

The **Split/Merge** tool can be used to merge individual pairs of triangles. Doing this manually for large numbers of elements takes a lot of time. The **Merge Triangles** command from the *Elements* menu automatically merges a selected set of triangles simultaneously. If no elements are selected when this command is executed, all triangles in the finite element mesh will be processed.

This command uses the *Merge triangles feature angle* specified in *Element Option* dialog. In order to form quadrilateral elements with the best aspect ratios, SMS starts with a feature angle of ninety degrees and checks for any elements that can be merged. Then, a series of steps are performed, each time lowering the feature angle and checking for elements that can be merged. This ensures that the quadrilaterals which are formed are as close to rectangular as possible. In general, after the automatic merging process is complete, a limited number of triangles will still exist.

Split Quadrilaterals

The **Split Quadrilaterals** command in the *Elements* menu is used to split a set of quadrilaterals into triangles. If no elements are selected, all quadrilateral elements in the mesh will be split. The quadrilaterals are split along the shortest diagonal.

Quad8 ↔ Quad9

The **Quad8↔Quad9** item from the *Elements* menu is used to convert between eight- and nine- noded quadrilaterals. FESWMS supports nine-noded quadrilaterals. Both FESWMS and TABS support eight-noded quadrilaterals. If no elements are selected when this command is performed, all elements are converted.

Linear ↔ Quadratic

Linear elements (three node triangles and four node quadrilaterals) can be converted to quadratic elements (six node triangles and eight node quadrilaterals) and vice versa by selecting the **Linear ↔ Quadratic** item from the *Elements* menu. A finite element mesh must be made of either all linear elements or all quadratic elements. Linear elements do not have midside nodes while quadratic elements do.

Refine

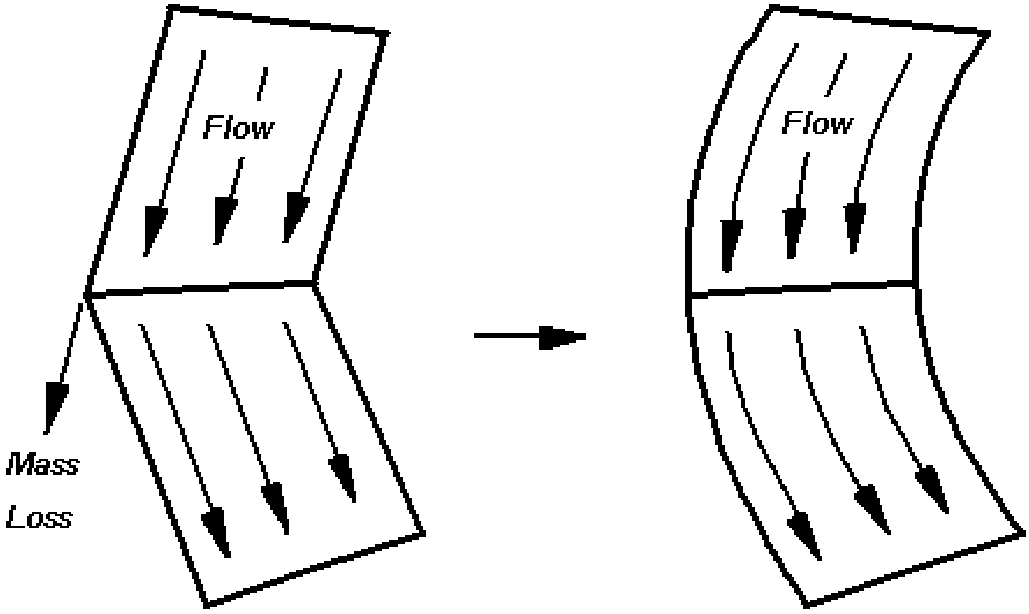
At times, there is not enough definition in a finite element mesh. The **Refine** command from the *Elements* menu splits each of the selected elements into smaller elements. After the selected elements have been refined, SMS automatically creates transitions, from the refined area of higher density to the unrefined area of lower density, using triangular elements. Refine options are set in the *Element Options* Dialog.

Relax

The process of creating and editing a finite element mesh can result in poor quality elements. These elements may have poor interior angles or may violate the area change guideline for adjacent elements. The **Relax** command from the *Elements* menu can improve adjacent element areas and interior angles by moving nodes. This command moves nodes to improve the elements shape. Several options are available. Relaxation is an iterative process. The number of iterations performed and other options are specified in the *Element Options* dialog. If no elements are selected, then the relaxation is performed on all elements in the mesh.

Smooth Nodestring

Quadratic elements have a node located at the midpoint of each edge. These nodes are generally referred to as midside nodes. The angular corners resulting from such elements are discontinuous. Such a discontinuity may result in inaccuracy in the numerical model sometimes referred to as a mass loss. Mass loss occurs because water artificially flows out of the mesh.



To minimize the abrupt change in flow direction, element edges can be curved by slightly moving the midside node. This can be done by hand using the **Select Nodes** tool with the nodes unlocked. Moving large numbers of nodes becomes tedious. However, element edges along a selected nodestring can be smoothed by SMS with the **Smooth Nodestring** command.

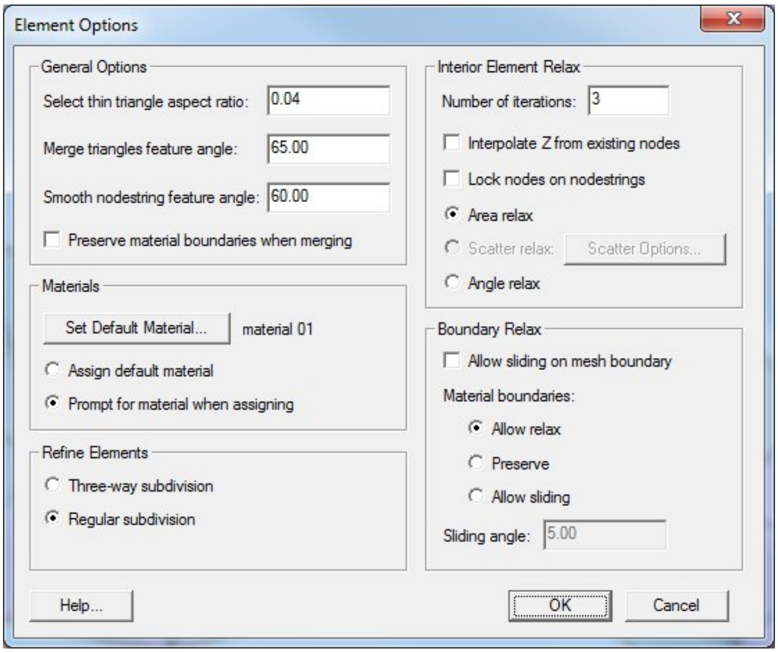
Normally, element edge smoothness is only a concern along the mesh boundary. However, if the analysis includes regions that become dry, interior boundaries should also be smoothed. To avoid smoothing corners that should be sharp, SMS provides a *Smooth nodestring feature angle* in the *Element Options* dialog. A corner will only be smoothed if it is less than the specified angle.

Related Topics

- 2D Mesh Elements Menu

Mesh Element Options

Certain parameters governing the creation and manipulation of nodes are set using the *Element Options* dialog, which is opened by selecting the **Options** command from the Elements menu. This dialog is divided into four sections.



Mesh *Elements Options* dialog

General Options

The *General Options* section of the *Element Options* dialog specifies the following parameters for general element operations:

Select thin triangle aspect ratio

When SMS finds thin elements, only elements with an aspect ratio (element width divided by element length) less than this value are selected. This value is also used by in the model checker mesh quality checks.

Merge triangle feature angle

This angle should be between zero and ninety degrees. Any two adjacent triangles are merged into a quadrilateral if all angles in the resulting quadrilateral are greater than the merge triangles feature angle.

Smooth nodestring feature angle

When a nodestring is smoothed, the smoothing will not be applied around a corner whose angle is greater than this value. See the convert elements article for a discussion on nodestring smoothing.

Preserve material boundaries

When turned on, triangles will not be merged into quadrilaterals if they are assigned different materials types, even if they satisfy the merge triangle feature angle criteria.

Materials

The *Materials* section of the *Element Options* dialog controls how materials are assigned to elements using the following options:

Set default material

This defines the default material assigned to elements as they are created.

Assign default material

When turned on, the material selected as the default material is assigned to selected elements when the assign material command is issued.

Prompt for material when assigning

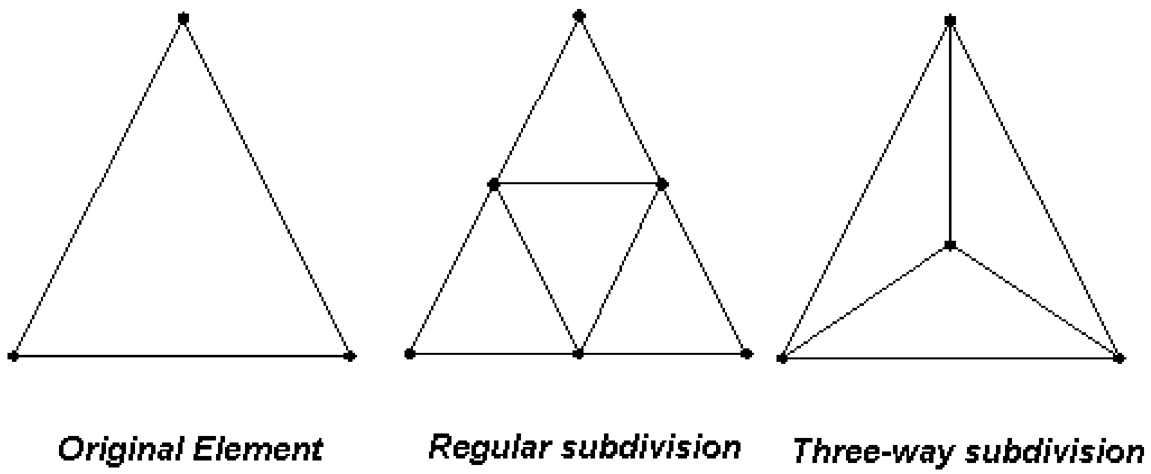
If this option is selected, the user must select from a list of existing materials to assign to the selected elements when the assign material command is issued.

Refine Elements

In some cases, a mesh does not have enough elements in a particular region of the mesh to ensure stability. Rather than inserting supplemental nodes and re-creating the mesh, it is possible to refine a selected region of the mesh using the **Refine Elements** command in the *Mesh* menu. This increases the mesh density of a selected area of the mesh. If no elements are selected, the entire mesh is refined. The elevations of the new nodes are interpolated from the existing nodes.

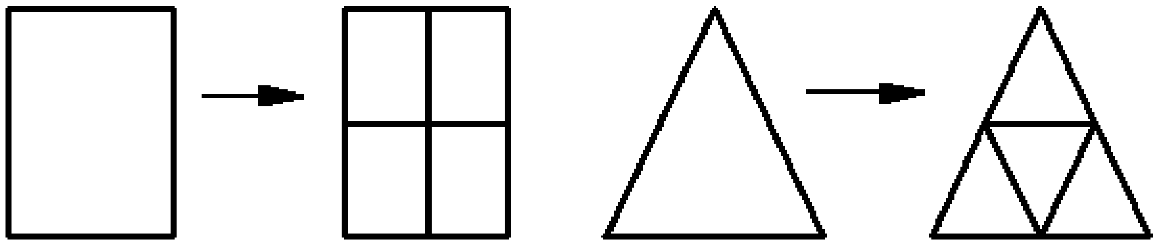
Three-way subdivision

Refines triangular element into three elements. Quadrilateral elements are not refined.



Regular subdivision

Refines triangular and quadrilateral elements into four elements.



Relax Elements

The Relax Elements section of the *Element Options* dialog controls the following relaxation parameters:

Number of iterations

This is the number of iterations to perform during the relaxation process.

Interpolate Z from existing mesh

When turned on, the nodal Z coordinate is interpolated from the old mesh so that the contours do not change. When this is turned off, the nodal Z coordinates are not changed when they are moved.

Lock nodes on nodestrings

Preserve material boundaries when relaxing. Previous versions of SMS would lock any nodes on a material or mesh boundary. Starting with version 7.0 of SMS, nodes on these boundaries will slide along the boundary unless it is part of a nodestring and this option is turned on.

Relaxation Method

- **Area relax** – equalize the area of elements adjacent to each node.
- **Scatter relax** – space the nodes according to the specified size dataset.
- **Angle relax** – equalize the angle of elements adjacent to each node.

Boundary Relax

Allow sliding on mesh boundary

When turned on, relaxation may modify the location of nodes on the mesh boundary.

Material boundaries

- **Allow relax** – Allows nodes located on material boundaries to move in all directions.
- **Preserve** – Prevents nodes located on material boundaries from moving.
- **Allow sliding** – Allows nodes located on material boundaries to move along the material boundary.
- **Sliding angle** – When a node is smoothed, the smoothing will not be applied around a material boundary whose angle is greater than this value.

Related Topics

- 2D Mesh Elements Menu
- Convert Elements

Relax Elements

The process of creating and editing a finite element mesh can result in poor quality elements. These elements may have poor interior angles or may violate the area change guideline for adjacent elements. The **Relax** command from the *Elements* menu can improve adjacent element areas and interior angles by moving nodes. This command moves nodes to improve the elements shape. Several options are available. Relaxation is an iterative process. The number of iterations performed and other options are specified in the *Element Options* dialog. If no elements are selected, then the relaxation is performed on all elements in the mesh.

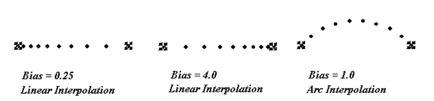
Related Topics

- Element Options Dialog
- 2D Mesh Elements Menu

3.8.b.3.2. 2D Mesh Nodes Menu

2D Mesh Nodes Menu

2D Mesh Nodes are the basic building blocks of elements in finite element meshes. Nodes are also required to create nodestrings and assign boundary conditions. The following commands are available when working with 2D Mesh Nodes:

Command	Description
Interpolation Options	The **Interpolation Options** item from the *Nodes* menu opens the *Node Interpolation Options* dialog. Using the options that are set in this dialog, a set of new nodes can be interpolated between any two selected nodes. If two nodes are selected when this dialog is invoked, the distance between the two nodes is displayed at the top of the dialog. The number of new nodes can be specified in three ways: • *Number of intervals in string.* If this option is chosen, the number of new nodes is one less than the number of intervals specified. • *Number of interpolated nodes.* If this option is chosen, the number of new nodes is exactly specified. • *Total number of nodes in string.* If this option is chosen, the number of new nodes is two less than the number of nodes specified. The *Bias factor* controls the distribution spacing of the new nodes. This factor can be any number between 0.1 and 10.0. A smaller factor will make new nodes be closer to the first selected node while a larger factor will make new nodes be closer to the second selected node. For example, a bias of 2.0 makes the first new node spaced twice as far as the last new node. The *Linear/Arc* option controls the distribution shape of the new nodes. The Linear option causes all new nodes to be in a straight line while the *Arc* option causes all new nodes to form an arc. If the arc option is used, a Radius must also be specified. The arc will be created counter-clockwise from the first selected node to the second.  Interpolation option examples.
Interpolate	After the interpolation options are set up, nodes can be interpolated between any two selected nodes by choosing the Interpolate item from the *Nodes* menu. This operation may be performed multiple times with a single set of interpolation options by selecting any two nodes and invoking the command again. The elevation of each new node depends on the *Insert nodes into triangulated mesh* option in the *Node Options* dialog (see section 1.6.8). If this option is turned on and the new node is inside the finite element mesh, then the elevation is interpolated from the mesh. If this option is turned off or the new node is not inside the finite element mesh, the elevation is interpolated from the two selected nodes.
Find Node	When the **Find Node** command is executed, the *Find 2D Mesh Node* Dialog opens allowing the user to search for a specific 2D mesh node.
Select or Delete Duplicate Nodes	Duplicate nodes are either selected or deleted, according to the option defined in the *Node Options* dialog. The menu item shows either *Select Duplicate nodes* or *Delete duplicate nodes* based on the setting. Two nodes are considered to be duplicates if they are closer together than the *Tolerance* in the *Node Options* dialog. When deleting duplicate nodes, elements attached to deleted nodes will also be removed, unless the *Merge adjacent elements when deleting* option is turned on in the *Node Options* dialog.
Select Disjoint Nodes	Disjoint nodes can be found automatically and selected by choosing the **Select Disjoint Nodes** option from the *Nodes* menu. Disjoint nodes are nodes that are not connected to any elements. Before saving a simulation, it is important to make sure there are no disjoint nodes in the mesh.

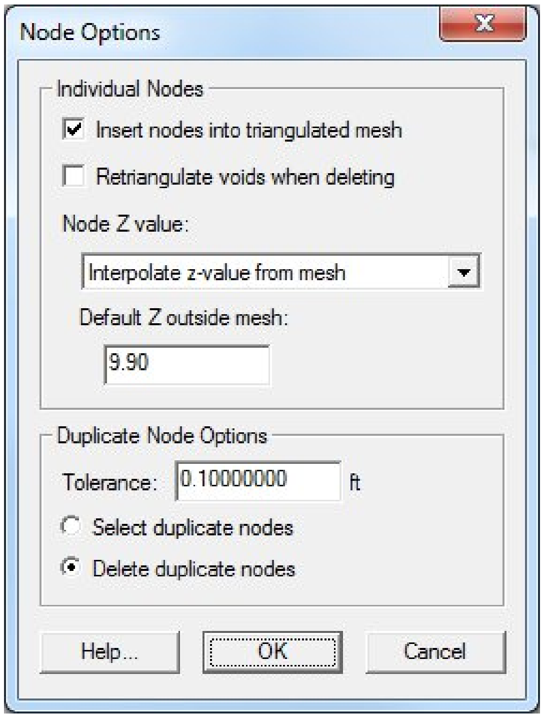
Locked	The nodes in a mesh can be dragged with the mouse cursor if they are unlocked and the Select Nodes tool is selected. The **Locked** item in the *Nodes* menu toggles on and off the node locked status. If nodes are locked, a check mark is shown next to the menu text. The default status is locked so that nodes are not accidentally moved.
Reduce Nodal Connectivity	This command searches through the active mesh looking for wagon wheel nodes. When such a node is found, SMS reduces the nodal connectivity by inserting new node(s).
Renumber	The **Renumber** command from the *Nodes* menu is used to order the IDs of the nodes and elements to make numeric calculations more efficient. The goal is to make the matrix used in calculations as diagonal as possible by having related nodes numbered with indices as close as possible to each other. SMS now utilizes a Cuthill-McKee global renumbering scheme to update these indices. When a mesh is generated, it is resequenced, however, after performing hand edits the mesh should be sequenced again. Multiple invocations of the scheme may result in slightly different sequences.
Transform	The **Transform** command from the *Nodes* menu is used to move a group of selected nodes. If there are no selected nodes, the transformation will be applied to all nodes of in mesh. When this command is executed, the *Nodes Transform* dialog opens. In this dialog, the transformation type can be chosen and then appropriate parameters can be entered. The following transformation types are available: Scaling, translation, datum conversions, and rotations are supported. By default, the image will be framed after the transformation takes place. However, this can be turned off by using the Frame image after transformation option.
Options	Parameters governing the creation and manipulation of nodes are set using the *Node Options* dialog.
Interpolate Nodal Boundary Conditions	If two non-adjacent boundary nodes have been assigned boundary conditions, and the two nodes are selected, this command interpolates the boundary conditions to each of the boundary nodes between the two.

Related Topics

- Mesh Module Tools
- Mesh Module

2D Mesh Node Options Dialog

Parameters governing the creation and manipulation of nodes are set using the *Node Options* dialog, which is opened by selecting the menu *Nodes* | **Options**.



The Mesh Module *Node Options* dialog

Individual Node Options

- **Insert nodes into triangulated mesh** – When a node is created inside the mesh boundary with the Create Mesh Nodes Tool, it can become part of the mesh. If this option is turned off, new nodes are not added to the mesh triangulation and remain disjoint. This option also applies to nodes created using the Interpolate command from the Nodes menu.
- **Retriangulate voids when deleting** – When deleting a node is deleted, all elements attached to the node are also deleted. The void in the mesh left by the deleted elements can be automatically filled by triangulating the surrounding nodes. If this option is turned off, then the void will remain.
- **Node Z value** – The z-value of a node created with the Create Mesh Nodes Tool is based on the chosen option:
  - Interpolate z-value from mesh – The Z coordinate is determined by interpolation from the existing mesh. If this option is turned off or the node is created outside the existing mesh boundary, the default Z coordinate is assigned.
  - Assign default z-value – The Z coordinate is assigned the default value.
  - Prompt for z-value – A dialog will prompt for the Z coordinate of each node after it is created.
  - Interpolate z-value from active scatter – The Z coordinate is determined by interpolation from the active scatter set. If the node is created outside the active scatter set boundary, the default Z coordinate is assigned.

Duplicate Node Options

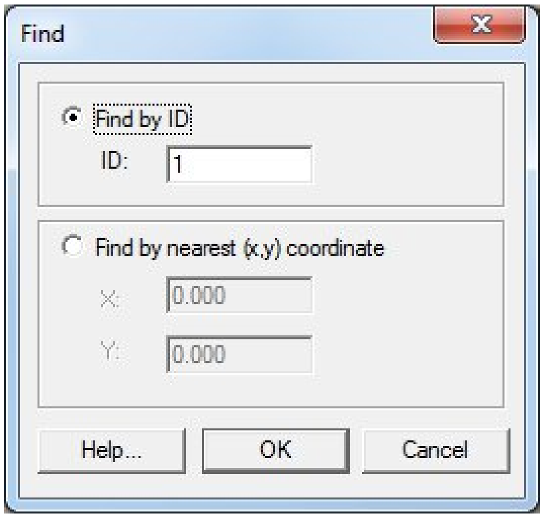
- **Merge adjacent elements when deleting** – When a duplicate node is removed, the adjacent elements are merged.
- **Tolerance** – Tolerance for duplicate node selection and deletion. Also used by the automated mesh generation algorithms of SMS as a minimum node spacing. The tolerance should be specified in feet or meters. If using a Geographic Coordinate System, the tolerance is automatically converted by SMS to meters.
- **Select/Delete duplicate nodes** – The Nodes | [Select/Delete] Duplicate Nodes command is based on this selection.

Related Topics

- 2D Mesh Nodes Menu
- Mesh Module

Find 2D Mesh Node Dialog

The *Find 2D Mesh Node* dialog is used to locate a node using the following methods:



Find 2D mesh node dialog

Find by ID

When the *Find by ID* option is selected, then the node with the specified ID is highlighted with a red circle. If there is no node with the specified id, then an error message is given.

Find by Nearest (x,y) coordinate

When the *Find by nearest (x,y) coordinate* option is selected, the node closest to the specified (x, y) location is highlighted with a red circle.

With either of these methods, if the current tool is the Select Mesh Nodes tool, then the found node becomes selected in addition to being highlighted.

Related Topics

- Mesh Module Menus
- Mesh Module Tools

Renumber

Renumbering a mesh improves the computational efficiency (how fast a model produces a result) of a numeric mesh but should not affect the end results.

Global Renumber

To renumber a mesh select the renumber command in the nodes menu.

Upon execution of this command, the nodes and elements are renumbered using a global renumbering process known as the Cuthill-McKee or Inverse Cuthill-McKee scheme. Other global resequencing methods may be added in future versions. The Cuthill-McKee method searches for a global optimum, but since are often multiple options with the same efficiency level (bandwidth), invoking the command multiple times usually results in different numbering patterns. Each time a mesh is generated, SMS invokes a renumbering command. When nodes are manually added/removed from a mesh, it should be renumbered.

It is important to realize that after renumbering the finite element mesh, any previous boundary condition file or solution file may no longer be valid!

In the case of boundary conditions, SMS associates the specified conditions with the nodestrings, elements or nodes, but the model specific files must be resaved with the new numbering scheme.

In the case of solution files, the numeric engines output values associated with a node id, so the solution is associated with a specific mesh, right down to the numbering of the nodes. The solution would not map to the renumbered mesh correctly and must be regenerated. (The old solution is still valid for the old mesh, but renumbering in effect creates a new mesh.)

Front Width and Band Width

There are two measures of efficiency of a matrix. These include front width and band width. Both can be computed in multiple ways from the grid. The Cuthill-McKee scheme has its own method of computing band width and reports the band width before and after renumbering. In addition, SMS provides an estimate as to how large the front width and half band width may become when running the finite element solver. These estimates are shown in the Mesh Information dialog, which can be opened by performing the *File* | **Get Info** command while in the Mesh Module.

Background information

Due to the number of questions that are asked regarding this subject, this section will attempt to describe, in a broad sense, why renumbering is important.

The finite element solvers use an iterative, banded numerical solver to solve the governing differential equations. If the computer had to simultaneously solve the thousands of equations, much more memory would be required and the process is much less efficient (more time). Meshes with gaps in numbering could lead to errors or singular matrices resulting in no solution with many finite element solvers.

Renumbering organizes the equations in the system of equation so that they can be decomposed and efficiently solved.

Historical

In the past, SMS used a trial an error method of renumbering. The user would select a nodestring and issue the **Renumber** command either from the *Nodestrings* menu or when right clicking on a selected nodestring. SMS would use a sweeping algorithm progressing from the selected nodestring to reassign the node and element numbers. The "row" of elements and nodes adjacent to the string is numbered first. The elements and nodes adjacent to the first set of nodes and elements are numbered next, and so on until all of the nodes and elements have been renumbered. Since the front proceeds from one set of elements to an adjacent set of elements, disjoint portions of the mesh were not visited in the renumbering process. Unvisited nodes and elements were numbered arbitrarily. It was then up to the user to try various starting points, comparing the computed front widths, and keep the numbering that resulted in the smallest front width.

This process did not ensure an optimal numbering, and could become tedious. With version 11.1 of SMS, the global renumbering methodology was added, making the process more efficient. When this modification was implemented, the command to renumber was moved from the *Nodestrings* menu to the *Nodes* menu. However, for convenience, the command was left in the right click menu on selected nodestrings (since many users were accustomed to finding it there). SMS uses the global renumbering algorithm even when the command is issued from this right click menu.

Related Topics

- Nodestrings Menu

Reduce Nodal Connectivity

Many finite element engines (including ADCIRC) have limits on the number of elements that may be attached to a single node. When many elements attach to a single element, the node appears as a hub with many spokes radiating from it. Thus, it is termed a wagon wheel node.

When many elements are connected to a single node, each element has a smaller interior angle. This results in more severe deformations in numerical space. Ideal triangular elements have internal angles of 60 degrees. Ideal quadrilateral elements have internal angles of 90 degrees. For triangles, this would result in six elements connected (or constructed using) a single node. For quadrilaterals, at most four elements would converge at each node.

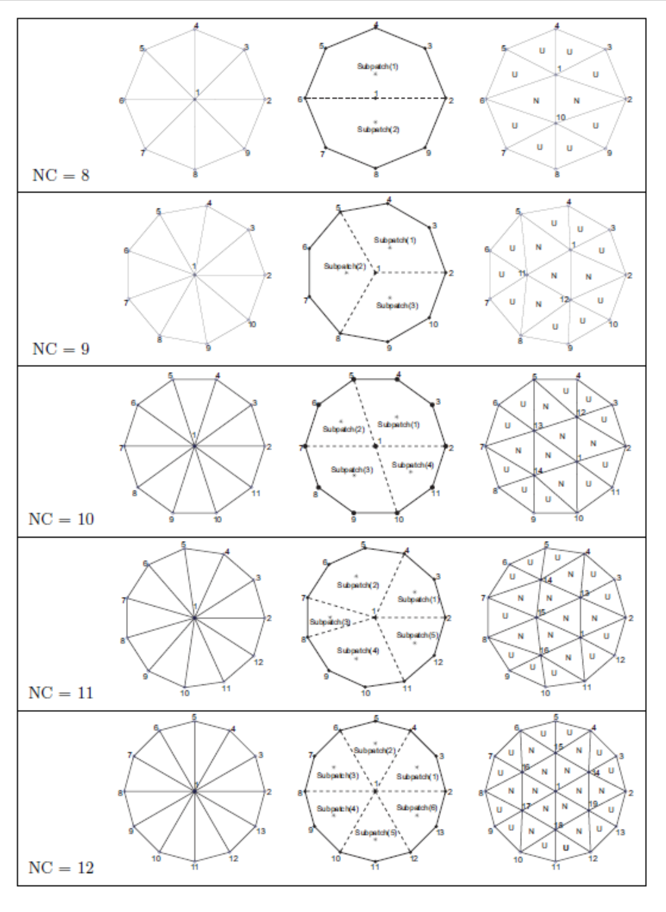
The *node* menu of the Mesh module includes the **Reduce Nodal Connectivity** command which inserts one or more new nodes in the area of wagon wheel nodes resulting in a maximum of 7 adjacent elements.

The command may need to be applied recursively. The first time may reduce connectivity from 12 or more to 8. Then a second application reduces connectivity to 6.

Specifically, if the current connectivity is:

- 8 elements – 1 new node added
- 9 elements – 2 new nodes added
- 10 elements – 3 new nodes added
- 11 elements – 4 new nodes added
- 12 elements – 6 new nodes added.

In each of these cases, the resulting elements all have connectivity of 6 elements. The patterns of insertion are illustrated below:



Related Topics

- Mesh Module Menus
- Mesh Module Tools

3.9 Particle Module

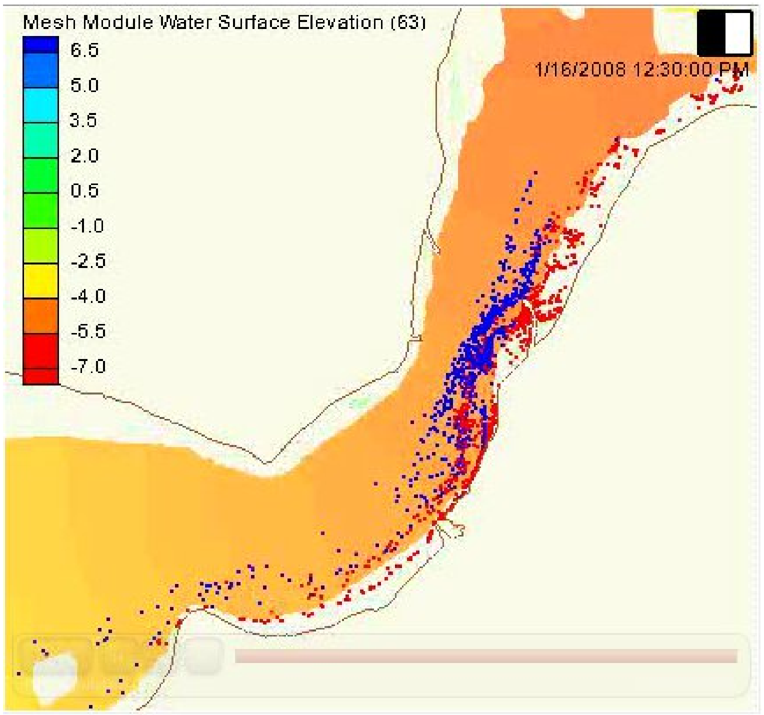
Particle Module

At a glance

- Visualize particle/path data
- Supports PTM module which computes particle positions through time based upon hydrodynamics and wave effects

The particle module contains tools used to work with particle data. Particles can have time varying location and scalar data. The particle module currently includes interfaces for:

- PTM – Lagrangian particle tracker designed to allow the user to simulate particle transport processes.



PTM Particle Trace

Particle Module Tools

See Particle Module Tools for more information.

Particle Module Menus

See Particle Module Menus for more information.

Particle Module Display Options

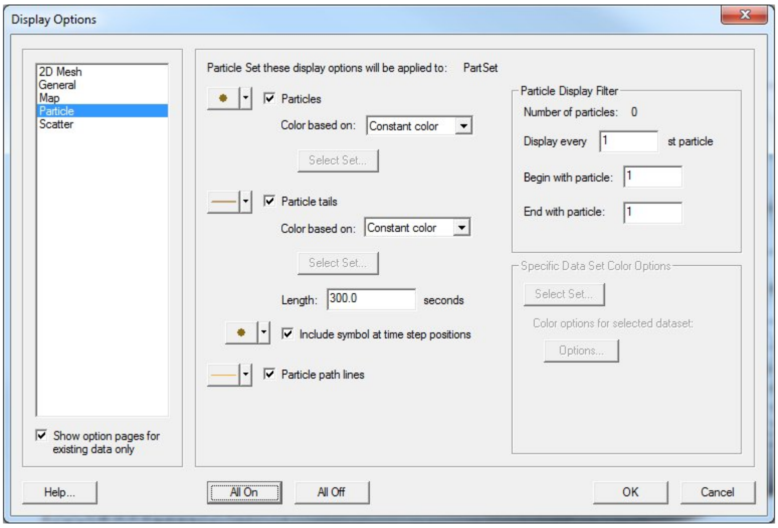
- Display Options

Related Topics

- SMS Modules
- Particle Tracking Model (PTM)

Particle Module Display Options

The properties of all particle tracking data that SMS displays on the screen can be controlled through the *Particle* tab of the *Display Options* dialog. This dialog is opened by selecting *Display* | **Display Options** from the menu bar, the display options macro, or the *Ctrl+D* quick keys.



Display Options dialog showing *Particle* options

The display options defined will only be applied to the active Particle Set displayed in the *Project Explorer*. The active Particle Set is listed at the top of this display options tab. To change to another set, close this window, left-click on the desired Particle Set in the *Project Explorer*, and then reenter the display options.

The entities associated with the Particle module with display options are shown below. These entities also show an **Options** button to the right. For these entities, additional display controls are available. The available particle tracking display options are:

- **Particles**

  A symbol is placed around each particle. The user can specify the symbol, size, and color of these representations by using the **Options** button. The toggle controls the display of the particles. The Color based on sets whether the particles will be colored normally or dynamically. The choices include:

  - **Default –** Displayed using the color defined by the **Options** button.
  - **Active Dataset –** Displayed using the coloring of the active dataset. Available only when a Particle Set exists with a dataset.
  - **Defined Dataset –** Displayed using the coloring of the specified dataset defined using the **Select Set…** button. The name of the specified set will be displayed beside this button. Available only when a Particle Set exists with a dataset.

- **Particle tails**

  A tail is drawn out from the particle as it moves through time. The farther the particle has moved, the longer the tail may be; if the particle has remained stationary, then a tail may not appear. The user can specify the tail style, width, and color by using the **Options** button. The toggle controls the display of the particles. Tail Length, in seconds, controls the amount of tail displayed. The longer (time length) the particle has a tail, the longer the tail may be; if the particle has remained stationary for the duration of the tail length, then no tail will appear. Lengths can be fractions of time steps and the default is 10 time steps. Include symbol can be turned on to display little filled circles at time step positions within the tail. The Color based on sets whether the particle tails will be colored normally or dynamically. The choices include:

  - **Default –** Displayed using the color defined by the **Options** button.
  - **Active dataset –** Displayed using the coloring of the active dataset. Available only when a Particle Set exists with a dataset.

   - **Defined dataset –** Displayed using the coloring of the specified dataset defined using the **Select Set…** button. The name of the specified set will be displayed beside this button. Available only when a Particle Set exists with a dataset.
   - **Same as above –** Displayed using the coloring of the same dataset specified from the Particles. Available only when the Particles are colored based on a defined set and the set has been selected.

- **Particle path lines**

  A path line is drawn from the original position of a particle to every position the particle inhabits thereafter. The path line will remain even after the particle has settled or has crossed the edge of the domain. The user can specify the path line style, width, and color using the **Options** button.

All off unchecks all three particle display options (Particles, Particle tails, and Particle path lines). This also disables the *'Particle Display Filtr* since nothing is selected to be displayed.

All on checks all three particle display options.

Particle Display Filter allows the user to specify the range of Particles, Particle tails, and Particle path lines to be displayed. Filtering the particle set can increase the displaying speed and improve visibility of specific particles or groups of particles. The total Number of particles within the active particle set is displayed for convenience. Display every allows the user to display series of particles, i.e. an input of 1 displays every particle, but an input of 7 will display particles 1, 8, 15, 22, 29… and so forth. Begin with particle denotes the first particle within the range to be displayed. The input cannot be less than 1 or more than the displayed particle set total. End with particle denotes the last particle within the range to be considered for display. If the input is not a multiple of the Display every input plus one, then the particle will not be displayed. For example, an input of 46 with Display every input of 15 will display the last particle because 3 * 15 + 1 = 46. If the input was 45, the particles 1, 16, and 31 will only be displayed (particle 45 is considered, but is filtered out). The input cannot be less than the Begin with particle input or more than the displayed particle set total. One or more of the display options must be on to enable the filter controls.

Specific Dataset Color Options allows the user to Select a specific dataset and adjust its color options (similar to geometry contour options, but specific to each Particle Set dataset) by clicking on **Options**. Color options is available only when a Particle Set exists with a dataset.

Related Topics

- Display Options
- Particle Module

Particle Module Menus

The following menus are available in the Particle Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

Particle Module Data Menu

Most of the SMS modules have a *Data* menu, but the items in this menu are different for each module. The Mesh Module commands include:

Dataset Tools

- Data Calculator
- Dataset Toolbox
- Create Datasets
- Compute Grid Datasets

Visualization Tools/Options

- Film Loop

Model Specific Menus

- PTM

Particle Module Tools

The Particle Module tools are contained in the Dynamic Tools portion of the tool palette when the Particle Module is active.

Select Particles

The **Select Particles** tool selects a single particle with a left mouse click. A group of particles can be selected by dragging a box around them. Particles may be added to the selection by holding the *SHIFT* key and selecting additional particles. The main use of selecting particles in the particle module is to query the particle properties. When a single particle is selected, the *Info Window* will show the ID and the edit window will show the location and value of the active particle dataset. If exactly two particles are selected, the *Info Window* will also show the exact distance between the two selected particles. If more than one particle is selected, the minimum, maximum, and average value of the active particle dataset will be shown.

Related Topics

- Particle Module

Particle Report

When creating a drogue plot, SMS can write out the following statistical information for each particle:

- The particle's starting and ending location.
- The start and end times for each particle in decimals days.
- The total distance it traveled over the length of the animation.
- The minimum, maximum, and average velocity at which it traveled.

Be aware that when a particle leaves the domain, it can no longer be tracked so the ending location will be the point at which it left the domain. You can click the **Browse** button to set the name of the report file.

Related Topics

- Film Loop Setup – General Options
- Film Loop Setup – Drogue Plot Options

Extract Particle Subset

This command writes a portion of a particle set to a new a particle file. This portion may be a subset of the timesteps for the particle solution, a subset of the particles, or both. You get to this command by right-clicking on a particle set and choosing **Extract subset (times/particles)**.

The required information for this command includes:

- The filename to write the particle subset to.
- The first and last time steps that define the range of time you want export.
- Whether to write every time step, every other time step, every third time step, etc.

You may define a subset of the particles to export by setting up filters (see PTM Particle Filters). For example, you may decide to only write particles with a specific range of grain size, or particles from a specific source.

The extracted particle set will have the same datasets that exist in the original particle set.

3.9.a. Particle Module Datasets

Particle Grid Dataset Bin Elevations

This article documents a feature that is under development

Datum Definition

The particle module compute grid dataset bin elevations dialog is accessed through the *Compute grid datasets* dialog. The bin elevations are specified according to the datum selected in the *Compute grid datasets* dialog.

Fixed Datum

When using a fixed datum, the bins have a constant z-elevation. Be careful if your model uses a depth dataset. For example, if the water surface is at a constant value of 0.0 meters and the depth is a constant value of 10.0 meters, you would want your bins to range from 0.0 to -10.0 meters.

Bathymetry as Datum

When using the bathymetry as a datum, the bins are offsets from the specified bathymetry dataset. This is useful when determining the effects of concentrations on bottom dwelling species, oyster beds, etc. Offsets should always be positive values.

Water Surface Elevation as Datum

When using the bathymetry as a datum, the bins are offsets from the specified bathymetry dataset. This is useful when determining the effects of concentrations on fish migrating at a specified depth, etc. Offsets should always be positive values.

Related Topics

- Particle Module Compute Grid Datasets Dialog

Particle Module Compute Grid Datasets

The particle module *compute grid datasets* dialog is accessed through the particle module data menu. This dialog is also used to compute datasets on fence diagrams (defined in a coverage).

The datasets are added to the active cartesian grid. It should be noted that the resolution of the grid will have an impact on the numerical values computed for several of these datasets. For example, the accumulation in a cell is computed as the volume of particles which have settled in a cell divided by the cell area. If the cells are larger, this will result in a smaller accumulation. You may want to experiment with a few different grid resolutions.

Currently the following grid datasets can be created in the particle module *create datasets* dialog:

- Particle Count
  - The number of particles in the Cartesian grid cell.
  - Accumulation
    - The depth of particles in the Cartesian grid cell. The volume of particles is calculated using the particle mass and density dataset for particles which are inactive (based on the state dataset) and in the cell. The volume in each cell is divided by the area of the cell to calculate an average depth in the cell. No voids ratio is included at this time, however the Data Calculator can be used to change the resulting Cartesian grid dataset.
    - Rate of accumulation
      - The change in accumulation (as described above) over time.
      - Deposition
        - The change in depth of particles in the cartesian grid cell during the focus time. The volume of particles is calculated using the particle mass and density dataset for particles which have become inactive (based on the state dataset) during the focus time. This volume is then divided by the area of the cell to calculate an average depth in the cell. No voids ratio is included at this time, however the Data Calculator can be used to change the resulting cartesian grid dataset.
        - Concentration
          - The concentration of particles in the Cartesian grid cell. The volume of particles is calculated using the particle mass and density dataset for particles which are active (based on the state dataset) and in the cell. This volume is then divided by the volume of the cell using the bathymetry and water surface elevation datasets. For this purpose, the bathymetry must be specified as an *elevation* dataset. This means that the values must be specified as positive upwards from the same datum the water surface dataset is measured from. For models such as CMS-Flow and ADCIRC, which require that the user specify positive depths (values measured down from mean sea level), the geometry (Z) dataset must be inverted to use for this application. The depth of the water column in a cell of the resulting grid is computed as $WSE - Elevation$. This depth is combined with the cell area to compute a cell volume. SMS is capable of extracting ground elevations (bathymetry) and water surface elevations from either a finite element mesh, or a scattered data set. If the data is only available on a Cartesian grid (i.e. a CMS-Flow simulation), the user must convert this to a scatter set. Right-clicking on the grid and selecting the *Convert*|**2D Grid → 2D Scatter**, will accomplish this. The datasets on the grid will be converted to the scattered data as well.
          - Exposure
            - The cumulative concentration relative to time in the Cartesian grid cell.
            - Dosage
              - The exposure in the Cartesian grid cell during the focus time.

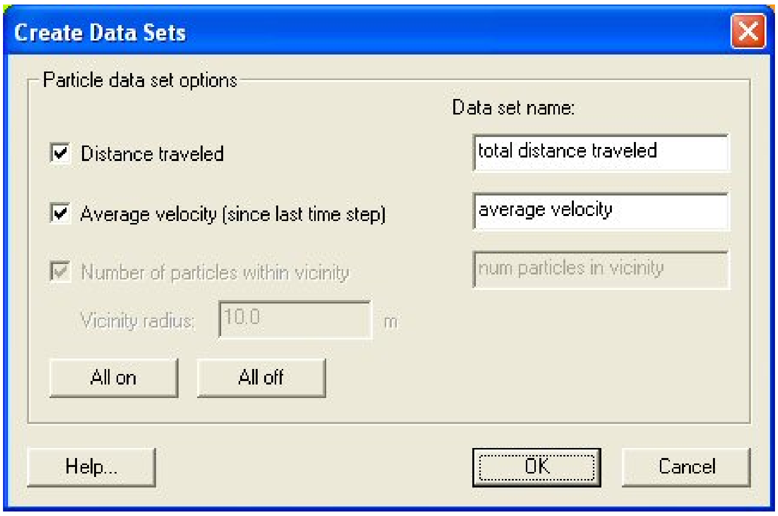
Related Topics

- Bin elevations dialog

Particle Module Create Datasets

The particle module create datasets dialog is accessed through the particle module data menu. Currently the following particle datasets can be created in the particle module create datasets dialog:

- Distance traveled
  - For each time step, computes the total distance each particle has traveled since the particle was born.
  - Average velocity (since last time step)
    - For each time step, computes the average velocity of the particle since the previous time step.
    - Number of particles within vicinity (under development)



Example of the *Create Dataset* dialog.

Related Topics

- Particle Module Menus

PTM Create Grid Datasets – Fence Diagrams

The Particle Module Compute Grid Datasets page describes how to represent particle data on a rectilinear grid. The computations include things such as count, accumulation, and concentrations on the grid cells. In addition to computing these values on 2D grid cells, some of the datasets can be computed in layers creating 3D data. These datasets include concentration, exposure and dosage.

3D Fence Options

If the "create fence diagram" option is selected SMS will build a 3D mesh and datasets for each of the selected 3D datasets. SMS will also turn on the option to display fences (found in the display options dialog) and set the coverage used for fences. See 3D Fence Diagrams below for information on adjusting the display of the fence.

There will be two datasets generated for each type of 3D data computed. One dataset represents the concentration/exposure/dosage that is experienced by the cell. These values only make sense when applied to a volume. These datasets will display as a block filled value in each cell. Some people prefer viewing smooth contours rather than block filled values. The second dataset that is created represents the cell based values averaged to the nodes to provide for smooth contours. These datasets have "smoothed" in their names to distinguish them from the cell based data.

3D Fence Diagrams

3D fence diagrams allow you to view a cross-section of a 3D solution. To create/view a 3D fence do the following:

Displaying 3D fences requires:

1. A 3D mesh with solution datasets.
2. A coverage of any type that has one or more feature arcs without any vertices. This defines where the fences will be located. The arcs cannot have vertices since only planar surfaces can be represented.

3D fences can be turned on in the display options dialog. The coverage used for the fence definitions is specified in the display options dialog. The fences will use the current contour settings and are always represented with color-filled contours.

Remember to rotate out of plan view to see the fence.

3.10 Raster Module

Raster Module

At a glance

- Open and visualize raster data
- Supports many gridded elevation file formats. A complete list can be found at: www.globalmapper.com/product/formats.htm [1]
- One or more rasters are placed under a raster set in the project explorer.
- Convert raster to TIN (scatter set)
- Interpolate data from raster to TINs /2D Mesh/2D Grid
- Use rasters with observation profile plots

Rasters contain data (usually elevation) stored in pixels. Their resolution can vary, depending on the number of x and y cells the raster contains. The Raster Module allows us to open and visualize rasters of various formats and convert them into TIN (scatter sets), and interpolate their data into scatter sets, 2D meshes, and 2D grids.

Raster Sets

Each raster will be stored in a raster set. A raster set may contain multiple rasters. Ofter the data in one raster is also associated to another raster, when each raster is used to cover an area. When this is the case, you should group the raster's together under a single raster set. Raster sets are used in doing interpolation and plotting.

A new raster set can be created by right-clicking on the root raster item in the project explorer, then select **New Raster Set**. Existing raster's can be moved into a raster set by simply dragging the raster into the raster set.

Raster Functionalities

Profile Plot

SMS can generate many different types of 2D plots. Scalar data contained in visible rasters may be plotted using a specific type of 2D plot called a profile plot. When creating a profile plot for a raster using the Plot Wizard and 'specified data' is selected in Step 2, be sure to set the appropriate "Raster Set". Data will be plotted from the rasters in the selected raster set.

Interpolation

Values contained in rasters can be interpolated to existing scattersets, 2D meshes, and 2D grids. To interpolate the values from a raster, right-click on the raster set in the *project explorer* and choose **Interpolate**. Then choose the appropriate option for which object you wish to interpolate to.

Convert to Scatter

Values from rasters can also be converted into scatter data. This is done by right-clicking on the raster item in the *Project Explorer* and selecting *Convert* | **Raster→2D Scatter**.

In special cases, one would want to convert only a selected portion of the raster to scatter data. For more information, see Raster Tools.

Raster Values as Elevation / Z Data

By default data associated with the raster is stored as elevation/z data. If a rasters projection is changed and the units change from meters to feet, the z values also would change. This becomes problematic if the raster values is used to represent NCLD land data and not elevation data. To keep NCLD land data from being projected, you can specify this in the raster set. This is done by right-clicking on the raster set and selecting **Options..**. A dialog appears allowing you to specify if the values are elevations or not elevations. If raster values are set as not elevations, when doing a projection the z values will remain unchanged.

Related Topics

- Plot Window

Raster Module Interface

Clicking the raster module icon in the module tool bar will bring the raster module interface to display on the screen. The raster module interface can also be brought to display by clicking on a raster item in the project explorer. The raster module interface consists of the menu, tools, and project explorer right-click menus.

Raster Menu Items

The menu bar that appears in the Raster Module:

- File – Has standard features as in other models.
- Edit – Has standard features as in other models.
- Display – Under the *Raster* tab in *Display Options*, you can choose to *Display as raster* or *Display as surface*. *Display as raster* is a 2D representation, while *Display as surface* uses data from pixels as elevation to show a 3D surface. It should be noted that these display options affect performance. *Display as raster* takes less memory and performs faster because it is a more efficient data structure. If *Display as surface* is selected, you will be able to turn on or off contours, edges, or boundaries.
- Web – Has standard features as in other models.
- Window – Has standard features as in other models.
- Help – Has standard features as in other models.

Raster Tools

The Raster Module contains only one dynamic tool called the **Select Points** tool. Using this tool, one can edit/select one or more corners (points) of the pixels which make up the raster. Selecting cells with the **Select Points** tool will show information about the points in the status bar.

Right-Click Option

When one or more points are selected, one can right-click in the *graphics window*, and the option to convert selected raster points to a scatter set appears. If this option is clicked, a new scatter set will appear in the *project explorer*.

Raster Right-Click Menus

Right-Click Menu of "Raster Set" Project Explorer Item

Right-clicking on the raster item in the *Project Explorer* will bring up the following options:

- Delete – deletes the current coverage.
- Rename – lets user rename the current coverage.
- Options – Clicking this will bring up a *raster options* dialog. This allows you to specify if your z values are elevations. If NCLD land data is being used, you should specify that raster values are not elevations. Then if you were to change projections, the z values that store your land data would not be affected.
- Interpolate – Data from raster sets may be interpolated to already existing TINs, meshes and 2D grids.

Right-Click Menu of "Raster" Project Explorer Item

Right-clicking on the raster item in the *Project Explorer* will bring up the following options:

- Delete
- Rename
- Convert – SMS allows rasters to be converted into TINs.
- Zoom to Raster

Right-Click Menu of Dynamic Tools

See Raster Tools above.

Related Topics

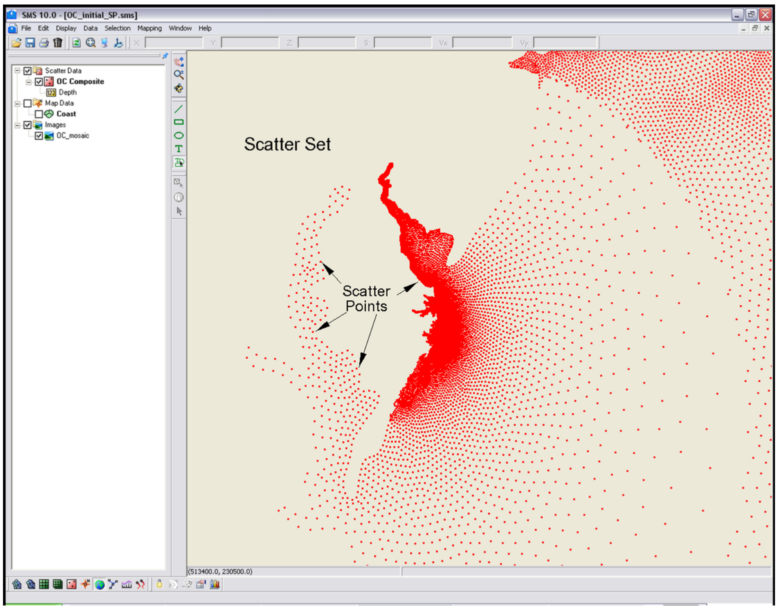
- Project Explorer
- Toolbars
- Menu Bar
- Help or Status Window
- Dynamic Tools

3.11 Scatter Module

Scatter Module

At a glance

- Used to create, edit, and visualize triangulated irregular networks
- DEMs can be read in and converted to TINs
- Filter scatter sets to eliminate redundant data
- Datasets can be interpolated to other modules (meshes, grids, etc)



Scatter Set Example

Overview

The Scatter Module (previously known as the Scattered Data Module) is used to interpolate spatial data values from groups of scattered data points or ordered grids (DEMs) to the other data types (i.e., meshes and grids). SMS supports three interpolation schemes including linear, natural neighbor and inverse distance weighted. The module is also used to view and edit survey data (i.e. SHOALS data).

Interpolation is useful for setting up input data for analysis codes. Generally, the data gathered from a site to be modeled varies in density. Generating a finite element mesh directly from these points would result in a very low quality mesh. Further this data does not lie in a grid for use as a finite difference grid. Interpolation allows the gathered data points to be used as background information. The user may then generate a base mesh or grid in the Mesh Module, the Grid Module or the Map Module. The only consideration of bathymetry for such a mesh or grid would be the definition of element edges along geometric or property features. The actual bathymetry comes from the scattered data. SMS interpolates this data to the created mesh or grid points.

Interpolation may also be used to create datasets for one mesh from data related to another mesh of the same region. For example, a user may have a mesh of a river reach for which analysis has been preformed. If a bridge is to be added to the reach, the mesh topology changes. The data from the first mesh can be converted to a scattered dataset and then interpolated to the second mesh. This data may be used as initial conditions for the second mesh, or compared to results of analysis run on the second mesh using the Dataset Toolbox.

A third purpose of interpolation is to create additional datasets from either observed, or calculated data.

Data Sources

There are various potential sources for background data in an SMS project. These include:

- **local surveys**

  Local surveys must be formatted into an SMS supported format. The most intuitive format and easiest to use is a tabular file of coordinates.

  If this data is to be augmented with previous models or historical surveys, the coordinate system of the local survey must be defined relative to the historical survey or a global coordinate system.

- **historical surveys**

  There are several sources of historic surveys. These include previous studies done by a modeler or company and compiled databases such as GEO-DAS or ETOPO. These data sources can be imported into SMS and used either as the basis of a finite computation domain (mesh or grid), or as a scattered dataset or DEM. Care must be taken into account the age and quality of the data and make sure all data sources are converted to a single coordinate system.

- **digital elevation maps**

  Digital Elevation Maps (DEMs) are regular structured grids of elevation values. Since the data is structured, it can be read, stored, displayed and utilized more efficiently than scattered datasets. These data sources are becoming more prevalent and can be obtained for topographic regions of the entire United States and several other area of the world from web sites such as Terraserver.

  Unfortunately, most DEMs available online do not include bathymetric portions of the domain, which makes their use in SMS limited. New data bases are being developed, but due to the dynamic nature of bathymetric information, the feasibility of an extensive database is very slight.

  It may be useful to convert scattered datasets into DEMs for faster processing inside of SMS.

- **electronic charts**

  Since surveys can be expensive to obtain, and DEMs may not be applicable, another option available for the hydraulic modeler is the use of topographic/bathymetric charts or historic nautical charts. If these types of maps can be digitized into an electronic format, they can be read into SMS and displayed on the screen. The goal is to create a scattered dataset from this electronic chart. The steps to do this include:

  1. Scan the paper map and save it as an image (*.tif, *.jpg, ...).
  2. Register the image (you may want to mark the map with your register points prior to scanning it).
  3. Select the Create Vertex tool in the Data Module.
  4. Digitize (click on the image on the screen) to create a vertex on a contour line in the image.
  5. In the z edit box of the edit window set the z value to the contour value of the line.
  6. Digitize along the specified contour value (the spacing of points along the contour lines should be approximately the same distance as the spacing between adjacent contours).
  7. Repeat steps 4–6 for each contour line. Spot elevations can be entered by setting the z value to the value of the spot elevation and then creating a vertex at that location.
  8. Triangulate the vertices once you are done.

  This method becomes tedious for larger areas, but is ideally suited for smaller areas where there are not too many contours to be digitized.

  In addition, when DEM data is brought into SMS, the data is triangulated and stored as a scatter set.

  It is also possible to convert CAD and GIS data into scatter sets. This is accomplished by right-clicking on the object in the project explorer and selecting the **Map → Scatter** command. This command searches the data for triangular and quadrilateral faces and converts them to triangles in a triangulated surface (TIN). Points along contours or polylines are not converted using this command. In order to use these points in a scattered dataset format, first convert them to feature objects in the Map module.

Practical Notes

Does SMS have a way of measuring the difference in volume between two bathymetric surveys?

To do this you need to do the following:

1. Interpolate the elevation from one survey onto the other.
2. Use the *data calculator* to compute the difference between the two elevations. You may want to do max(0.0, z1-z2) as well as max(0.0, z2-z1) to get both deposition and erosion volume.
3. Turn on the *Volume* option in the *info options* dialog.
4. Select the triangles you are interested in. The volume appears in the info window at the bottom of the screen. You can also direct these values to a file or another window through the Info Options settings.

How do I compare datasets from different scatter point sets?

Datasets within a scatter set are associated with the geometry of that scatter set. To compare datasets from different scatter sets, it is necessary to first interpolate the datasets to a common geometry. Below are guidelines on how to do this with a mesh and with a scatter grid.

- Mesh
  1. Interpolate the first dataset to mesh.
  2. Interpolate the second dataset to the mesh.
  3. Use the *data calculator* (*Data* | **Data Calculator**) to compare the two datasets.
- Scatter Grid
  1. Select first dataset.
  2. Select *Scatter* | *Interpolate to Scatter* | **…to Scatter Grid**. Specify extents and resolution of grid.
  3. Select second dataset.
  4. Select *Scatter* | *Interpolate to Scatter* | **…from other scatter set**. Specify the second scatter set.
  5. Select *Data* | **Data Calculator** with the new scatter grid selected to compare the two datasets.

Scatter Module Tools

See Scatter Module Tools for more information.

Scatter Module Menus

See Scatter Module Menus for more information.

Scattered Datasets

The Scatter Point Module is used to visualize and apply various types of data. This data typically comes from surveys, digital maps, previous numerical analysis or digitization on screen. The data is stored as sets or groups of 2D scattered data points with associated values. The most common value is bathymetry and is used to create the geometric representation of the area being modeled.

SMS connects the scattered data points into triangles forming a Triangulated Irregular Network (TIN). TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for a source of bathymetric or other data in a numerical model. TINs can also be used to compute areas, volume, distances, gradients and several other geometric parameters.

SMS applies data from scattered datasets to finite element networks or grids via interpolation. This allows poorly distributed elevation data to be assigned to a well-structured set of elements to create the bathymetry of the entire mesh. A variety of interpolation schemes are supported. Internally, the scattered data sets are triangulated to create surfaces for continuous interpolation. Since the connectivity of the triangulation affects the interpolation, SMS provides tools to allow for the manipulation of this triangulation. The triangulation also allows contouring of the scattered data set to visualize the data.

Multiple scatter point sets can exist at one time in memory. One of the scatter sets is always designated as the "active" scatter point set. The active scatter set can be changed by changing the *Scatter Set* combo box in the top *Edit Window*. Whenever a new scatter set is created, it becomes the active set.

Related Topics

- Digital Elevation Maps
- Mesh Generation
- Scatter Interpolation
- Scatter Module Display Options

External Links

- Jun 2002 ERDC/CHL CHETN-IV-43 SHOALS Toolbox: Software to Support Visualization and Analysis of Large, High-Density Datasets [1]

3.11.a. Interface Components

Scatter Interface Components

The scatter module interface consists of the display options, menus, right-click menus, and tools associated with the scatter module.

- Display Options – accessed through the *Display Options* dialog. Provides options as to how Scatter Module elements will be displayed.
- Menus – besides the standard menus, the Scatter Module has five menus associated with it: Data, Vertices, Breakline, Triangles, and Scatter.
- Right-Click Menus – the Scatter Module has right-click menus for root folders and scatter set items.
- Tools – the Scatter Module has eight unique tools for creating and manipulating scatter data in the Graphics Window.

Related Topics

- Display Options
- SMS Menus
- Dynamic Tools

Scatter Module Menus

The following menus are available in the the Scatter Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Data
- Vertices
- Breaklines
- Triangles
- Scatter

Related Topics

- Scatter Module

Scatter Module Display Options

The properties of the scatter data SMS displays on the screen can be controlled through the *Display Options* dialog. The entities associated with the scatter module with display options are shown below. Some of these entities also show an **Options** button to the right. For these entities, additional display controls are available. The available display options include the following:

- **Points** – A symbol is drawn at each point. The user can specify the type, radius, and color of these symbols. The toggle below the Points item allows the user to specify that rather than coloring the symbols with the specified color, a contour color should be used based on the current scalar value at the point. This gives a contouring effect without generating/displaying the contours.
- **Triangles** – Triangle edges are drawn using the specified line attributes. Line attributes include color, thickness, and style (dashed/solid).
- **Boundary** – A line around the perimeter of the scatter set can be drawn. This is useful when the triangles are turned off. User specifies line color and thickness.
- **Contours** – The scatter contours are drawn for the active scalar data set for the active scatter set. All standard contour display options are supported for scatter contours.
- **Velocity Vectors** – The scatter vectors are drawn for the active vector data set of the active scatter set. Display options are set through the *Vector Display Options* dialog.
- **Inactive Color** – Only the active scatter set is displayed in its color. All other scatter sets are displayed using the inactive color. This helps to avoid clutter on the screen.
- **Nautical Grid**
- **Point Names** – The name of the selected scatter set can be changed.
- **Point Numbers** – The scatter point id number can be displayed next to each node. User selects font and color.
- **Scalar values** – The scalar value of the active function is displayed next to each point. The options button opens the *Scalar Value Options* dialog.
  - **Use contour color scheme** – Use the color ramp specified for the contours for text color rather than a specified color.

  The display of individual scatter sets can be turned on or off through the Project Explorer.

Related Topics

- Display Options
- Scatter Module

3.11.a.1. Scatter Module Tools

Scatter Module Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Scatter Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Scatter tool palette.

Tool	Tool Name	Description	Right-Click Menu
	Select Scatterpoint	The **Select Scatter Point** tool is used to select scatter points (also known as vertices). A single point is selected by left-clicking directly on it. Multiple points can be selected at once by dragging a box. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. Additional scatter points can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new points without holding the *SHIFT* key will first clear the selection list and then add the newly selected points. A selected point can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Scatter points are locked by default so they are not accidentally dragged, but can be unlocked using the *Scatter Vertices* Menu. When a single point is selected, its location is shown in the *Edit Window*. The Z coordinates can be changed by typing in the edit field. The active scalar function is updated when the Z coordinate changes. If multiple points are selected, the Z Coordinate value shown is the average scalar value of all selected points. If this value is changed, the new value will be assigned to all selected points.	**Find –** Available when nothing selected. Brings up the "Find..." dialog to search for a vertex by ID or location. Like the menu command. **Scatter Options –** Available when nothing selected. Brings up the *Scatter Options...* dialog. Like the menu command. **Triangulate –** Available when nothing selected. Triangulates the entire scatter set. Like the menu command. **Select Thin Triangles –** Available when nothing selected. Selects thin triangles around the boundary. 'Thin' is defined by the aspect ration in the scatter options. Like the menu command. **Delete Long Triangles –** Available when nothing selected. Deletes long triangles around the boundary. 'Long' is defined by the aspect ration in the scatter options. Like the menu command. **Select All –** Available when nothing selected. Selects all vertices. **Delete –** Available when one or more vertices selected. Deletes the selected scatter vertex/vertices.

		The *Graphics Window's* status bar will display information on the selected items depending on the settings find through the *File* | **Info Options** command in the *File* Menu. Selected scatter points can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* Menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Triangles attached to the deleted scatter points are deleted. The resulting void can be retriangulated. This tool is available when one or more scatter points exist.	**Split Breaklines –** Available when one or more vertices selected. Splits all breaklines that go through the selected vertex into two breaklines (one on each side of the selected vertex). If the vertex is the end of the breakline or is not used in any breaklines, the command has no impact. **Assign Point Name... –** Available when one or more vertices selected. Prompts the user for a name for this point. The default name will be "Point #" where '#' is the ID of the point. Points do not have names unless specifically specified. Names can be displayed in the Scatter Module display options. **Clear Selection –** Available when one or more vertices selected. Unselects all the selected vertices. **Invert Selection –** Available when one or more vertices selected. Selects all the unselected vertices and unselects all the selected vertices. This can be useful when many vertices are to be deleted. First select those to be kept and then invert the selection. **Zoom to Selection –** Available when one or more vertices selected. Recursively zooms in on the selected objects. Invoke this command multiple times to zoom in tighter.
	Create Scatter Point	The **Create Scatter Point** tool is used to place new scatter point in a Scatter Set. A single point or vertex is created at a time by left-clicking at the coordinate desired. The newly created point is selected to allow Z Coordinate changes in the Edit Window. Scatter points are locked so X and Y Coordinates can not be edited. This tool is always available, however, creating a scatter point is only allowed while in plan view.	N/A

	Select Breaklines	The **Select Breaklines** tool is used to select breaklines. Holding the *SHIFT* key while selecting breaklines will add breaklines to the selection that are not already selected and remove breaklines that are already selected from the selection. Multiple nodestrings can be selected by dragging a box around the breaklines. Holding the *CTRL* key and clicking will enable selection with a polygon. Holding the *CTRL* key and dragging will enable selection with an arrow. The *Edit Window* shows the number of breaklines selected.	**Delete Selected** – deletes the selected scatter breakline(s). **Merge Selected** – Merges multiple breaklines that connect end to end into a single breakline. An error is reported if two (or more) breaklines are selected that do not have common end points or a single break line is selected. **Force Breaklines** – Swaps the triangle edges that cross the selected breakline(s) to ensure no triangles cross the breakline(s). **Clear Selection** – Unselects all the selected breaklines. **Zoom to Selection** – Recursively zooms in on the selected objects. Invoke this command multiple times to zoom in tighter. If no breaklines are selected, this command zooms to all breaklines. **Select All** – Selects all breaklines.

	Create Breaklines	The **Create Breaklines** tool is used to create breaklines. Breaklinesare used to control the connectivity of a scatter set. To create a breakline:
		1. Click on a scatter vertex. The vertex will be highlighted in red and a prompt will be shown in the *Help Window*.
		2. Click on any vertex to add it to the breakline. The selected vertex is also highlighted in red and a solid red line is drawn between the two vertices. Continue adding vertices to the breakline in this manner.
		  1. Note: Vertices in the breakline can be adjacent, but this is not required. A breakline will usually be made of vertices which are not adjacent.
		  2. Press the *BACKSPACE* key to backup one vertex. Press the *ESC* key to abort the breakline creation.
		  3. Double-click a vertex or press the *ENTER* key to end the breakline creation.
		3. The *SHIFT* and *CTRL* keys assist in creating large breaklines which include sections made up of adjacent nodes. These can be used after at least one vertex has been selected and function as follows:
		  1. *SHIFT* – Holding down the *SHIFT* key and selecting another vertex will add to the breakline all vertices between the two. The path chosen is the shortest distance between the two vertices that follows triangle edges.
		  2. *CTRL* – Holding down the *CTRL* key and selecting another vertex will add to the breakline all vertices on the scatter set boundary between the two, going counter clockwise from the first vertex to the second vertex. Both vertices must be on the boundary of the scatter set or SMS will beep.
		  3. *CTRL + SHIFT* – Holding down both the *CTRL* and *SHIFT* keys and selecting another vertex will add to the breakline all vertices on the scatter set boundary between the two, going clockwise from the first vertex to the second vertex. Both vertices must be on the boundary of the scatter set or SMS will beep.
			N/A

	Select Triangle	The **Select Triangle** tool is used to select triangles. A single triangle is selected by clicking inside it. Multiple triangles can be selected at once by dragging a box or an arrow. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. To drag a selection arrow, left-click while holding the *CTRL* key (the *CTRL* key can be released after the click) and hold the mouse button and drag the mouse to form an arrow of the desired length and direction; release the button to impale and select the triangles through which the arrow passes. Additional triangles can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new triangles without holding the *SHIFT* key will first clear the selection list and then add the newly selected triangles. A selected triangle can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESCAPE* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. When a single triangle is selected, its centroid location is shown in the *Edit Window*. If multiple points are selected, the Z value shown is the average scalar value of all selected triangles. The *Graphics Window's* status bar will display information on the selected items depending on the settings find through the *File* | **Info Options** command in the *File* Menu. Selected triangles points can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* Menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. The Scatter vertices (points) of the deleted triangles are not deleted. The resulting void can be retriangulated. This tool is available when one or more scatter triangles exist.	**Select Thin Triangles –** Selects thin triangles around the boundary. 'Thin' is defined by the aspect ration in the scatter options. Like the menu command. **Delete Long Triangles –** Deletes long triangles around the boundary. 'Long' is defined by the aspect ration in the scatter options. Like the menu command. **Select All –** Available when nothing selected. Selects all vertices. Process Boundary Triangles – Available when nothing selected. Like the menu command. **Delete –** Available when one or more triangles selected. Deletes the selected scatter triangle(s). **Clear Selection –** Available when one or more triangles selected. Unselects all the selected triangles. **Invert Selection –** Available when one or more triangles selected. Selects all the unselected triangles and unselects all the selected triangles . This can be useful when many triangles are to be deleted. First select those to be kept and then invert the selection. **Zoom to Selection –** Available when one or more triangles selected. Recursively zooms in on the selected objects. Invoke this command multiple times to zoom in tighter.

	Create Triangle	Most triangles in SMS will be created using automatic triangulation. At times, however, it is necessary to create a triangle, especially after deleting triangles. To create a single triangle, select the vertices of the desired triangle by left-clicking on each scatter point or by dragging a selection box. As points are selected individually, they will be highlighted. Also, triangle edge will be drawn between the first and second vertex. To remove the last highlighted point from the desired triangle, press the *DELETE* or *BACKSPACE* keys. To abort the creation of a triangle, press the Escape key. When three points are selected, SMS will try to create the desired triangle. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. The selection box must only contain the three desired vertices to create the triangle. If the new triangle will overlap existing triangles the triangle will not be created. If more then three points are selected (via a selection box) then SMS will not attempt to create a new triangle. This tool is available when three or more scatter points exist.	N/A
	Swap Triangle Edge	The **Swap Edges** tool is used to manually swap the edges of two adjacent triangles. This is useful in such cases as avoiding an artificial dam in a channel. Left-click on the desired triangle edge to swap it. The edges of the triangles will only be swapped if the resulting triangles are valid. This tool is available when two or more scatter triangles exist. Swapping edges is only allowed while in plan view.	N/A
	Create Mesh Nodes Contour	The **Swap Edges** tool is used to temporarily display a Z value on a TIN. Left-click at a location to display the value. The labels will not reappear if the graphic window view changes. This tool is available when at least one scatter triangle exists. Contour labeling is only allowed while in plan view.	N/A

Related Topics

- Scatter Module

3.10.a.2. Scatter Module Menus

Scatter Menu

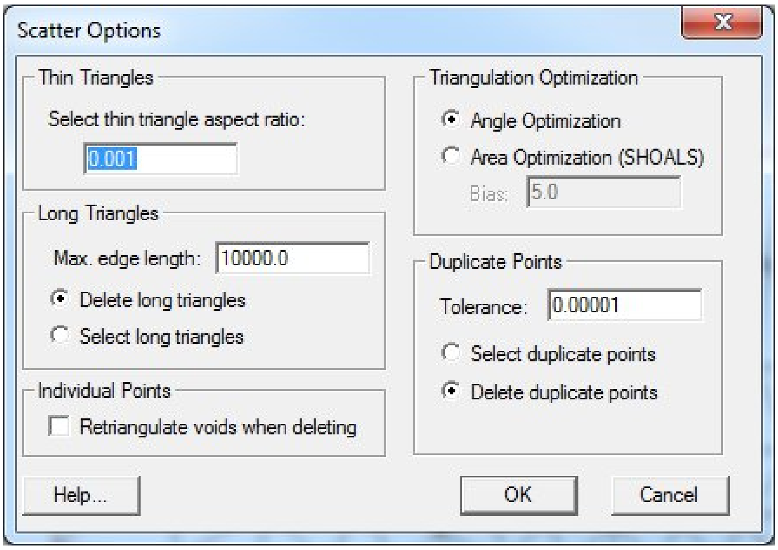
The items in the *Scatter* menu in the Scatter module are described below. The menu items all work with the active scatter set unless otherwise noted.

General Commands

Listed below are the general commands in the *Scatter* menu.

Scatter Options

Scatter options are accessed through the *Scatter* menu, *Scatter Options* dialog in the Scatter Module.



'''Scatter Options *dialog*

Triangulation Options

This section lets the user adjust the maximum aspect ratio of a thin triangle. The aspect ratio is the ratio of the triangle width to the triangle height. All triangles with an aspect ratio less than what is specified are considered thin.

Long Triangles

This section contains options for deleting or selecting long triangles.

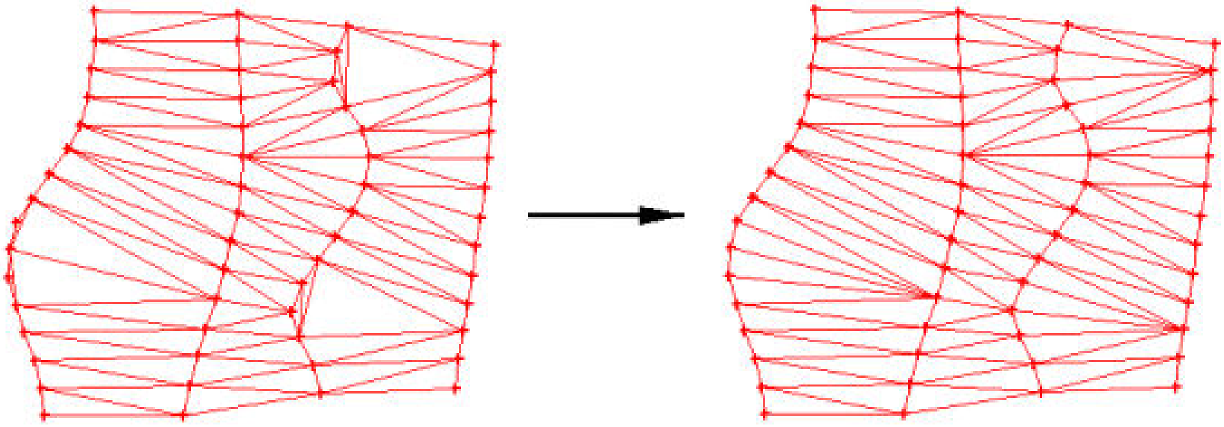
Individual Points

This section contains the retriangulate voids when deleting option. When scatter points are deleted, the triangles attached to the scatter points (if any) are also deleted. If this option is on, surrounding triangles are retriangulated to fill the void.

Triangulation Optimization Options

When the optimize triangulation command is invoked, the triangles are optimized in one of two ways:

- *Angle Optimization* – The triangles are swapped to conform to the Delaunay Criterion.
- *Area Optimization* (SHOALS) – The triangles are swapped to align with other triangles. The swapping is done by comparing the area of one triangle to its neighbor. The user sets a Bias. If the area of the smaller triangle is less than the area of the larger triangle divided by the bias, the triangles are swapped. This is useful for optimizing the triangulation of surveys such as SHOALS surveys.



Duplicate Vertex Options

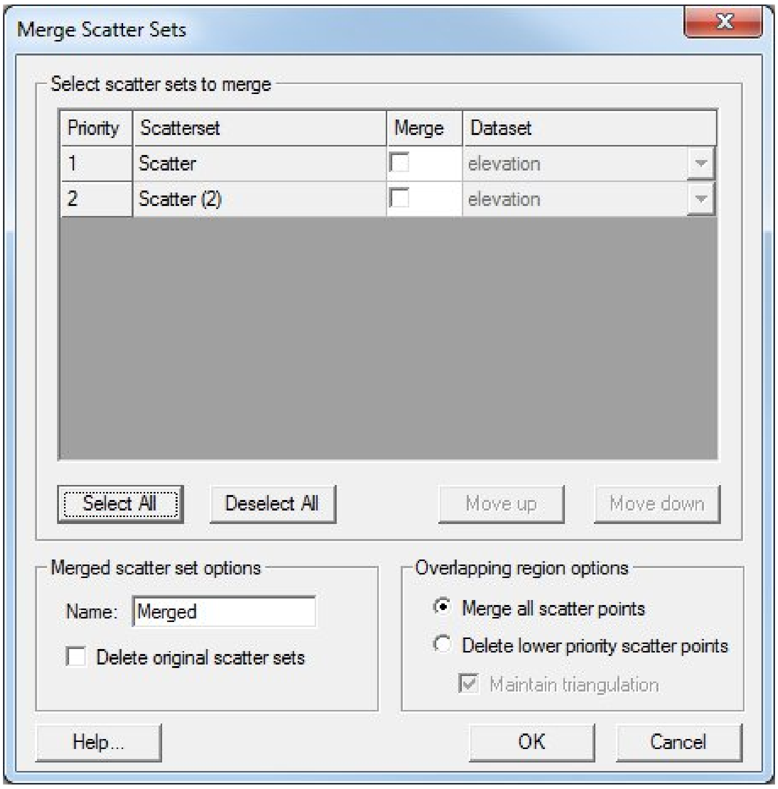
When the **Select/Delete Duplicate Points** menu item is selected, points within a tolerance of other points are selected or deleted. The user sets the Tolerance and whether to delete or select the points when the command is invoked.

Delete Scatter Set

This option is found in the Scatter module in the *Scatter* menu. If one scatter set exists, the user will be asked if they want to delete the active scatter set. If more than one scatter set exists, a dialog appears. The scatter sets can be flagged for deletion in this dialog. Double-clicking on a scatter set in the window or pushing the **Delete** button flags or unflags a scatter set for deletion. **Select All** or **Deselect All** will flag or unflag all sets. A set is flagged if the letter "d" appears to the left of the scatter name.

Merge Scatter Sets

Multiple scatter sets can be merged into a single scatter set using the *Merge Scatter Sets* dialog. The menu command *Scatter* | **Merge Sets** opens the *Merge Scatter Sets* dialog.



Scatter Module *Merge Scatter Sets* dialog

Selecting scatter sets to merge

The *Merge Scatter Sets* dialog contains a spreadsheet listing all of the scatter sets currently loaded into SMS. Scatter sets to merge are specified by checking the *Merge* box in the *Merge column* of the spreadsheet. When merging scatter sets, only one dataset is transferred to the merged scatter set. The dataset to be transferred is specified for each scatter set in the *Dataset column* of the spreadsheet.

The Priority column of the spreadsheet is only used if the Overlapping region option is set to *Delete lower priority scatter points*. This option is explained below.

Merge Options

The following options are available when merging scatter sets:

Merged scatter set options

- **Name –** Specify the name for the new, merged scatter set.
- **Delete original scatter sets –** The scatter sets to be merged are deleted after the new, merged set is created.

Overlapping region options

- **Merge all scatter points –** All scatter points from all scatter sets to be merged are combined into one set and retriangulated.
- **Delete lower priority scatter points –** In regions where scatter points and triangles overlap, the scatter points and triangles from the lower priority scatter set are deleted. The priority is based on the **Priority** column of the **Select scatter sets to merge** spreadsheet. The **Move up** and **Move down** buttons can be used to adjust the priority of the scatter sets when this option is selected.
  - **Maintain triangulation –** The triangulation of the original scatter sets is maintained. New triangles are created to connect the original scatter sets into a single, merged scatter set.

Merge Report

When the merge finishes a merge report will be displayed on the screen. This report shows statistics for the scatter sets that were merge such as number of vertices and triangles before and after the merge. If desired vertices are being deleted, check the duplicate points tolerance. This is found at Scatter→Scatter Options dialog.

Create Scatter Subset

All selected points from the original scatter set are moved from the original set into a new scatter set. The user is prompted for the name of the new scatter set. If all points for the current scatter set are selected, nothing occurs. The two scatter sets, the original and the new, are retriangulated.

Interpolate to Mesh

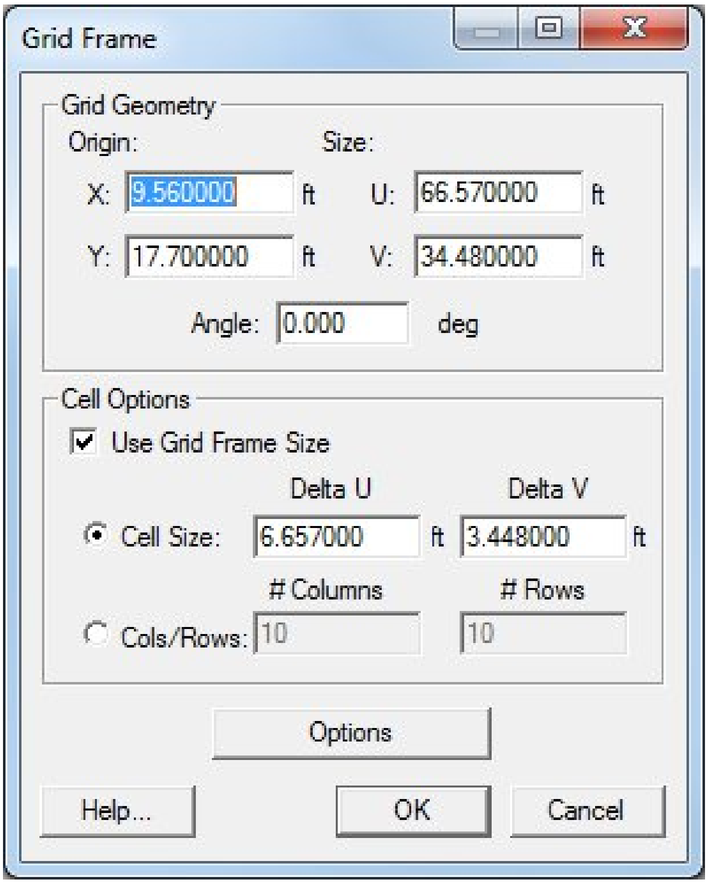
If mesh nodes exist, the *Interpolation* dialog appears where the user sets the interpolation options. The scatter point datasets are then interpolated to the mesh nodes using the user specified interpolation options.

Interpolate to Cartesian Grid

If a cartesian grid exists, the *Interpolation Dialog* appears where the user sets the interpolation options. The scatter point function values are then interpolated to the center of each grid cell.

Interpolate to Scatter Grid

If a scattered dataset exists, the **Interpolate to Scatter Grid** menu item (Scatter module, *Scatter* menu) brings up the *Grid Frame* dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. When the user pushes **OK**, a new scatter set is created with scatter points at the corners of each grid cell. The original scatter set is interpolated to the new scatter grid set using linear interpolation.



Grid Frame dialog

Using the scatter grid is a form of data decimation: a dense scatter set can be represented as a less dense scatter set.

Interpolate to Nautical Grid

This option creates a nautical chart. A nautical chart divides a scatter set into bins and finds the maximum, minimum, and average depth over each bin. The **Interpolate to Nautical Grid** menu item (Scatter module, Scatter menu) brings up the *Grid Frame* dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. When the user pushes *OK*, a new scatter set is created with scatter points at the center of each grid cell. Three functions are created for each scatter point from the active scalar function of the original scatter set:

- **Average** – The average depth over each bin.
- **Minimum** – The minimum depth over each bin.
- **Maximum** – The maximum depth over each bin.

Requirements to Interpolate to a Nautical Grid

1. A scatter dataset must exist.
2. Active coverage type must allow grid frames to be created. The Cartesian Grid Module model coverage types allow the creation of grid frames.

Interpolate from Other Scatter

This option interpolates one scattered dataset to another set. Two sets must exist for this option to be enabled. A dialog appears and the user selects the scatter set to interpolate from. That scatter set is interpolated to the active scatter set. The **Options** button brings up the *Interpolation Options* dialog, allowing the user to set the interpolation type. The interpolation uses an extrapolation value of 0.0.

- Interpolate to Map – This option allows the user the convert Scatter Breaklines to Featuer Arcs. Right-click on Scatter Module in the *Project Explorer* to convert.

Related Topics

- Scatter Module
- Scatter Module Menus
- Scatter Triangles Menu

Scatter Data Menu

Most of the SMS modules have a *Data* menu, but the items in this menu are different for each module. The menu items work with the active scatter set unless otherwise noted. The Scatter Module commands include:

Dataset Commands

- Data Calculator – Can be used to perform mathematical operations with datasets to create new datasets.
- Dataset Toolbox – Opens the *Data Toolbox* containing various tools to work with datasets.
- Create Datasets – Brings up the *Create Datasets* dialogue for creating functions for the active scatter set.
- Smooth Size Dataset – Opens the *Smooth Dataset* dialog that can be used to condition scattered data scalar values.
- Transform – Brings up the *Transform* dialog. The **Transform** command is used to move scatter points.
- Zonal Classification – Can be used to identify areas that meet a set of criteria. The criteria can be based upon scalar dataset values and/or specific material ids in a coverage. Opens the *Classification Wizard*.

Visualization Commands / Options

- **Contour Options** – Brings up the *Display Options* dialogue. See Contour Options for more information.
- **Vector Options** – Brings up the *Display Options* dialogue. See Vector Display Options for more information.
- **Film Loop** – Opens the *Film Loop Setup Wizard*. See Animation for more information.

Data Conversion Commands

- **Scatter → Mesh**

  The scatter points are converted to a mesh by this command. All functional data, scalar and vector, is copied to the mesh. The mesh nodes can be triangulated in the Mesh module using the menu command *Elements* | **Triangulate**.

- **Scatter Contour → Feature**

  When this command is invoked, the *Create Contour Arcs* dialog opens. For more information, see Create Contour Arcs.

- **Boundary → Feature**

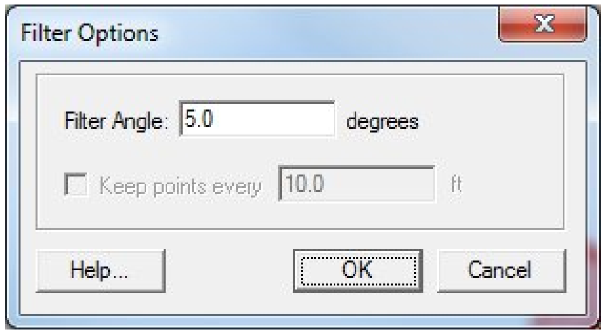
  The outer boundary of the scatter set is converted to Map module arcs. Arcs are created where scatter triangles do not border another triangle.

Scatter Commands

- **Find**

  This menu item allows you to find a specific scatter point either by specifying its ID or by specifying the nearest (x,y) coordinate. This can be useful if one specific scatter point needs to be located in a large scatter set.

- **Scatter Filter**



Filter Options in the Scatter *Data* menu

There are two filter options. The first option is to filter by adjacent triangle normal angle filter. The second option is to filter by using the VTK Decimate Pro.

You can access the Filter Options dialog by going to *Data* | **Filter...** while in the Scatter Module.

- **Filter – Adjacent Triangle Normal Angle Filter**

  Redundant and overlapping data may exist in a scatter. SMS offers the ability to filter the data and remove unnecessary data points in relatively flat areas in the Filter option from the Scatter module, *Data* menu. The user specifies an angle. Each data point is checked to see if it is in a flat region by dotting the normals of the surrounding triangles.

  If the normals are all within the specified angle, the region is flat and the point is deleted.

- **Filter – VTK Decimate Pro**

  VTK Decimate Pro is a filter to reduce the number of triangles in a triangle mesh, forming a good approximation to the original geometry. To get a detailed description of the options to set when performing this kind of filter, please visit the website:

  http://www.vtk.org/doc/release/4.0/html/classvtkDecimatePro.html [1]

- **Select/Delete Duplicate Points**

  This menu item changes according to the option set in the *Scatter Options* dialog. In the options, the user can opt to select or delete duplicate points and set a tolerance. The check works by checking each point and selecting/deleting any point that is within the tolerance. Points with lower ids are checked first; the point with the higher id is therefore selected/deleted.

Related Topics

- Scatter Module
- Scatter Module Menus
- Create Coastline
- Coastline Files

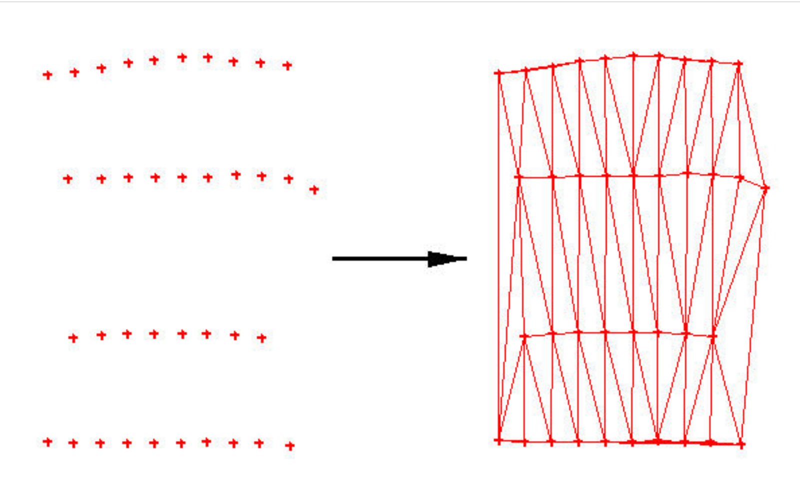
Scatter Triangles Menu

The items unique to the Scatter module are listed below. The menu items operate on the active scatter set unless otherwise noted:

General Commands

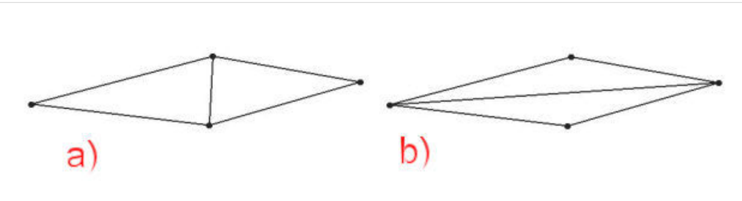
Triangulate

Scatter points or mesh nodes can be triangulated to form piecewise linear surfaces. For scattered data, these surfaces are also referred to as TINs (Triangular Irregular Networks). For mesh nodes, they form a finite element mesh. The points/nodes are connected into surfaces as scatter sets or meshes are created, but at times it may be necessary to reconnect the points (i.e. after deleting individual points/nodes or triangles/elements). New triangles are constructed in mass by triangulating a set of points when the **Triangulate** command from the *Triangles* menu is executed. The selected points are connected with a series of triangles. If points are not selected, then all points will be triangulated.



Delaunay Criterion

The resulting triangulation satisfies the Delauney criterion. The Delauney criterion ensures that no vertex lies within the interior of any of the circumcircles of the triangles in the network as shown below:

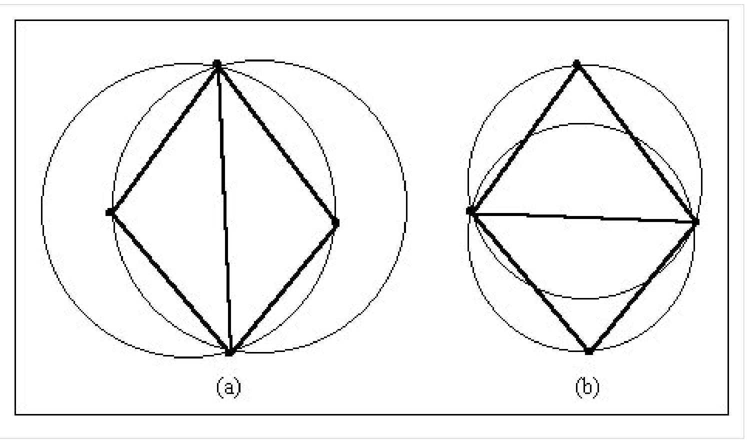


Two Adjacent Triangles Which (a) Violate and (b) Honor the Delauney Criterion.

The result of enforcing the Delauney criterion is that long thin triangles are avoided as much as possible.

Triangulate

The vertices associated with the active scatter set can be triangulated using the **Triangulate** command from the *Triangles* menu in the Scatter module. Mesh nodes (either the selected nodes, or all nodes) can be triangulated using the **Triangulate** command from the *Elements* menu in the 2D Mesh module.



Optimize Triangulation

At times, the user will perform manual mesh editing using the **Swap Edge** tool. This makes the Delauney criterion no longer hold. Selected elements can be returned to the Delauney state by choosing the **Optimize Triangulation** command from the *Elements* menu.

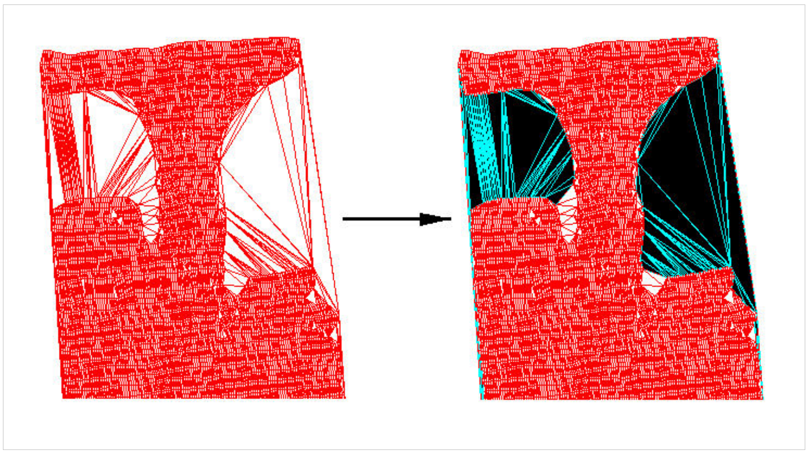
Select Thin Triangles

During the process of triangulation, a mesh of triangular elements is created around existing nodes. This usually creates triangular elements outside the desired mesh boundary. Many of these exterior triangles are very skinny, and some are virtually invisible. The **Select Thin Triangles** command from the *Elements* Menu finds and selects skinny triangular elements which are on the mesh boundary.

Thin triangles interior to the mesh will not be selected when this command is performed, since deletion of interior triangles would result in gaps in the mesh. After the thin triangles have been selected, they can be removed by selecting the Delete macro .

Select/Delete Long Triangles

This option in the Scatter Module, *Triangles* Menu finds triangles longer than the length specified in the *Scatter Options* dialog. The *Scatter Options* also allows the user to select the option to delete or select the long triangles. Selecting/deleting long triangles is useful for deleting triangles that span regions where interpolation is not desired, such as over regions of land (see figure below, the selected triangles are over land).



Process Boundary Triangles

When scatter points are [triangulated, the resulting convex hull [1] often contains triangles outside the desired mesh boundary. The *Process Boundary Triangles* dialog was developed to help remove invalid boundary triangles. To open the *Process Boundary Triangles* dialog:

- In the Scatter Module, make the **Select Triangle** tool active
- Select **Process Boundary Triangles...** from the mouse right-click menu

OR

- In the Scatter Module, select **Process Boundary Triangles...** from the *Triangles* menu

See the article Process Boundary Triangles for more information.

Related Topics

- Scatter Module
- Scatter Module Menus

Scatter Vertices Menu

The menu items operate on the active scatter set unless otherwise noted:

- **Locked –** The points in a scatter set can be dragged with the mouse cursor if they are unlocked and the **Select Scatter Points** tool is selected. The Locked item in the *Vertices* menu toggles on and off the locked status. If scatter points are locked, a check mark is shown next to the menu text. The default status is locked so scatter points are not accidentally moved.

Related Topics

- Scatter Module
- Scatter Module Menus

3.11.a.3. Project Explorer Items

Scatter Project Explorer Items

In the *Project Explorer*, the Scatter Data folder houses all of scatter data information that is manipulated by the Scatter module. The Scatter Data folder will not appear in the *Project Explorer* until scatter data is opened in SMS. Once scatter data is opened, the Scatter Data folder will appear with the Scatter set item below it. There will also be elevation information that appears under the Scatter set item. The Scatter module may be activated by clicking on the Scatter set item in the *project explorer*. Once active, the *Scatter* module tool bar menu will appear to the right of the *Project Explorer*.

Related Topics

- Scatter Module Right-Click Menus

Scatter Module Right-Click Menus

The following Project Explorer right-click menus are available when the mouse right-click is performed on a Scatter Module item.

Scatter Module Root Folder Right-Click Menus

Right-clicking on the Scatter module root folder in the project explorer invokes an options menu with the following options:

- **New Scatter Set –** Creates a new, empty scatter set.
- Display Options

Scatter Set Item Right-Click Menus

Right-clicking on a Scatter Set item in the Project Explorer invokes an options menu with the following module specific options:

- **Split –** Creates a new scatter set containing the selected scatter vertices. Selected scatter vertices are removed from the original scatter set.
- **Autogenerate breaklines –** Automatically creates breaklines following specified elevations.

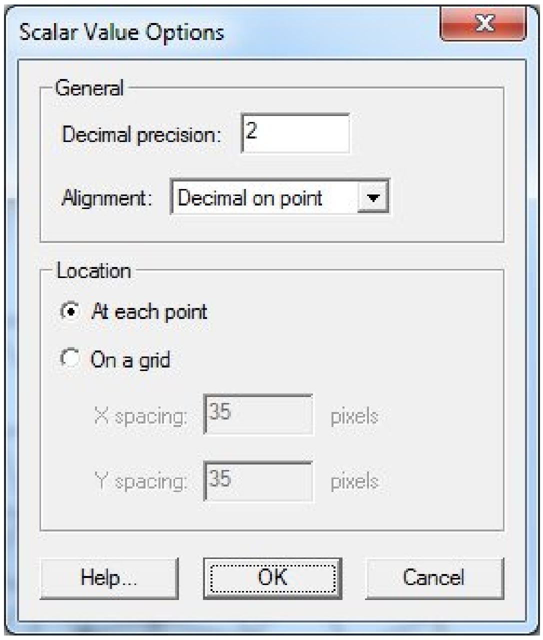
Related Topics

- Project Explorer Right-Click Menus
- Menu Bar

3.11.b. Functionalities

Scalar Value Options

The *Scalar Value Options* dialog controls the display options for the scalar values displayed next to each scatter point. The option to display the scalar value of the active scatter function next to each scatter point is turned on and off in the Scatter Module *Display Options* dialog.



Scalar Value Options dialog

The following options are available in the *Scalar Value Options* dialog:

- **Decimal Precision –** The decimal precision used.
- **Alignment –** The location used. The following options are available:
  - On Point
  - Left
  - Center
  - Right
- **Text Options –** Change the color, size, and font for the text.
- **Location Options –** Display the text at:
  - Each point
  - On a grid (works well for dense data). The user specifies the x and y pixel spacing. Because the spacing is based on pixels, the spacing stays constant during zooming.

Related Topics

- Scatter Module Display Options
- Display Options
- Scatter Module

Scatter Datasets

The Scatter Point Module is used to visualize and apply various types of data. This data typically comes from surveys, digital maps, previous numerical analysis or digitization on screen. The data is stored as sets or groups of 2D scattered data points with associated values. The most common value is bathymetry and is used to create the geometric representation of the area being modeled.

SMS connects the scattered data points into triangles forming a Triangulated Irregular Network (TIN). TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for a source of bathymetric or other data in a numerical model. TINs can also be used to compute areas, volume, distances, gradients and several other geometric parameters.

SMS applies data from scattered datasets to finite element networks or grids via interpolation. This allows poorly distributed elevation data to be assigned to a well-structured set of elements to create the bathymetry of the entire mesh. A variety of interpolation schemes are supported. Internally, the scattered datasets are triangulated to create surfaces for continuous interpolation. Since the connectivity of the triangulation affects the interpolation, SMS provides tools to allow for the manipulation of this triangulation. The triangulation also allows contouring of the scattered dataset to visualize the data.

Multiple scatter point sets can exist at one time in memory. One of the scatter sets is always designated as the "active" scatter point set. The active scatter set can be changed by changing the *Scatter Set* combo box in the top *Edit Window*. Whenever a new scatter set is created, it becomes the active set.

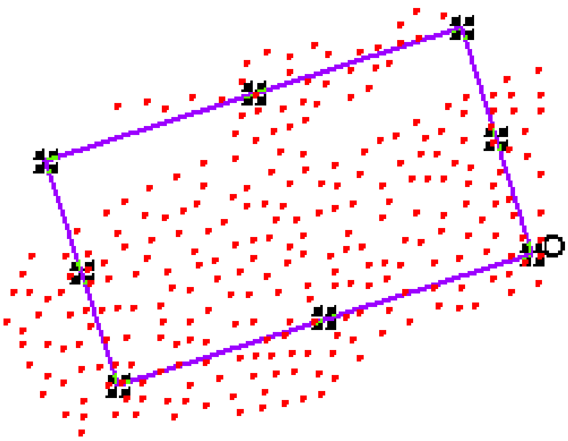
Related Topics

- Scatter Module

Grid Frame Dialog

Grid Frames are created from:

- *Feature Objects* | **Grid Frame** in the Map module for Cartesian Grid coverages
- *Scatter* | **Interp to Scatter Grid** in the Scatter module
- *Scatter* | **Interp to Nautical Grid** in the Scatter module

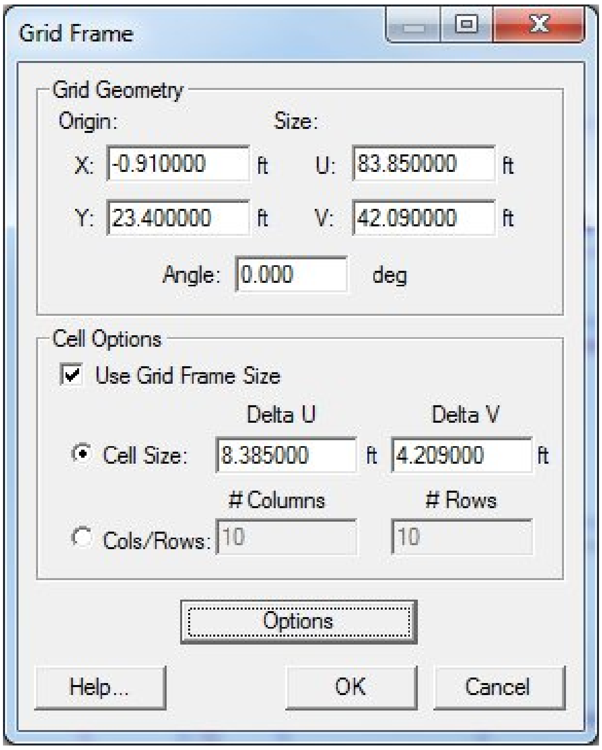


The grid frame is interactive with the *Graphics Window* and the *Grid Frame* dialog. Interactive options include:

- **Move Frame** – Click inside or on an edge where the frame is not highlighted and drag.
- **Resize Frame** – Click a highlighted corner or edge and drag to resize.
- **Rotate Frame** – Click inside the circle near the bottom right corner of the frame and drag to rotate the frame.
-  – Redraw the screen.
-  – Zoom to the extents of the data in the screen.

Dialog options include:

- **Grid Origin** – Set the bottom left corner location of the grid frame.
- **Angle of rotation** – Set the angle the grid frame will be rotated counter-clockwise from the +x axis.
- **Snap to grid** – After turning this option on, push the button to see grid snap points overlay the screen. The Snap Spacing controls the spacing of the grid snap points. As the user drags and resizes the grid frame, the grid frame edges snap to the grid snap points.
- **New/Delete Grid** – Available for Cartesian Grid frame only. Create or delete the current grid frame. For the other Scatter interpolation applications, a temporary grid frame is created for the specific application and then deleted.
- **Grid Dimensions** – Set the length and width of the grid frame. The Cell Dimensions are updated: Cell X = Grid X / Number of Columns, Cell Y = Grid Y / Number of Rows.



Grid Frame dialog using **Interp to Scatter Grid** command

- **Cell Dimensions** – Set the length and width of each cell for the grid to be created.. The Grid Dimensions are updated: Grid X = Cell X * Number of Columns, Grid Y = Cell Y * Number of Rows.
- **Number of Columns/Rows** – Set the number of rows and columns for the grid to be created.
- **Options** – Set the interpolation options for the Scatter applications. Not used for the Cartesian Grid application.

Related Topics

- Feature Objects Menu
- Grid Frame Properties

Use of DEMs in the Scatter Module

The Scatter Point Module is used to visualize and apply various types of data. This data typically comes from surveys, digital maps, previous numerical analysis or digitization on screen. The data is stored as sets or groups of 2D scattered data points with associated values. The most common value is bathymetry and is used to create the geometric representation of the area being modeled.

SMS connects the scattered data points into triangles forming a Triangulated Irregular Network (TIN). TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for a source of bathymetric or other data in a numerical model. TINs can also be used to compute areas, volume, distances, gradients and several other geometric parameters.

SMS applies data from scattered datasets to finite element networks or grids via interpolation. This allows poorly distributed elevation data to be assigned to a well-structured set of elements to create the bathymetry of the entire mesh. A variety of interpolation schemes are supported. Internally, the scattered datasets are triangulated to create surfaces for continuous interpolation. Since the connectivity of the triangulation affects the interpolation, SMS provides tools to allow for the manipulation of this triangulation. The triangulation also allows contouring of the scattered dataset to visualize the data.

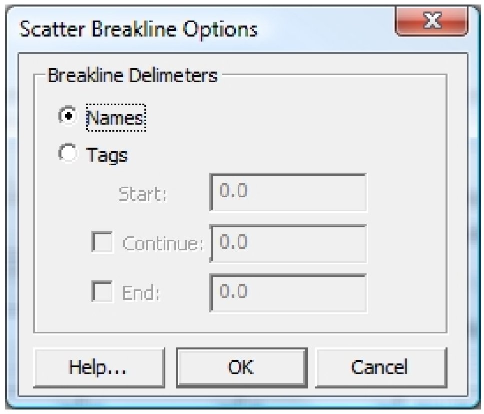
Multiple scatter point sets can exist at one time in memory. One of the scatter sets is always designated as the "active" scatter point set. The active scatter set can be changed by changing the *Scatter Set* combo box in the top *Edit Window*. Whenever a new scatter set is created, it becomes the active set.

Related Topics

- Data Acquisition
- TINs

Scatter Breakline Options

When importing a Scatter Set using the File Import Wizard, selecting "Breaklines" as a Field will open the *Scatter Breakline Options* dialog. Breaklines are useful for maintaining the correct triangulation in a TIN.



Scatter Breakline Options

Breakline Delimeters

The available breakline delimeter options and samples of the associated file formats are:

- **Names –** Breaklines are identified using a unique name or ID
- **Tags –** Breaklines are identified using "start" and optionally "continue" and "end" values

No vertex can be in more than one breakline, and the breakline must be defined sequentially in the data file.

Named Breaklines

The breakline column of the data file includes the name of the breakline this vertex belongs to. If the breakline column is empty, the associated vertex is not included in any breaklines.

Example Files

Example of a tab delimited file using breakline names:

```
xcoord      ycoord      zcoord   name
215962.9    85203.098   1.483    Breakline1
215957.638  85193.069   1.483    Breakline1
215963.278  85184.35    1.483    Breakline1
215979.111  85179.328   1.483    Breakline1
216056.51   85209.371   1.483    Breakline1
215992.462  85201.477   7.034    Breakline2
216127.386  85264.681   7.034    Breakline2
216267.187  85327.936   7.034    Breakline2
216371.217  85381.431   7.034    Breakline2
219261.939  90247.944   8.763
219461.211  90220.556   9.167
219678.994  90179.064   9.468
```

Tagged Breaklines

The breakline column of the data file includes a tag or string defining when a breakline starts and stops. There are three types of tags including:

- **Start** – Identifies the start of a new breakline
- **Continue** – Indicates the vertex should be included in the current breakline
- **End** – Identifies the end of the current breakline

These tags may be used in a number of configurations:

- **Start**, **Continue**, **End** – When all three tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter should have a **Continue** tag until the line with the **End** tag. Lines with no entry in the Breakline column between breaklines are read as vertices not belonging to any breakline.
- **Start**, **End** – When the start and end tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter is searched for an **End** tag. All intervening lines are assumed to belong sequentially to a breakline. If two **Start** tags are encountered without an intervening **End** tag, the break line is terminated and another started.
- **Start**, **Continue** – When the start and continue tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter is searched for a **Continue** tag. If any line is encountered without a **Continue** tag, the breakline is terminated and the vertex associated with that line is not included in a breakline.

Example of a tab delimited file using the following breakline tags:

- Start: 1
- Continue: 2
- End: 4
- Not in breakline: 5

```
xcoord        ycoord       zcoord   breakline_tag
215962.9      85203.098    1.483    1
215957.638    85193.069    1.483    2
215963.278    85184.35     1.483    2
215979.111    85179.328    1.483    2
216056.51     85209.371    1.483    4
215992.462    85201.477    7.034    1
216127.386    85264.681    7.034    2
216267.187    85327.936    7.034    2
216371.217    85381.431    7.034    4
219261.939    90247.944    8.763    5
219461.211    90220.556    9.167    5
219678.994    90179.064    9.468    5
```

Related Topics

- File Import Wizard
- Breaklines

Scatter Breakline Menu

The menu items operate on the active scatter set unless otherwise noted:

- **Force Breaklines –** force scatter triangle edges to follow the selected breaklines by swapping triangle edges. If no breaklines are selected, all breaklines will be forced.
- **Merge –** merge selected breaklines to form a single breakline. Available if more than one breakline is selected.
- **Split –** split a single breakline into multiple breaklines at the selected scatter vertex. Available if a scatter vertex is selected that is connected to a breakline.

Related Topics

- Scatter Module
- Scatter Module Menus
- Importing Scatter Breaklines

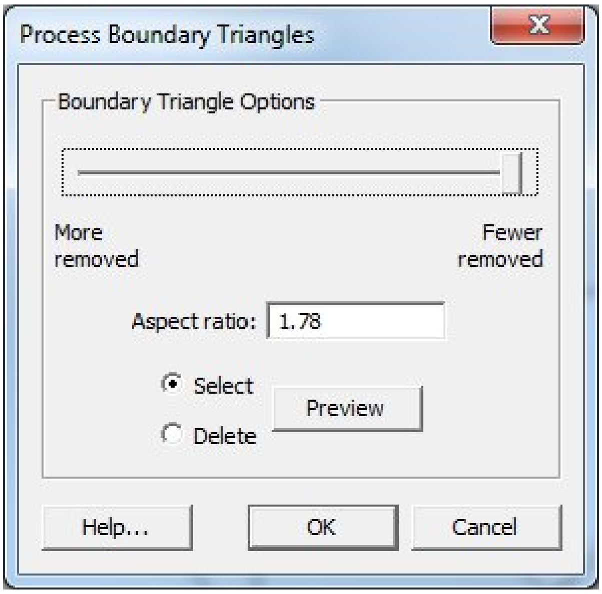
Process Boundary Triangles

When scatter points are triangulated, the resulting convex hull [1] often contains triangles outside the desired mesh boundary. The *Process Boundary Triangles* dialog was developed to help remove invalid boundary triangles. To open the *Process Boundary Triangles* dialog:

- In the Scatter Module, make the **Select Triangle** tool active
- Select **Process Boundary Triangles...** from the mouse right-click menu

OR

- In the Scatter Module, select **Process Boundary Triangles...** from the *Triangles* menu



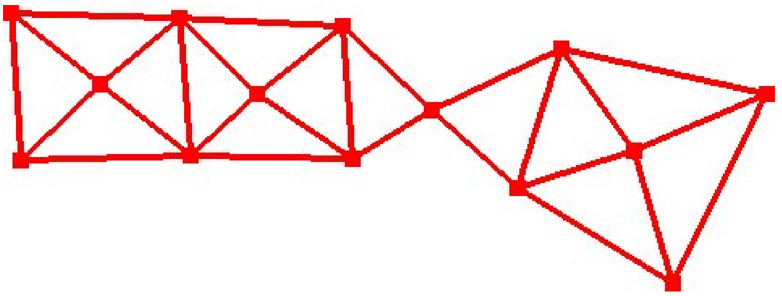
Process Boundary Triangles dialog

Removal Options

- **Aspect Ratio –** A small aspect ratio (sometimes called edge ratio) will result in the removal of more triangles than a large edge ratio. Specifying too small of an aspect ratio will result in the removal of valid triangles. The aspect ratio is calculated by dividing the length of the triangle edge on the boundary by the length of the smallest triangle edge connected to the scatter vertex end points of the triangle edge on the boundary.
- **Select –** Triangles meeting the specified aspect ratio are selected when **OK** is pressed
- **Delete –** Triangles meeting the specified aspect ratio are deleted when **OK** is pressed
- **Preview –** Preview which triangles will be processed based on the specified aspect ratio

Using the Tool / Practical Notes

- Areas on the interior of the triangulation, such as islands, should be manually "seeded" by deleting one of the unneeded triangles in the interior region.
- Triangles will not be removed if their removal would result in:
  - the removal of a scatter vertex from the triangulation
  - the creation of a "bow tie" in the scatter set. A bow tie, a point in the scatter set where the scatter set is "pinched" to a single point, is shown in the image below.



"Bow tie" in a scatter set.

Related Topics

- Select Thin Triangles

Interpolate to Scatter Set

This option interpolates one scattered dataset to another set. Two sets must exist for this option to be enabled. A dialog appears and the user selects the scatter set to interpolate from. That scatter set is interpolated to the active scatter set. The **Options** button brings up the *Interpolation Options* dialog, allowing the user to set the interpolation type. The interpolation uses an extrapolation value of 0.0.

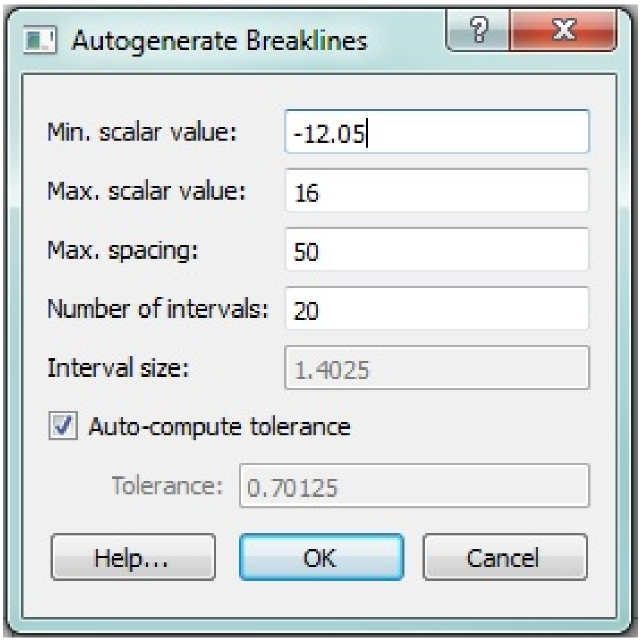
Related Topics

- Grid Frame Dialog
- Scatter Menu
- Scatter Data Menu
- Scatter Triangles Menu

Generate Contour Breaklines

Breaklines in scatter sets can greatly improve the representation of a physical surface. The **Generate Contour Breaklines** tool is used to create breaklines following specified dataset values. This option was designed to work on scatter data that has been digitized to follow features.

The following dialog is brought up by right-clicking on the scatter set and selecting **Generate Contour Breaklines...**



The options in the *Autogenerate Breaklines* dialog are as follows:

- Min. Scalar Value – The minimum of the range to be used for autogenerating breaklines (defaulted to the minimum scalar value of the scatter data)
- Max. Scalar Value – The maximum of the range to be used for autogenerating breaklines (defaulted to the maximum scalar value of the scatter data)
- Max. Spacing – The maximum distance allowed between two adjacent vertices in a breakline
- Number of Intervals – The number of dataset intervals from which the breaklines will be created
- Interval Size – The size of each interval (this is equal to the entire range; defined by the min. and max. scalar values; divided by the number of intervals)
- Auto-compute Tolerance – SMS will automatically compute the tolerance such that all datasets falling within an interval will be included in the breakline generation. The auto-computed tolerance is equal to half of the interval size. This value can be overridden.

SMS creates the breaklines for each contour interval by gathering all vertices in the scatterset that are within the specified tolerance. Starting with the first vertex, SMS will search for the next closest vertex in the scatterset that is also within the specified tolerance. If the closest vertex is within the specified maximum spacing, it becomes the next vertex in the breakline. SMS continues adding vertices to the breakline until it cannot find a vertex within the spacing limit. At this point, SMS will end the breakline and begin to create another breakline using the same method.

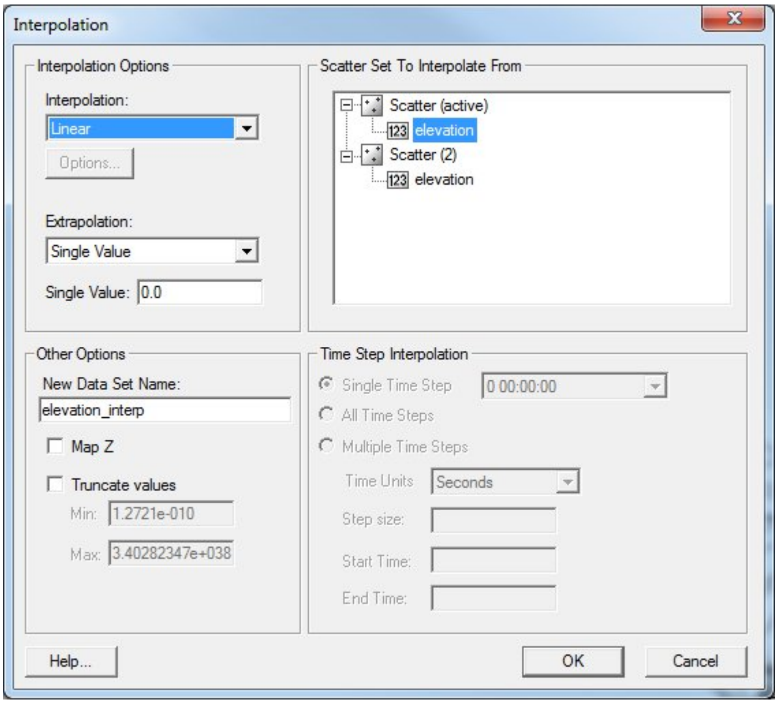
3.11.c. Scatter Interpolation

Scatter Interpolation

Interpolation can be invoked explicitly or implicitly. The implicit invocation is part of the automatic mesh or grid generation. Explicit interpolation occurs when the user selects an **Interpolate to _____...** option in the *Scatter* menu in the Scatter Module. These commands require that at least one scatter set exist with at least one function associated. A mesh or grid must also exist in order for the associated interpolation command to be available.

Interpolation Dialog

When the user selects an interpolation command, the *Interpolation Option* dialog appears labeled to indicate what interpolation is being performed. The user selects the appropriate options and once the **OK** button is selected, the interpolation procedure is performed. The user specified options include:



Interpolation dialog

- **New interpolated dataset name –** The name of the new dataset (or function) created by interpolation.
- **Map Elevation –** For interpolation to mesh nodes, the new dataset is mapped to be the elevation function.
- **Interpolation Method –** Since no interpolation scheme is superior in all cases, SMS supports three interpolation techniques. Many other methods are possible, however, since surface water modeling requires a fairly rich dataset, the more simplistic interpolation methods are more applicable. The user selects a current method that is used for all interpolation until the user selects another method. The supported methods include:
    1. Linear
    2. Inverse Distance Weighted (IDW)
    3. Natural Neighbor (NN)
    4. Laplacian Interpolation (Cartesian grids only)
- **Default Extrapolation Value –** If the scatter set does not bound the data being interpolated to, the extrapolation value is used (for Linear and NN interpolation only).
- **Existing Dataset Value –** The corresponding value from a specified existing dataset can be used for locations outside of the bounds of the scatter set. The dataset must be from the same object being interpolated to and must be of the same type (i.e. scalar, vector).

- **Truncate Values –** When interpolating a set of values, it is sometimes useful to limit the interpolated values to a specific range. For example, when interpolating contaminant concentrations, a negative value of concentration is meaningless. However, many interpolation schemes will produce negative values even if all of the scatter points have positive data values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by entering a minimum and maximum interpolation value.
- **Scatter Set/Function –** The user selects the desired scatter point set and the function to interpolate from.
- **Time Steps –** The user selects the option to interpolate a single time step or multiple time steps (if the function is transient).
- **Time Step Interpolation –** If interpolating multiple time steps, the user can select the beginning time, the step size, and the number of time steps to interpolate. The user can also choose to interpolate between time steps or to match time steps that fall within the specified time range.

Related Topics

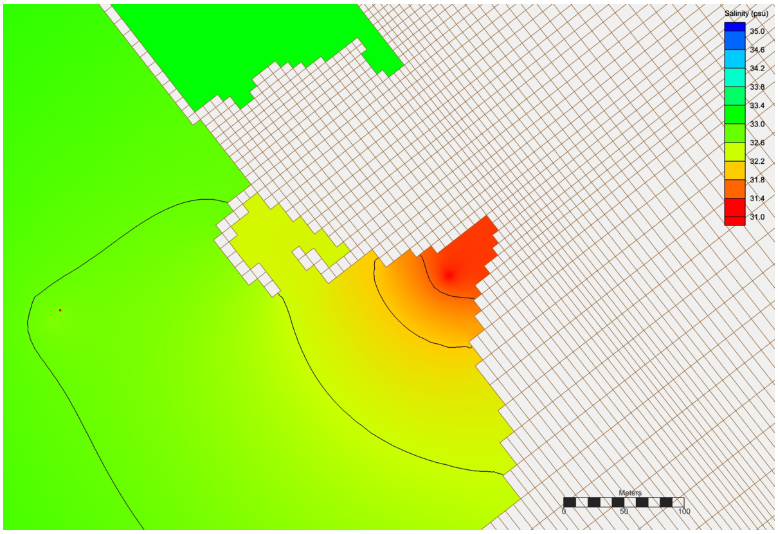
- Scatter Module
- Scatter Menu

Laplacian Interpolation

Laplacian interpolation is an interpolation/extrapolation approach that tries to give smooth gradients between areas of known values. You can only perform Laplacian interpolation on a cell-centered Cartesian grid.

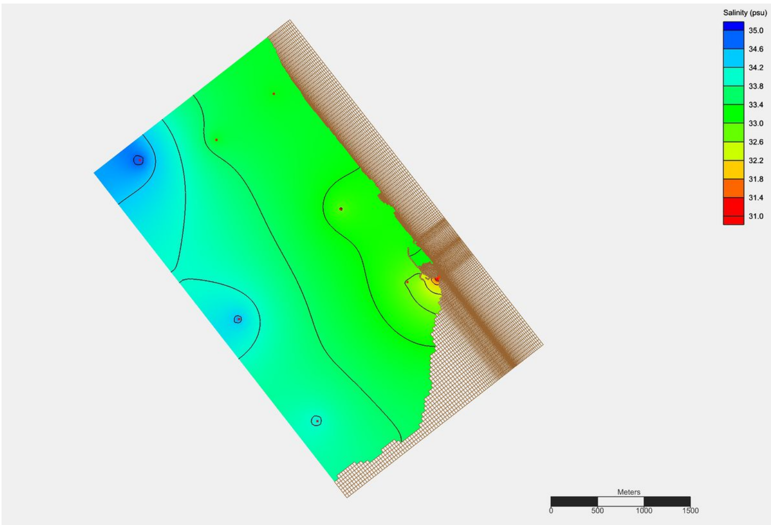
When you perform Laplacian interpolation, the values at each scatter vertex are considered known values and assigned to the Cartesian grid cells that they lie within. We will refer to the known locations/values as seed location/values. The seed values are held constant throughout the process. The algorithm assigns the average seed value to every cell in the grid. Once the initial values have been set, the algorithm repeatedly sweeps the grid smoothing the data values. The result gives the seed values at each of their locations and smooth transitions between the values. The algorithm will also extrapolate outside of the original seed values.

For models that distinguish between wet and dry cells (like CMS-Flow), the Laplacian interpolation will only be performed inside of wet (computational) cells. This gives the often desirable effect that the gradients will follow flow-paths. Cells that are nearby but do not have a water connection, will have a greater difference in value than cells that have a water connection. This effect can be seen in the image below.



Laplacian interpolation for salinity values

The picture below is of the same domain but zoomed out so you can see the full domain. Notice the smooth transitions between distinct values.



Laplacian interpolation for salinity full domain

Inverse Distance Weighted Interpolation

One of the most commonly used techniques for interpolation of scatter points is inverse distance weighted (IDW) interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced most by the nearby points and less by the more distant points. The interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases. Several options are available for inverse distance weighted interpolation. The options are selected using the *Inverse Distance Weighted Interpolation Options* dialog. This dialog is accessed through the **Options** button next to the Inverse distance weighted item in the *2D Interpolation Options* dialog. SMS uses Shepard's Method for IDW:

Shepard's Method

The simplest form of inverse distance weighted interpolation is sometimes called "Shepard's method" (Shepard 1968). The equation used is as follows:

$$F(x,y) = \sum_{i=2}^{n} w_i f_i$$

where $n$is the number of scatter points in the set, $f_i$are the prescribed function values at the scatter points (e.g. the dataset values), and $w_i$are the weight functions assigned to each scatter point. The classical form of the weight function is:

$$w_i = \frac{h_i^{-p}}{\sum_{j=i}^{n} h_j^{-p}}$$

where $p$is an arbitrary positive real number called the power parameter (typically, $p=2$) and hi is the distance from the scatter point to the interpolation point or

$$h_i = \sqrt{x - x_i)^2 + (y - y_i)^2}$$

where $(x,y)$are the coordinates of the interpolation point and <math<(x_i,y_i)</math> are the coordinates of each scatter point. The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases. The weight functions are normalized so that the weights sum to unity.

The effect of the weight function is that the surface interpolates each scatter point and is influenced most strongly between scatter points by the points closest to the point being interpolated.

Although the weight function shown above is the classical form of the weight function in inverse distance weighted interpolation, the following equation is used in SMS:

$$w_i = \frac{\left[\frac{R-h_i}{Rh_i}\right]^2}{\sum_{j=i}^{n}\left[\frac{R-h_j}{Rh_j}\right]^2}$$

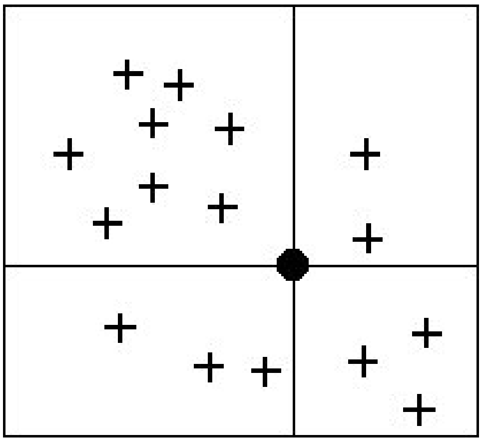
where $h_i$is the distance from the interpolation point to scatter point i, $R$is the distance from the interpolation point to the most distant scatter point, and $n$is the total number of scatter points. This equation has been found to give superior results to the classical equation (Franke & Nielson, 1980).

The weight function is a function of Euclidean distance and is radially symmetric about each scatter point. As a result, the interpolating surface is somewhat symmetric about each point and tends toward the mean value of the scatter points between the scatter points. Shepard's method has been used extensively because of its simplicity.

Computation of Nodal Function Coefficients

In the *IDW Interpolation Options*' dialog, an option is available for using a subset of the scatter points (as opposed to all of the available scatter points) in the computation of the nodal function coefficients and in the computation of the interpolation weights. Using a subset of the scatter points drops distant points from consideration since they are unlikely to have a large influence on the nodal function or on the interpolation weights. In addition, using a subset can speed up the computations since less points are involved.

If the *Use subset of points* option is chosen, the **Subsets** button can be used to bring up the *Subset Definition* dialog. Two options are available for defining which points are included in the subset. In one case, only the nearest N points are used. In the other case, only the nearest N points in each quadrant are used as shown below. This approach may give better results if the scatter points tend to be clustered.



The Four Quadrants Surrounding an Interpolation Point.

If a subset of the scatter point set is being used for interpolation, a scheme must be used to find the nearest N points. Two methods for finding a subset are provided in the Subset Definition dialog: the global method and the local method.

Global Method

With the global method, each of the scatter points in the set are searched for each interpolation point to determine which N points are nearest the interpolation point. This technique is fast for small scatter point sets but may be slow for large sets.

Local Method

With the local methods, the scatter points are triangulated to form a temporary TIN before the interpolation process begins. To compute the nearest N points, the triangle containing the interpolation point is found and the triangle topology is then used to sweep out from the interpolation point in a systematic fashion until the N nearest points are found. The local scheme is typically much faster than the global scheme for large scatter point sets.

Computation of Interpolation Weights

When computing the interpolation weights, three options are available for determining which points are included in the subset of points used to compute the weights and perform the interpolation: subset, all points, and enclosing triangle.

Subset of Points

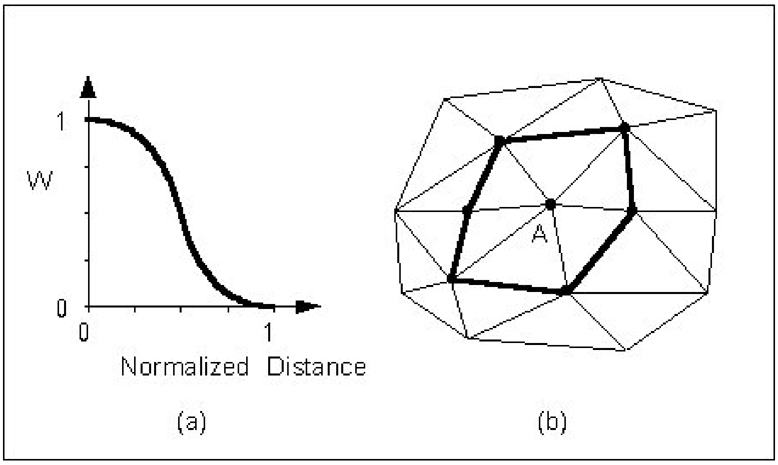
If the Use subset of points option is chosen, the Subset Definition dialog can be used to define a local subset of points.

All Points

If the Use all points option is chosen, a weight is computed for each point and all points are used in the interpolation.

Enclosing Triangle

The Use vertices of enclosing triangle method makes the interpolation process a local scheme by taking advantage of TIN topology (Franke & Nielson, 1980). With this technique, the subset of points used for interpolation consists of the three vertices of the triangle containing the interpolation point. The weight function or blending function assigned to each scatter point is a cubic S-shaped function as shown in part a of the figure below. The fact that the slope of the weight function tends to unity at its limits ensures that the slope of the interpolating surface is continuous across triangle boundaries.

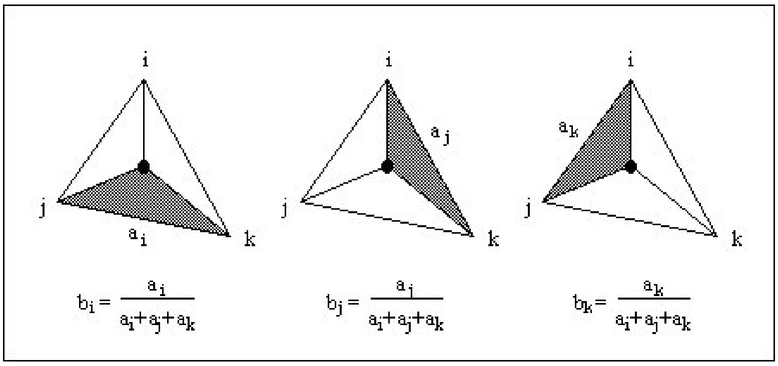


(a) S-Shaped Weight Function and (b) Delauney Point Group for Point A.

The influence of the weight function extends over the limits of the Delauney point group of the scatter point. The Delauney point group is the "natural neighbors" of the scatter point, and the perimeter of the group is made up of the outer edges of the triangles that are connected to the scatter point as shown in part b. The weight function varies from a weight of unity at the scatter point to zero at the perimeter of the group. For every interpolation point in the interior of a triangle there are three nonzero weight functions (the weight functions of the three vertices of the triangle). For a triangle T with vertices i, j, & k, the weights for each vertex are determined as follows:

$$w_i(x,y) = b_i^2(3-2b_i) + 3\frac{b_i^2 b_j b_k}{b_i b_j + b_i b_k + b_j b_k}$$
$$\left\{ b_j \left[\frac{|e_i|^2 + |e_k|^2 - |e_j|^2}{|e_k|^2} \right] + b_k \left[\frac{|e_i|^2 + |e_j|^2 - |e_k|^2}{|e_j|^2} \right] \right\}$$

where ‖ei‖ is the length of the edge opposite vertex i, and b_i, b_j, $b_k$are the area coordinates of the point (x, y) with respect to triangle T. Area coordinates are coordinates that describe the position of a point within the interior of a triangle relative to the vertices of the triangle. The coordinates are based solely on the geometry of the triangle. Area coordinates are sometimes called "barycentric coordinates." The relative magnitude of the coordinates corresponds to area ratios as shown below:



Barycentric Coordinates for a Point in a Triangle.

The XY coordinates of the interior point can be written in terms of the XY coordinates of the vertices using the area coordinates as follows:

$$x = b_i x_i + b_j x_j + b_k x_k$$
$$y = b_i y_i + b_j y_j + b_k y_k$$
$$1.0 = b_i + b_j + b_k$$

Solving the above equations for b_i, b_j , and $b_k$yields:

$$b_i \frac{1}{2A}[(x_j y_k - x_k y_j) + (y_j - y_k)x + (x_k - x_j)y]$$

$$b_j \frac{1}{2A}[(x_k y_i - x_i y_k) + (y_k - y_i)x + (x_i - x_k)y]$$
$$b_k \frac{1}{2A}[(x_i y_j - x_j y_i) + (y_i - y_j)x + (x_j - x_i)y]$$
$$A = \frac{1}{2}(x_i y_j + x_j y_k + x_k y_i - y_i x_j - y_j x_k - y_k x_i)$$

Using the weight functions defined above, the interpolating surface at points inside a triangle is computed as:

$$F(x,y) = w_i(x,y)Q_i(x,y) + w_j(x,y)Q_j(x,y) + w_k(x,y)Q_k(x,y)$$

where w_i, w_j, and $w_k$are the weight functions and Q_i, Q_j, and $Q_k$are the nodal functions for the three vertices of the triangle.

Related Topics

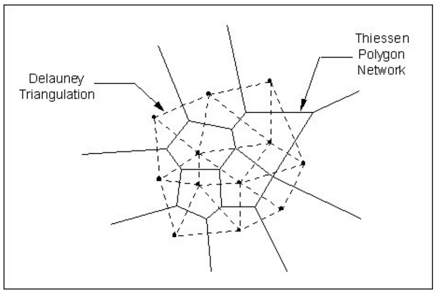
Scatter Interpolation

Natural Neighbor Interpolation

Natural neighbor interpolation is also supported in SMS. Natural neighbor interpolation has many positive features. It can be used for both interpolation and extrapolation and it generally works well with clustered scatter points. Natural neighbor interpolation was first introduced by Sibson (1981). A more detailed description of natural neighbor interpolation in multiple dimensions can be found in Owen (1992).

The basic equation used in natural neighbor interpolation is identical to the one used in IDW interpolation:

$$\lambda_m(n) = \frac{K_m(n)}{K(n)}$$



Delauney Triangulation and Corresponding Thiessen Polygon Network for a Set of Scatter Points.

SMS uses nodal functions with IDW. The nodal function can be selected using the Natural Neighbor Interpolation Options dialog. The difference between IDW interpolation and natural neighbor interpolation is the method used to compute the weights and the method used to select the subset of scatter points used for interpolation.

Natural neighbor interpolation is based on the Thiessen polygon network of the scatter point set. The Thiessen polygon network can be constructed from the Delauney triangulation of a scatter point set. A Delauney triangulation is a TIN that has been constructed so that the Delauney criterion has been satisfied.

There is one Thiessen polygon in the network for each scatter point. The polygon encloses the area that is closer to the enclosed scatter point than any other scatter point. The polygons in the interior of the scatter point set are closed polygons and the polygons on the convex hull of the set are open polygons.

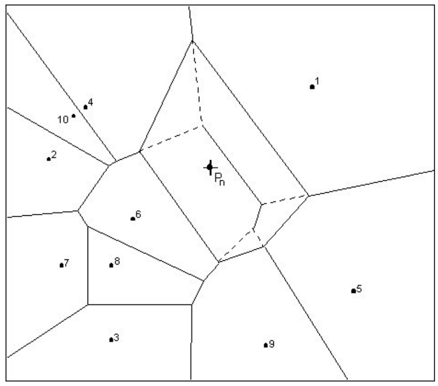
Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points. The vertices of the Thiessen polygons correspond to the centroids of the circumcircles of the triangles.

Local Coordinates

The weights used in natural neighbor interpolation are based on the concept of local coordinates. Local coordinates define the "neighborliness" or amount of influence any scatter point will have on the computed value at the interpolation point. This neighborliness is entirely dependent on the area of influence of the Thiessen polygons of the surrounding scatter points.

To define the local coordinates for the interpolation point, P_n, the area of all Thiessen polygons in the network must be known. Temporarily inserting $P_n$into the TIN causes the TIN and the corresponding Thiessen network to change, resulting in new Thiessen areas for the polygons in the neighborhood of P_n.

The concept of local coordinates is shown graphically in the following figure. Points 1–10 are scatter points and $P_n$is a point where some value associated with points 1–10 is to be interpolated. The dashed lines show the edges of the Thiessen network before $P_n$is temporarily inserted into the TIN and the solid lines show the edges of the Thiessen network after $P_n$is inserted.



Overlapping Thiessen Polygon Areas Used in Computation of Local Coordinates.

Only those scatter points whose Thiessen polygons have been altered by the temporary insertion of $P_n$are included in the subset of scatter points used to interpolate a value at P_n. In this case, only points 1, 4, 5, 6, & 9 are used. The local coordinate for each of these points with respect to $P_n$is defined as the area shared by the Thiessen polygon defined by point $P_n$and the Thiessen polygon defined by each point before point $P_n$is added. The greater the common area, the larger the resulting local coordinate, and the larger the influence or weight the scatter point has on the interpolated value at P_n.

If we define $K(n)$as the Thiessen polygon area of $P_n$and $K_m(n)$as the difference in the Thiessen polygon area of a neighboring scatter point, P_m, before and after $P_n$is inserted, then the local coordinate $\lambda_m(n)$is defined as:

$$\lambda_m(n) = \frac{K_m(n)}{K(n)}$$

The local coordinate $\lambda_m(n)$varies between zero and unity and is directly used as the weight, $w_m(n)$, in the interpolation equation. If $P_n$is at precisely the same location as P_m, then the Thiessen polygon areas for $P_n$and $P_m$are identical and $\lambda_m(n)$has a value of unity. In general, the greater the relative distance $P_m$is from P_n, the smaller its influence on the final interpolated value.

Related Topics

- Scatter Interpolation

Linear Interpolation

If the linear interpolation scheme is selected, the 2D scatter points are first triangulated to form a temporary TIN. If the surface is assumed to vary linearly across each triangle, the TIN describes a piecewise linear surface which interpolates the scatter points. The equation of the plane defined by the three vertices of a triangle is as follows:

$$Ax + By + Cz + D = 0$$

where A, B, and C, and $D$are computed from the coordinates of the three vertices (x_1, y_1, z_1), (x_2, y_2, z_2), & (x_3, y_3, z_3):

$$A = y_1(z_2 - z_3) + y_2(z_3 - z_1) + y_3(z_1 - z_2)$$
$$B = z_1(x_2 - x_3) + z_2(x_3 - x_1) + z_3(x_1 - x_2)$$
$$C = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$$
$$D = -Ax_1 - By_1 - Cz_1$$

The plane equation can also be written as:

$$z = f(x, y) = -\frac{A}{C}x - \frac{B}{C}y - \frac{D}{C}$$

which is the form of the plane equation used to compute the elevation at any point on the triangle.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the linear interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the *Interpolation Options* dialog.

Related Topics

- Scatter Interpolation

Article Sources and Contributors

Modules *Source*: http://www.xmswiki.com/xms/index.php?oldid=50159 *Contributors*: Azundel, Jcreer

Annotations *Source*: http://www.xmswiki.com/xms/index.php?oldid=65383 *Contributors*: Jcreer, Ksmith

1D Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=64914 *Contributors*: Jcreer, Jdhowlett

1D Grid Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64050 *Contributors*: Jcreer, UserRightsTester

1D Grid Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=63855 *Contributors*: Azundel, Jcreer, Jdhowlett

1D River Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=64110 *Contributors*: Jcreer, Jdhowlett

1D River Hydraulics Data Browser *Source*: http://www.xmswiki.com/xms/index.php?oldid=63859 *Contributors*: Abarney, Jcreer

1D River Hydraulics Profile Plots *Source*: http://www.xmswiki.com/xms/index.php?oldid=63860 *Contributors*: Abarney, Jcreer

1D River Module Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=63858 *Contributors*: Jcreer, Jdhowlett

Editing Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=63861 *Contributors*: Jcreer, Jdhowlett, Ladamson

Extracting Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=49505 *Contributors*: Jcreer, Jdhowlett

Interpolate Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=66101 *Contributors*: Jcreer, Jdhowlett

Managing Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=63862 *Contributors*: Jcreer, Jdhowlett

3D Cartesian Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=39966 *Contributors*: Jcreer, Jdhowlett

3D Cartesian Grid Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=49654 *Contributors*: Jcreer, Jdhowlett

Cartesian Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=48004 *Contributors*: Azundel, Jcreer

Cartesian Grid Module Overview *Source*: http://www.xmswiki.com/xms/index.php?oldid=64090 *Contributors*: Jcreer

Cartesian Grid Coordinates *Source*: http://www.xmswiki.com/xms/index.php?oldid=53269 *Contributors*: Jcreer

Cartesian Grid Data Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=64083 *Contributors*: Azundel, Jcreer, Jdhowlett

Cartesian Grid Module Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64053 *Contributors*: Jcreer, Jdhowlett

Cartesian Grid Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=64087 *Contributors*: Jcreer, Jdhowlett

Grid Frame Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=64091 *Contributors*: Jcreer

Grid Smoothing *Source*: http://www.xmswiki.com/xms/index.php?oldid=64062 *Contributors*: Jcreer, Jdhowlett

Refine Point Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64092 *Contributors*: Jcreer, Jdhowlett

Curvilinear Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=64114 *Contributors*: Jcreer

Curvilinear Grid Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64108 *Contributors*: Jcreer

Curvilinear Grid Module Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=64109 *Contributors*: Azundel, Jcreer

GIS Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=48453 *Contributors*: Jcreer, Rjones

Importing Shapefiles *Source*: http://www.xmswiki.com/xms/index.php?oldid=64382 *Contributors*: Abarney, Jcreer

GIS Module Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=64100 *Contributors*: Jcreer, Jdhowlett

GIS Module Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=64099 *Contributors*: Jcreer, Jdhowlett

GIS Module Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64098 *Contributors*: Jcreer, Jdhowlett

ArcObjects *Source*: http://www.xmswiki.com/xms/index.php?oldid=65672 *Contributors*: Jcreer, Jdhowlett

GIS to Feature Objects Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=64101 *Contributors*: Jcreer, Jdhowlett

Map Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=64159 *Contributors*: Azundel, Jcreer

Map Module Overview *Source*: http://www.xmswiki.com/xms/index.php?oldid=64226 *Contributors*: Jcreer

Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=64230 *Contributors*: Azundel, Jcreer

Generic Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=64227 *Contributors*: Jcreer

Area Property Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=64153 *Contributors*: Jcreer, Jdhowlett

Feature Stamping *Source*: http://www.xmswiki.com/xms/index.php?oldid=64228 *Contributors*: Jcreer, Jdhowlett

Mapping Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=64229 *Contributors*: Jcreer

Observations *Source*: http://www.xmswiki.com/xms/index.php?oldid=64156 *Contributors*: Jcreer, Jschultze

Particle/Drogue *Source*: http://www.xmswiki.com/xms/index.php?oldid=64231 *Contributors*: Azundel, Jcreer

Spatial Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=64155 *Contributors*: Jcreer, Jdhowlett

Spectral Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=64232 *Contributors*: Jcreer

Model Specific Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=64234 *Contributors*: Jcreer

1D Hyd Cross-Section Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=64238 *Contributors*: Jcreer, Jdhowlett

1D River Conceptual Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=64585 *Contributors*: Jcreer, Jdhowlett

ADCIRC *Source*: http://www.xmswiki.com/xms/index.php?oldid=64239 *Contributors*: Jcreer, Jdhowlett

ADCIRC Wind Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=66163 *Contributors*: Jcreer

ADH *Source*: http://www.xmswiki.com/xms/index.php?oldid=64241 *Contributors*: Jcreer, Sramiah

EFDC *Source*: http://www.xmswiki.com/xms/index.php?oldid=64242 *Contributors*: Jcreer

ESMF – Earth System Modeling Framework *Source*: http://www.xmswiki.com/xms/index.php?oldid=64243 *Contributors*: Jcreer

Generic Mesh Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=64244 *Contributors*: Jcreer

Generic Cartesian Grid Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=64247 *Contributors*: Jcreer

Synthetic Storm Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=64248 *Contributors*: Jcreer

TUFLOW Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=64865 *Contributors*: Jcreer, Jdhowlett

Interface Components *Source*: http://www.xmswiki.com/xms/index.php?oldid=64250 *Contributors*: Jcreer

Map Module Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64057 *Contributors*: Jcreer, Jdhowlett

Map Module Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=47197 *Contributors*: Jcreer, Jdhowlett

Map Feature Objects Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=58212 *Contributors*: Jcreer, Rjones

Map Module Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=64252 *Contributors*: Azundel, Jcreer, Jdhowlett

Project Explorer Items *Source*: http://www.xmswiki.com/xms/index.php?oldid=64251 *Contributors*: Jcreer

Feature Objects Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=47911 *Contributors*: Azundel, Jcreer, Sramiah

Attributes in Feature Objects Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=63756 *Contributors*: Jcreer, Jdhowlett

Map Module Selection *Source*: http://www.xmswiki.com/xms/index.php?oldid=64253 *Contributors*: Jcreer

Digitize *Source*: http://www.xmswiki.com/xms/index.php?oldid=64931 *Contributors*: Jcreer

Build Polygons *Source*: http://www.xmswiki.com/xms/index.php?oldid=64262 *Contributors*: Jcreer

Feature Object Modification: All *Source*: http://www.xmswiki.com/xms/index.php?oldid=64254 *Contributors*: Jcreer

Converting Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=64255 *Contributors*: Jcreer

Converting Feature Objects *Source*: http://www.xmswiki.com/xms/index.php?oldid=64175 *Contributors*: Jcreer, Jdhowlett

Unstructured Grid Generation from a Conceptual Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=64586 *Contributors*: Jcreer

Select/Delete Data... *Source*: http://www.xmswiki.com/xms/index.php?oldid=64257 *Contributors*: Jcreer

Arcs *Source*: http://www.xmswiki.com/xms/index.php?oldid=64258 *Contributors*: Azundel, Jcreer

Feature Object Commands *Source*: http://www.xmswiki.com/xms/index.php?oldid=64260 *Contributors*: Jcreer

Mesh Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=61916 *Contributors*: Jcreer, Jschultze

Mesh Generation *Source*: http://www.xmswiki.com/xms/index.php?oldid=51032 *Contributors*: Jcreer, Jdhowlett

Refine Attributes Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64225 *Contributors*: Jcreer, Jdhowlett

2D Mesh Options Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64200 *Contributors*: Azundel, Jcreer, Jdhowlett

2D Mesh Polygon Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=64201 *Contributors*: Jcreer, Jdhowlett

Advancing Front Triangulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64206 *Contributors*: Jcreer, Jdhowlett

Mesh Node Triangulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64199 *Contributors*: Jcreer, Jdhowlett

Merge 2D Meshes *Source*: http://www.xmswiki.com/xms/index.php?oldid=64194 *Contributors*: Jcreer

Patch *Source*: http://www.xmswiki.com/xms/index.php?oldid=61922 *Contributors*: Jcreer, Jdhowlett

Patches *Source*: http://www.xmswiki.com/xms/index.php?oldid=64197 *Contributors*: Jcreer, Jdhowlett

Paving *Source*: http://www.xmswiki.com/xms/index.php?oldid=64198 *Contributors*: Jcreer, Jdhowlett

Adaptive Tesselation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64204 *Contributors*: Jcreer, Jdhowlett

Size Function *Source*: http://www.xmswiki.com/xms/index.php?oldid=64205 *Contributors*: Jcreer

Mesh Module Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64202 *Contributors*: Bmerrell, Jcreer

Mesh Quality *Source*: http://www.xmswiki.com/xms/index.php?oldid=65268 *Contributors*: Jcreer, Jdhowlett

2D Mesh Module Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=64224 *Contributors*: Jcreer, Jdhowlett

Editing 2D Meshes *Source*: http://www.xmswiki.com/xms/index.php?oldid=50790 *Contributors*: Jcreer, Jdhowlett

2D Mesh Module Tools Right-Click Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=64907 *Contributors*: Jcreer

2D Mesh Module Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=64216 *Contributors*: Jcreer, Jdhowlett

2D Mesh Nodestrings Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=64215 *Contributors*: Jcreer, Jdhowlett

Mesh Data Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=64207 *Contributors*: Jcreer, Jdhowlett

2D Mesh Elements Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=64221 *Contributors*: Jcreer, Jdhowlett

Element types *Source*: http://www.xmswiki.com/xms/index.php?oldid=64220 *Contributors*: Jcreer, Unjedai

Boundary Triangles *Source*: http://www.xmswiki.com/xms/index.php?oldid=64219 *Contributors*: Abarney, Jcreer

Convert Elements *Source*: http://www.xmswiki.com/xms/index.php?oldid=64209 *Contributors*: Jcreer, Jdhowlett

Mesh Element Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64218 *Contributors*: Jcreer, Jdhowlett

Relax Elements *Source*: http://www.xmswiki.com/xms/index.php?oldid=64217 *Contributors*: Jcreer, Jdhowlett

2D Mesh Nodes Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=64213 *Contributors*: Azundel, Jcreer, Jdhowlett

2D Mesh Node Options Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64212 *Contributors*: Jcreer, Jdhowlett

Find 2D Mesh Node Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64211 *Contributors*: Jcreer, Jdhowlett

Renumber *Source*: http://www.xmswiki.com/xms/index.php?oldid=64208 *Contributors*: Azundel, Jcreer, Rjones

Reduce Nodal Connectivity *Source*: http://www.xmswiki.com/xms/index.php?oldid=64210 *Contributors*: Azundel, Jcreer

Particle Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=49459 *Contributors*: Jcreer, Rjones

Particle Module Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64058 *Contributors*: Jcreer, Jdhowlett

Particle Module Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=50713 *Contributors*: Jcreer, Jdhowlett

Particle Module Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=61924 *Contributors*: Jcreer, Jdhowlett

Particle Report *Source*: http://www.xmswiki.com/xms/index.php?oldid=61925 *Contributors*: Jcreer, Rjones

Extract Particle Subset *Source*: http://www.xmswiki.com/xms/index.php?oldid=47708 *Contributors*: Jcreer

Particle Grid Dataset Bin Elevations *Source*: http://www.xmswiki.com/xms/index.php?oldid=61928 *Contributors*: Jcreer

Particle Module Compute Grid Datasets *Source*: http://www.xmswiki.com/xms/index.php?oldid=61835 *Contributors*: Azundel, Bmerrell, Jcreer

Particle Module Create Datasets *Source*: http://www.xmswiki.com/xms/index.php?oldid=61834 *Contributors*: Jcreer, Jdhowlett

PTM Create Grid Datasets – Fence Diagrams *Source*: http://www.xmswiki.com/xms/index.php?oldid=61929 *Contributors*: Jcreer

Raster Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=64293 *Contributors*: Jcreer

Raster Functionalities *Source*: http://www.xmswiki.com/xms/index.php?oldid=64290 *Contributors*: Jcreer

Raster Module Interface *Source*: http://www.xmswiki.com/xms/index.php?oldid=64291 *Contributors*: Jcreer

Scatter Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=62098 *Contributors*: Jcreer, Wood

Scatter Interface Components *Source*: http://www.xmswiki.com/xms/index.php?oldid=64270 *Contributors*: Jcreer

Scatter Module Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=63783 *Contributors*: Jcreer, Jdhowlett

Scatter Module Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64060 *Contributors*: Jcreer, Jdhowlett

Scatter Module Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=63784 *Contributors*: Azundel, Jcreer, Jdhowlett

Scatter Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=64271 *Contributors*: Jcreer, Jdhowlett

Scatter Data Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=63788 *Contributors*: Jcreer, Jdhowlett

Scatter Triangles Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=63790 *Contributors*: Jcreer, Jdhowlett

Scatter Vertices Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=63782 *Contributors*: Jcreer, Jdhowlett

Scatter Project Explorer Items *Source*: http://www.xmswiki.com/xms/index.php?oldid=64272 *Contributors*: Jcreer

Scatter Module Right-Click Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=64273 *Contributors*: Jcreer, Jdhowlett

Scalar Value Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64275 *Contributors*: Jcreer, Jdhowlett

Scatter Datasets *Source*: http://www.xmswiki.com/xms/index.php?oldid=64276 *Contributors*: Jcreer

Grid Frame Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64277 *Contributors*: Jcreer, Jdhowlett

Use of DEMs in the Scatter Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=64278 *Contributors*: Cballard, Jcreer

Scatter Breakline Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64279 *Contributors*: Azundel, Jcreer

Scatter Breakline Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=63786 *Contributors*: Jcreer, Jdhowlett

Process Boundary Triangles *Source*: http://www.xmswiki.com/xms/index.php?oldid=64280 *Contributors*: Jcreer, Jdhowlett

Interpolate to Scatter Set *Source*: http://www.xmswiki.com/xms/index.php?oldid=66103 *Contributors*: Jcreer, Jdhowlett

Generate Contour Breaklines *Source*: http://www.xmswiki.com/xms/index.php?oldid=64282 *Contributors*: Jcreer

Scatter Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64263 *Contributors*: Jcreer, Jdhowlett

Laplacian Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64264 *Contributors*: Jcreer

Inverse Distance Weighted Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64266 *Contributors*: Abarney, Jcreer

Natural Neighbor Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64267 *Contributors*: Jcreer, Jdhowlett

Linear Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=64268 *Contributors*: Abarney, Jcreer

Image Sources, Licenses and Contributors

Image:SMSCurvGrid.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMSCurvGrid.jpg *License*: unknown *Contributors*: Jcreer
Image:SMSModulesToolbar.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMSModulesToolbar.jpg *License*: unknown *Contributors*: Jcreer
Image:AnnotationTree.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:AnnotationTree.jpg *License*: unknown *Contributors*: Djordan
Image:Scale.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scale.jpg *License*: unknown *Contributors*: Sramiah
Image:North-arrow.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:North-arrow.jpg *License*: unknown *Contributors*: Ksmith
Image:1D Grid Display Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:1D_Grid_Display_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:1DGridDataTree.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:1DGridDataTree.JPG *License*: unknown *Contributors*: Abarney
Image:dispopts.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dispopts.gif *License*: unknown *Contributors*: Jdhowlett
Image:Riv_profile_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Riv_profile_plot.gif *License*: unknown *Contributors*: Abarney
Image:cross_section_editor.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cross_section_editor.jpg *License*: unknown *Contributors*: Abarney
Image:Extract Cross Sections.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Extract_Cross_Sections.jpg *License*: unknown *Contributors*: Jcreer
Image:CsDb Management.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CsDb_Management.jpg *License*: unknown *Contributors*: Jcreer
Image:Cartesian Grid.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cartesian_Grid.png *License*: unknown *Contributors*: Rjones
Image:GridType1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:GridType1.gif *License*: unknown *Contributors*: Jhandy
Image:Create Transformed Grid.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Create_Transformed_Grid.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Cart Grid Display.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Cart_Grid_Display.jpg *License*: unknown *Contributors*: Jcreer
Image:CGridDataTree.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:CGridDataTree.JPG *License*: unknown *Contributors*: Abarney
File:1D Grid Frame Properties.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:1D_Grid_Frame_Properties.jpg *License*: unknown *Contributors*: Jcreer
Image:Grid Frame Properties.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Grid_Frame_Properties.jpg *License*: unknown *Contributors*: Jcreer
Image:Grid Frame Properties2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Grid_Frame_Properties2.jpg *License*: unknown *Contributors*: Jcreer
File:Cartesian Grid Smoothing Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cartesian_Grid_Smoothing_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:Refine Point2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refine_Point2.jpg *License*: unknown *Contributors*: Jcreer
Image:Refine Point.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refine_Point.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Display Curvilinear Grid.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Display_Curvilinear_Grid.jpg *License*: unknown *Contributors*: Jcreer
Image:SelectMeshElement.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectMeshElement.gif *License*: unknown *Contributors*: Jdhowlett
Image:GIS Data in SMS.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GIS_Data_in_SMS.jpg *License*: unknown *Contributors*: Azundel
File:GIS Get ArcObject Attributes Info.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GIS_Get_ArcObject_Attributes_Info.jpg *License*: unknown *Contributors*: Jcreer
Image:Add Data Dialog.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Add_Data_Dialog.jpg *License*: unknown *Contributors*: Jdhowlett
File:GIS Attribute Table.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GIS_Attribute_Table.jpg *License*: unknown *Contributors*: Jcreer
Image:Join Table.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Join_Table.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Map Properties.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map_Properties.jpg *License*: unknown *Contributors*: Jdhowlett
Image:SMS GIS Display.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_GIS_Display.jpg *License*: unknown *Contributors*: Jcreer
Image:GIS to Feature Objects Wizard -- Step 1 of 2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GIS_to_Feature_Objects_Wizard_--_Step_1_of_2.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Map Data Set.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map_Data_Set.png *License*: unknown *Contributors*: Rjones
Image:Project_explorer_coverages.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Project_explorer_coverages.jpg *License*: unknown *Contributors*: Abarney
File:Land Polygon Attributes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Land_Polygon_Attributes.jpg *License*: unknown *Contributors*: Jcreer
Image:StampedGuidebankPoints.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:StampedGuidebankPoints.jpg *License*: unknown *Contributors*: Azundel
Image:StampedCenterLine.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:StampedCenterLine.jpg *License*: unknown *Contributors*: Azundel
Image:WingWalls.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WingWalls.jpg *License*: unknown *Contributors*: Azundel
Image:Spillthrough.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Spillthrough.jpg *License*: unknown *Contributors*: Azundel
Image:Guidebank.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Guidebank.jpg *License*: unknown *Contributors*: Azundel
Image:StampedGuidebankArcs plan.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:StampedGuidebankArcs_plan.jpg *License*: unknown *Contributors*: Azundel
Image:StampedGuidebankArcs.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:StampedGuidebankArcs.jpg *License*: unknown *Contributors*: Azundel
Image:Mapping Arc Att.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mapping_Arc_Att.jpg *License*: unknown *Contributors*: Jcreer
Image:Mapping Polygon Attributes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mapping_Polygon_Attributes.jpg *License*: unknown *Contributors*: Jcreer
Image:Observ_arcs1.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Observ_arcs1.JPG *License*: unknown *Contributors*: Astrassburg
Image:Calibration_targets.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Calibration_targets.JPG *License*: unknown *Contributors*: Astrassburg
Image:SMS Observation.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Observation.jpg *License*: unknown *Contributors*: Jcreer
Image:TimeSeriesEditor.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TimeSeriesEditor.jpg *License*: unknown *Contributors*: Jdhowlett
Image:SelectCompassPlotTool.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectCompassPlotTool.jpg *License*: unknown *Contributors*: Jdhowlett
Image:CompassPlotPropertiesDlg.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CompassPlotPropertiesDlg.jpg *License*: unknown *Contributors*: Jdhowlett
File:SpectralEvents.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SpectralEvents.jpg *License*: unknown *Contributors*: Jcreer
Image:River_Conceptual Model.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:River_Conceptual_Model.jpg *License*: unknown *Contributors*: Abarney
File:Stormpath1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stormpath1.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stormpath2.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stormpath3.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stormpath4.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stormpath5.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath6.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stormpath6.jpg *License*: unknown *Contributors*: Jcreer
File:Storm Track Node Attributes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Storm_Track_Node_Attributes.jpg *License*: unknown *Contributors*: Jcreer
File:EFDC MC General.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EFDC_MC_General.jpg *License*: unknown *Contributors*: Jcreer
File:EFDC MC Time.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EFDC_MC_Time.jpg *License*: unknown *Contributors*: Jcreer
File:EFDC MC Geometry.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EFDC_MC_Geometry.jpg *License*: unknown *Contributors*: Jcreer
File:EDFC MC Output.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EDFC_MC_Output.jpg *License*: unknown *Contributors*: Jcreer
File:EFDC MC Advanced.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:EFDC_MC_Advanced.jpg *License*: unknown *Contributors*: Jcreer
File:SyntheticPBLCentralPressureGraph.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SyntheticPBLCentralPressureGraph.png *License*: unknown *Contributors*: Jcreer
Image:TUFLOW Cross-Section.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Cross-Section.jpg *License*: unknown *Contributors*: Jcreer
Image:TUFLOW Channel Attributes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Channel_Attributes.jpg *License*: unknown *Contributors*: Jcreer

Image:TUFLOW Flow Constriction3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Flow_Constriction3.jpg *License*: unknown *Contributors*: Jcreer
Image:TUFLOW Flow Constriction1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Flow_Constriction1.jpg *License*: unknown *Contributors*: Jcreer
Image:FC_poly_attributes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:FC_poly_attributes.jpg *License*: unknown *Contributors*: Jdhowlett
Image:FC_node_attributes_and_bridge.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:FC_node_attributes_and_bridge.jpg *License*: unknown *Contributors*: Jdhowlett
File:Bridge_chart.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Bridge_chart.jpg *License*: unknown *Contributors*: Jdhowlett
Image:zshape_property_dlg.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Zshape_property_dlg.jpg *License*: unknown *Contributors*: Jcreer
Image:TUFLOW Z Values Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Z_Values_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:TUFLOW Choose Arc Type.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Choose_Arc_Type.jpg *License*: unknown *Contributors*: Jcreer
File:TUFLOW Z Value.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Z_Value.jpg *License*: unknown *Contributors*: Jcreer
Image:TUFLOW Water Level Arc Att.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TUFLOW_Water_Level_Arc_Att.jpg *License*: unknown *Contributors*: Jcreer
Image:MapDataTree.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:MapDataTree.jpg *License*: unknown *Contributors*: Abarney
Image:SMS Map Display.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Map_Display.jpg *License*: unknown *Contributors*: Jcreer
File:SMS Map Legend Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Map_Legend_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:Create 1D Grid Frame.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Create_1D_Grid_Frame.jpg *License*: unknown *Contributors*: Azundel
Image:Select 1D Grid Frame.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Select_1D_Grid_Frame.jpg *License*: unknown *Contributors*: Azundel
File:Select_1D_Grid_Frame.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Select_1D_Grid_Frame.jpg *License*: unknown *Contributors*: Azundel
Image:featureobjects.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Featureobjects.gif *License*: unknown *Contributors*: Cballard
Image:Polygons.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Polygons.gif *License*: unknown *Contributors*: Cballard
File:Clean.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Clean.jpg *License*: unknown *Contributors*: Jcreer
Image:Flood_plain_domain.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Flood_plain_domain.gif *License*: unknown *Contributors*: Abarney
Image:Map_to_2D_mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map_to_2D_mesh.gif *License*: unknown *Contributors*: Abarney
Image:Map to 2D grid.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map_to_2D_grid.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Map to Scatter.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Map_to_Scatter.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Select Delete Data.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Select_Delete_Data.jpg *License*: unknown *Contributors*: Jcreer
File:Filter Arcs Tool.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Filter_Arcs_Tool.jpg *License*: unknown *Contributors*: Jcreer
File:Filter Arc Segments Tool.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Filter_Arc_Segments_Tool.jpg *License*: unknown *Contributors*: Jcreer
File:Offset Arcs Tool.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Offset_Arcs_Tool.jpg *License*: unknown *Contributors*: Jcreer
File:Redistribute Vertices.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Redistribute_Vertices.jpg *License*: unknown *Contributors*: Jcreer
File:Redistvert.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Redistvert.jpg *License*: unknown *Contributors*: Jcreer
File:BeforeSourceArcRedistribution.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:BeforeSourceArcRedistribution.jpg *License*: unknown *Contributors*: Jcreer
File:AfterSourceArcRedistribution.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:AfterSourceArcRedistribution.jpg *License*: unknown *Contributors*: Jcreer
File:Smooth Arcs Tool.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Smooth_Arcs_Tool.jpg *License*: unknown *Contributors*: Jcreer
File:Create Contour Arcs.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Create_Contour_Arcs.jpg *License*: unknown *Contributors*: Jcreer
File:Feature Arc Attributes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Feature_Arc_Attributes.jpg *License*: unknown *Contributors*: Jcreer
Image:TypesOfMeshElements.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TypesOfMeshElements.jpg *License*: unknown *Contributors*: Jdhowlett
Image:2D_Mesh_Poly_Properties.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:2D_Mesh_Poly_Properties.jpg *License*: unknown *Contributors*: Morelandtom
Image:Feature Point Node Attributes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Feature_Point_Node_Attributes.jpg *License*: unknown *Contributors*: Jcreer
Image:2D Mesh Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:2D_Mesh_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:2D Mesh Polygon Prop.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:2D_Mesh_Polygon_Prop.jpg *License*: unknown *Contributors*: Jcreer
Image:TriangulateExample.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:TriangulateExample.gif *License*: unknown *Contributors*: Abarney
Image:TriangleCircumcircle.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:TriangleCircumcircle.gif *License*: unknown *Contributors*: Abarney
Image:ValidMerge1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ValidMerge1.jpg *License*: unknown *Contributors*: Jcreer
Image:ValidMerge2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ValidMerge2.jpg *License*: unknown *Contributors*: Jcreer
Image:ValidMerge3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ValidMerge3.jpg *License*: unknown *Contributors*: Jcreer
Image:NoMerge1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NoMerge1.jpg *License*: unknown *Contributors*: Jcreer
Image:NoMerge2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NoMerge2.jpg *License*: unknown *Contributors*: Jcreer
Image:NoMerge3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NoMerge3.jpg *License*: unknown *Contributors*: Jcreer
Image:NoMerge4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NoMerge4.jpg *License*: unknown *Contributors*: Jcreer
Image:NoMerge5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:NoMerge5.jpg *License*: unknown *Contributors*: Jcreer
Image:Patch_Mesh_Polygon.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Patch_Mesh_Polygon.jpg *License*: unknown *Contributors*: Jdhowlett
Image:SMS Rectangular Patch.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Rectangular_Patch.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Triangular Patch.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Triangular_Patch.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Display Opts Mesh.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Display_Opts_Mesh.jpg *License*: unknown *Contributors*: Jcreer
Image:Mesh Quality.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mesh_Quality.jpg *License*: unknown *Contributors*: Jcreer
Image:create_node.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Create_node.gif *License*: unknown *Contributors*: Jdhowlett
Image:select_node.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Select_node.gif *License*: unknown *Contributors*: Jdhowlett
Image:CreateMeshNodestring.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:CreateMeshNodestring.JPG *License*: unknown *Contributors*: Jdhowlett
Image:SelectMeshNodestring.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectMeshNodestring.JPG *License*: unknown *Contributors*: Jdhowlett
Image:line2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Line2.gif *License*: unknown *Contributors*: Jdhowlett
Image:tri3.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tri3.gif *License*: unknown *Contributors*: Jdhowlett
Image:quad4.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Quad4.gif *License*: unknown *Contributors*: Jdhowlett
Image:line3.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Line3.gif *License*: unknown *Contributors*: Jdhowlett
Image:tri6.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tri6.gif *License*: unknown *Contributors*: Jdhowlett
Image:quad8.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Quad8.gif *License*: unknown *Contributors*: Jdhowlett
Image:quad9.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Quad9.gif *License*: unknown *Contributors*: Jdhowlett
Image:swap_edge.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Swap_edge.gif *License*: unknown *Contributors*: Jdhowlett
Image:split_merge.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Split_merge.gif *License*: unknown *Contributors*: Jdhowlett
Image:contour_label.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Contour_label.gif *License*: unknown *Contributors*: Jdhowlett
Image:Mesh Nodestring Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mesh_Nodestring_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:Delete.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Delete.jpg *License*: unknown *Contributors*: Abarney
Image:elem_1d_linear.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_1d_linear.png *License*: unknown *Contributors*: Unjedai

Image:elem_1d_quad.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_1d_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_transition.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_transition.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_linear_tri.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_linear_tri.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_tri.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_quad_tri.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_linear_quad.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_linear_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_quad.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_quad_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_quad_wcenter.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_quad_quad_wcenter.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_tet.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_tet.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_prism.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_prism.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_hex.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_hex.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_pyramid.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_pyramid.png *License*: unknown *Contributors*: Unjedai
Image:Breaklineexample.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Breaklineexample.gif *License*: unknown *Contributors*: Abarney
Image:Smoothnodestringexample.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Smoothnodestringexample.gif *License*: unknown *Contributors*: Abarney
File:Mesh Elements Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mesh_Elements_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:RefineTypesExample.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:RefineTypesExample.gif *License*: unknown *Contributors*: Abarney
Image:Refineelements.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refineelements.gif *License*: unknown *Contributors*: Abarney
File:InterpNodesExample.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:InterpNodesExample.gif *License*: unknown *Contributors*: Abarney
Image:Mesh Node Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mesh_Node_Options.jpg *License*: unknown *Contributors*: Jcreer
File:Find 2D Mesh Node.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Find_2D_Mesh_Node.jpg *License*: unknown *Contributors*: Jcreer
File:NewNodePattern.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:NewNodePattern.png *License*: unknown *Contributors*: Azundel
file:Ptm particle set.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ptm_particle_set.jpg *License*: unknown *Contributors*: Jcreer
Image:Particle Display.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Particle_Display.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS-Particle_Module_Create_Data_Sets.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS-Particle_Module_Create_Data_Sets.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Scatter Set Example.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Set_Example.png *License*: unknown *Contributors*: Rjones
Image:SelectScatterPoint.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectScatterPoint.JPG *License*: unknown *Contributors*: Jdhowlett
Image:Scatter Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:Triangulation_optimization.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Triangulation_optimization.jpg *License*: unknown *Contributors*: Abarney
Image:Scatter Merge Sets.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Merge_Sets.jpg *License*: unknown *Contributors*: Jcreer
Image:Scatter Grid frame.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Grid_frame.jpg *License*: unknown *Contributors*: Jcreer
Image:Scatter Filter Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Filter_Options.jpg *License*: unknown *Contributors*: Jcreer
File:Triangulate1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Triangulate1.jpg *License*: unknown *Contributors*: Abarney
File:Triangulate2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Triangulate2.jpg *License*: unknown *Contributors*: Abarney
File:DelaunayCriterion.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:DelaunayCriterion.jpg *License*: unknown *Contributors*: Jcreer
Image:Check_long_triangles.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Check_long_triangles.jpg *License*: unknown *Contributors*: Abarney
Image:Scalar Value Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scalar_Value_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:gridframe.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gridframe.gif *License*: unknown *Contributors*: Abarney
Image:Refresh.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refresh.jpg *License*: unknown *Contributors*: Abarney
Image:Frame.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Frame.JPG *License*: unknown *Contributors*: Jdhowlett
File:Scatter Grid Frame.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Grid_Frame.jpg *License*: unknown *Contributors*: Jcreer
Image:Scatter Breakline Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Breakline_Options.jpg *License*: unknown *Contributors*: Jdhowlett
Image:SMS Process Boundary Triangles.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Process_Boundary_Triangles.jpg *License*: unknown *Contributors*: Jcreer
Image:Scatter_Set_Bow_Tie_Triangles.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Set_Bow_Tie_Triangles.jpg *License*: unknown *Contributors*: Jdhowlett
File:Autobreaklines.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Autobreaklines.jpg *License*: unknown *Contributors*: Jcreer
Image:Interpolation.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Interpolation.jpg *License*: unknown *Contributors*: Jcreer
File:LaplacianInterpolation zoomed in.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:LaplacianInterpolation_zoomed_in.png *License*: unknown *Contributors*: Jcreer
File:LaplacianInterp.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:LaplacianInterp.jpg *License*: unknown *Contributors*: Jcreer
Image:Four_quadrants.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Four_quadrants.jpg *License*: unknown *Contributors*: Abarney
Image:WMSidw_fig2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSidw_fig2.jpg *License*: unknown *Contributors*: Jcreer
Image:WMSidw_fig3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:WMSidw_fig3.jpg *License*: unknown *Contributors*: Jcreer
Image:Delauney_triangulation.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Delauney_triangulation.jpg *License*: unknown *Contributors*: Abarney
Image:Thiessen_polygons.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Thiessen_polygons.jpg *License*: unknown *Contributors*: Abarney