SMS User Manual (v11.2)

Introduction and Functionalities

Contents

Articles

1. Introduction

What is SMS?

SMS (Surface-water Modeling System) is a complete program for building and simulating surface water models. It is a graphical user interface and analysis tool that allows engineers and scientists to visualize, manipulate, analyze, and understand numerical data and associated measurements. Many of the tools in SMS are generic. They are designed to facilitate the establishment and operation of numerical models of rivers, coasts, inlets, bays, estuaries, and lakes. It features 1D and 2D modeling and a unique conceptual model approach. Some of the currently supported models in SMS include ADCIRC, BOUSS-2D, CGWAVE, CMS-Flow, CMS-WAVE (WABED), FESWMS, GenCade, PTM, STWAVE, TABS, and TUFLOW.

Introduction to SMS

- The Highlights provide a summary of SMS capabilities.
- The SMS Tutorials are step-by-step guides for building models and using SMS features. They are an excellent place to begin learning how to use SMS.
- See Layout of the Graphical Interface for more information on the organization of the toolbars, menus, and windows in SMS.
- Much of the SMS functionality is divided into Modules based upon the type of data (grids, meshes, GIS, etc). SMS also contains features that are not tied to specific modules.
- SMS supports a number of Numerical Models with a variety of uses including hydraulics, wave modeling, and particle tracking.

History

SMS was initially developed by the Engineering Computer Graphics Laboratory (later renamed in September, 1998 to Environmental Modeling Research Laboratory or EMRL) at Brigham Young University [1] in the late 1980s on Unix workstations. The development of SMS was funded primarily by The United States Army Corps of Engineers [2]. It was later ported to Microsoft Windows platforms in the mid 1990s and support for HP-UX, IRIX, DEC-OSF, and Solaris platforms was discontinued.

In April 2007, the main software development team at EMRL entered private enterprise as Aquaveo LLC [3]. and continue to develop SMS and other software products, such as WMS (Watershed Modeling System) and GMS (Groundwater Modeling System).

References

[1] http://home.byu.edu/home/
[2] http://www.usace.army.mil/
[3] http://www.aquaveo.com/

Tutorials

A rich set of step-by-step tutorials has been developed to aid in learning how to use SMS.

Tutorial Installation

There are two options for installing tutorials. You can download the tutorials and files individually by subject matter in the SMS learning center or you can download an install that includes all of the core tutorials and files. Some of the additional tutorials (TUFLOW, PTM, CGWAVE) still need to be downloaded separately. Since most users only need a subset of the tutorials, the recommended approach is to download and install them as needed from the SMS learning center.

Accessing Tutorials Through the SMS Learning Center

The SMS Learning Center is the portal to all of the training you will need to learn how to use SMS. It is there that you can gain access to all of the SMS tutorial documents and files via download. To access the SMS Learning Center and tutorials click here [1], or follow these steps:

- Go to www.aquaveo.com [3]
- Click on the *SMS logo* found on the right side of the screen. This will take you to the SMS homepage.
- Once at the *SMS homepage*, click on the *SMS Learning Center* Icon.
- In the *SMS Learning Center* page, scroll down to the tutorials. There, the tutorials are divided into two groups: general SMS tutorials, and tutorials for specific models found within SMS. All are available for download.

Opening and Downloading Tutorials

Each tutorial consists of a PDF document, and its associated tutorial files.

- Open the PDF document by right-clicking on it, then selecting *Open* link in new tab.
- Then, click on the files icon and choose the *Save as...* option.
- You can choose the directory in which you would like to run and save your tutorial files.
- Once the files have finished downloading, select the *Open* option.
- When the data files come up, you must extract the files by clicking on the Extract all files option.
- Once the files are extracted, you are ready to begin with the tutorial.

Available Tutorials

Tutorials are available through the SMS Learning Center [1]. Below is a list of tutorials available at the SMS Learning Center.

SMS Tutorials		
Data Visualization	Feature Stamping	GIS
Google Earth	Import From Web	Mesh Editing
Observation	Overview	Scattered Datasets
Sensitivity		
SMS Model Tutorials		
ADCIRC	ADCIRC LTEA Meshing	BOUSS2D
CGWAVE	Additional CGWAVE	CMS Flow
CMS Wave	FESWMS	FESWMS Steering

FESWMS Weirs	Generic Mesh Model	PTM
RMA2	RMA2 Steering	RMA4
STWAVE	TUFLOW 1D	TUFLOW 2D
TUFLOW AD	Additional TUFLOW	

°*Note: Do not save the tutorial documents and files to the Program Files directory.*

example of pdf link: ADCIRC

Related Topics

- Sample Problems

References

[1] http://www.aquaveo.com/sms-learning/

Sample Problems

In addition to the tutorial files, numerous test cases are available for download from the Aquaveo Verification Repository [1].

The Aquaveo Verification Repository is designed to store case studies which can be used to verify the accuracy and capabilities of various numeric models. The case studies contain within the repository will eventually help to build a selection tool. The selection tool will use numeric model results to suggest appropriate models to use for a study. The results will be determined by the performance of the model when faced with certain site characteristics.

The cases can be searched using the "Search" links found in the navigation menu. Each model type is contained in a separate repository. The search page will allow you to search for case studies containing particular attributes. Performing a search with no selections will allow you to browse all studies contained in the repository.

SMS users may add test cases to the repository. In order to add studies to the repository you must create an account. The provided contact information will not be released to anyone. We invite everyone to contribute to the repository. More cases means the models can be tested more thoroughly, which will result in a better selection tool.

Related Topics

- SMS Tutorial Files

External Links

- Aquaveo Verification Repository [1]
- Hollingsworth, Jason M (2008). Foundational Data Repository for Numeric Engine Validation. Thesis, Brigham Young University. [2]

References

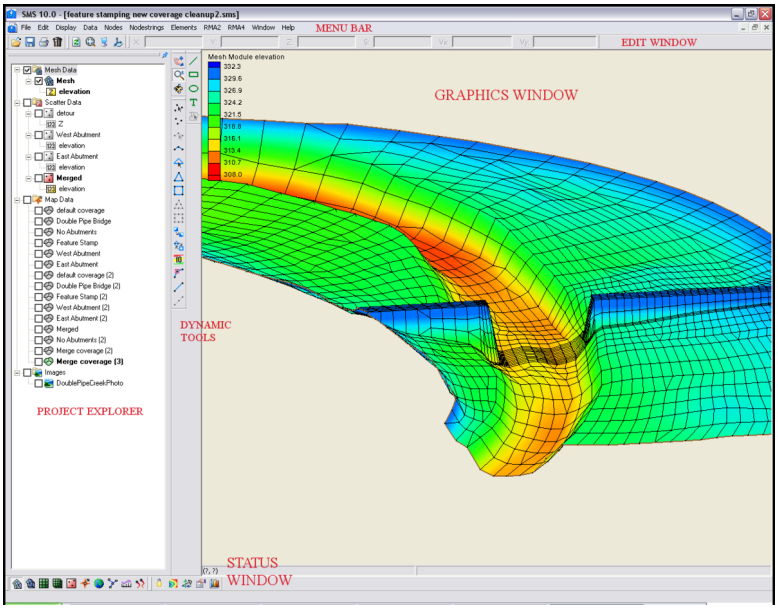
[1] http://verification.aquaveo.com/
[2] http://contentdm.lib.byu.edu/ETD/image/etd2661.pdf

Highlights

Layout

- The project explorer shows data currently loaded in project
- Menu bar depends upon the active module and model
- Edit window show x, y, z, scalar, and vector values
- Edit window values can be edited in some circumstances
- The status window on the bottom of the graphics window shows coordinates and selection information
- Help information is displayed at the bottom of the SMS screen
- Several toolbars are used in SMS. The dynamic tools change based upon the current module.

More Info...



SMS Layout

Modules

- Data is divided into modules based upon the data type

Module Toolbar

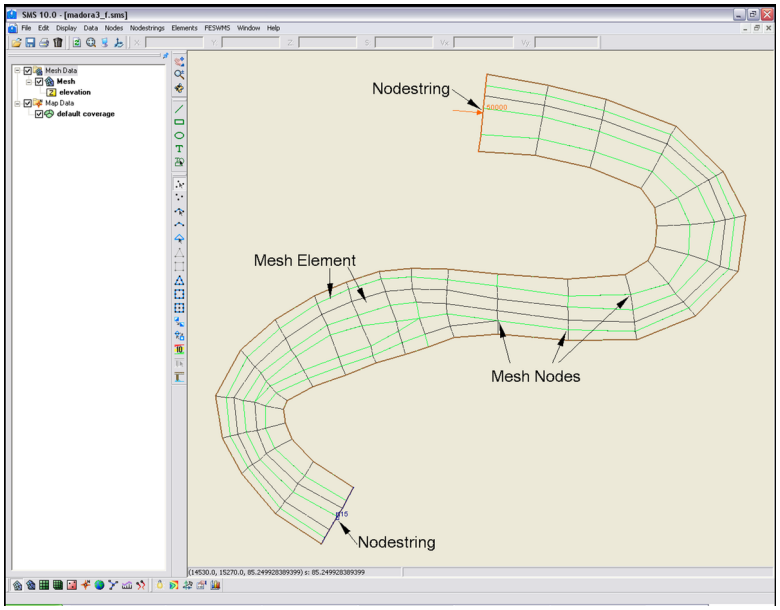
- There is always one active module
- The menus and toolbars are based upon the active module
- The current module may be selected in module bar or by selecting an object in project explorer

More Info...

Mesh Module

- Used to create, edit, and visualize mesh data
- Also referred to as unstructured grids or finite element meshes
- Meshes defined by nodes and elements
- Several element types are supported

More Info...

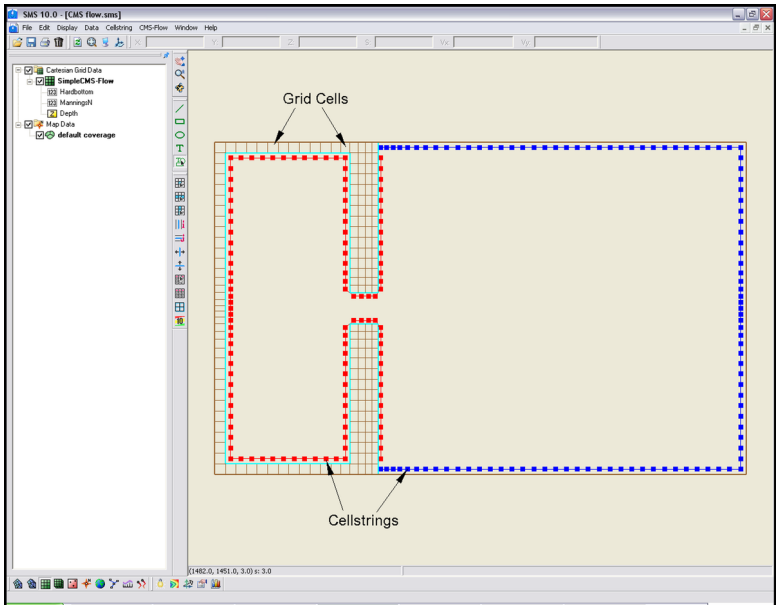


Mesh Module Example

Cartesian Grid Module

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

More Info...

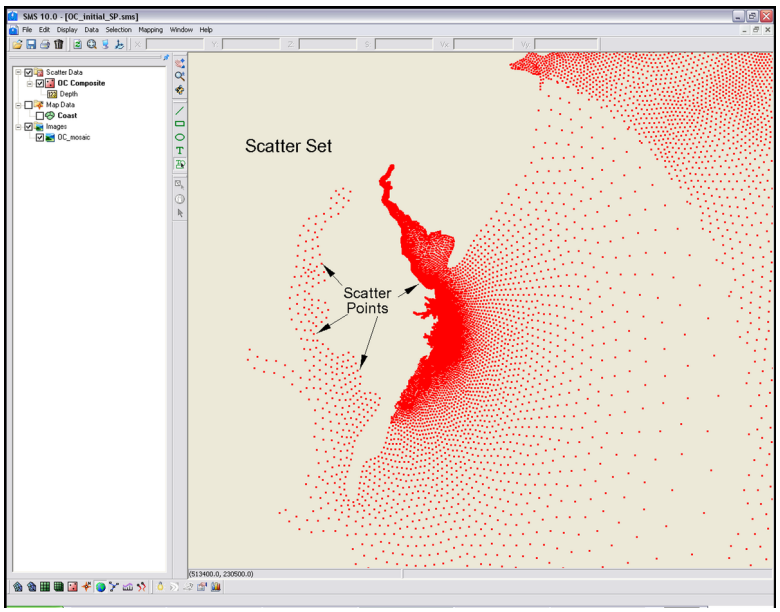


Cartesian Grid Example

Scatter Module

- 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)

More Info...

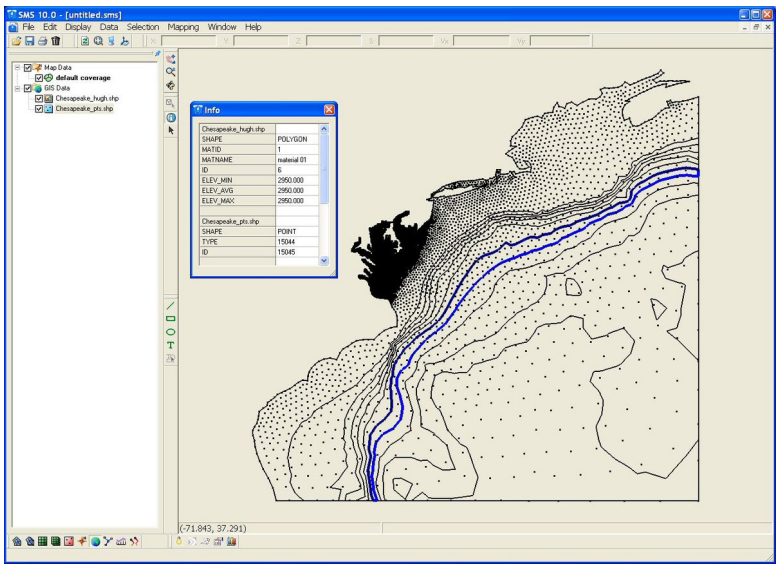


Scatter Set Example

GIS Module

- 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)

More Info...

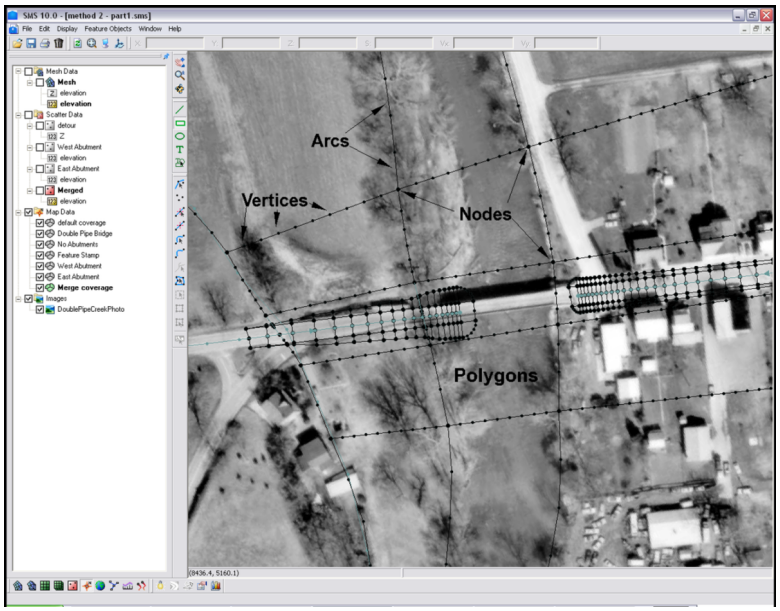


GIS Data in SMS

Map Module/Conceptual Models

- 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

More Info...

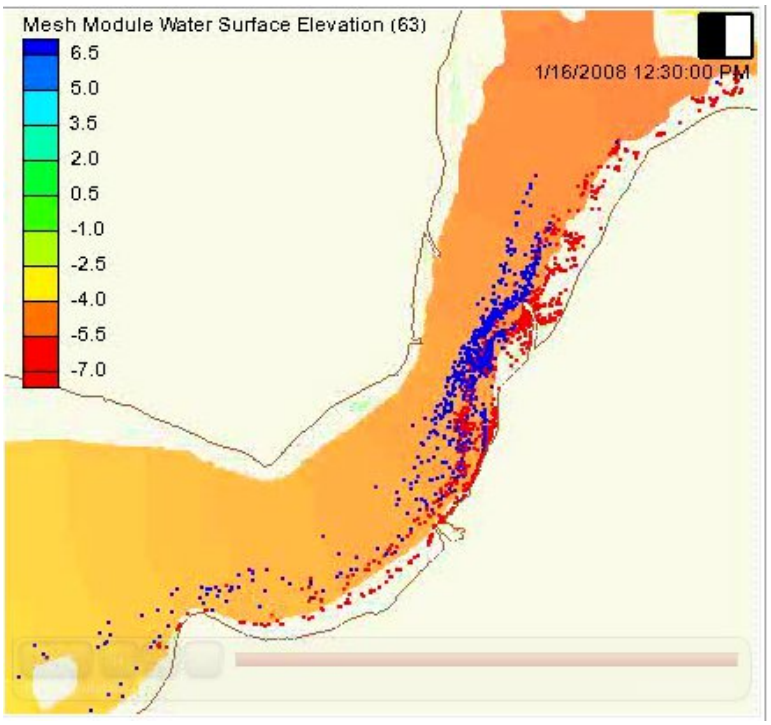


Map Module Example

Particle Module

- Visualize particle/path data
- Supports PTM module which computes particle positions through time based upon hydrodynamics and wave effects

More Info...



PTM Particle Trace

Models

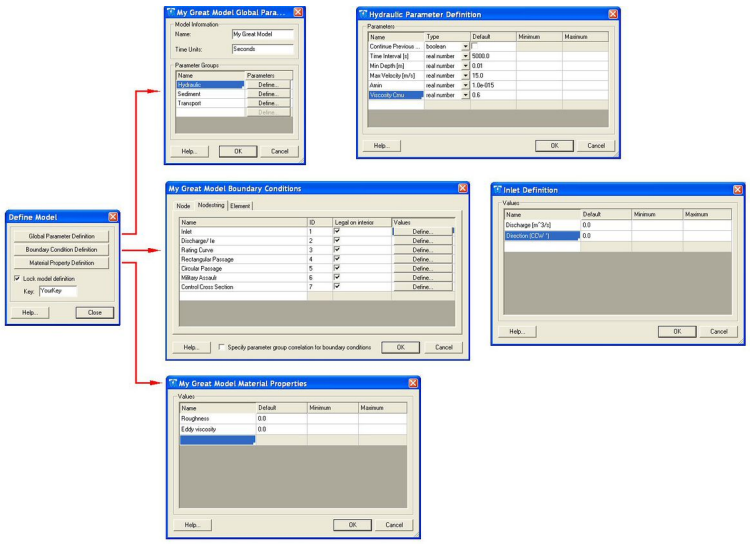
- SMS is a graphical interface that supports many numeric models
- The models were developed by government or private entities
- Hydrodynamic models compute water surface elevations and velocities
- Wave models compute wave characteristics
- Genesis is a shoreline model that predicts how the coastline will move based upon long term wave information
- PTM tracks particle positions through time based upon hydrodynamics and wave effects

More Info...

Generic Model Interface

- Allows creation of a user defined mesh module interface to use SMS with a model not natively supported
- User defines available model parameters and boundary condition options
- User defined interface can be used to build models
- User data is exported into ASCII data that can be read as input for a numeric model

More Info...



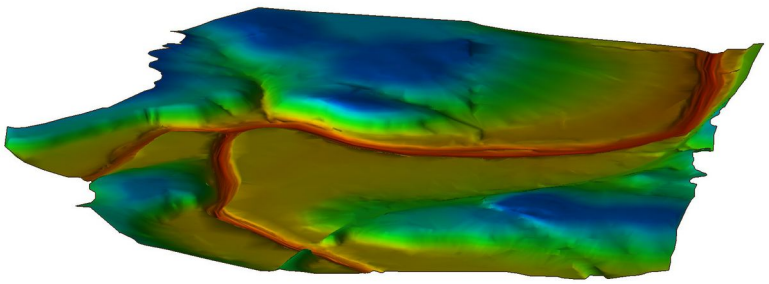
Generic Model Definition and use

Visualization Tools

Contours

- Visualize scalar datasets
- Linear, color filled or both at the same time
- Variable level of transparency
- Full control of ranges and colors
- Precision control for labels and legends

More Info...

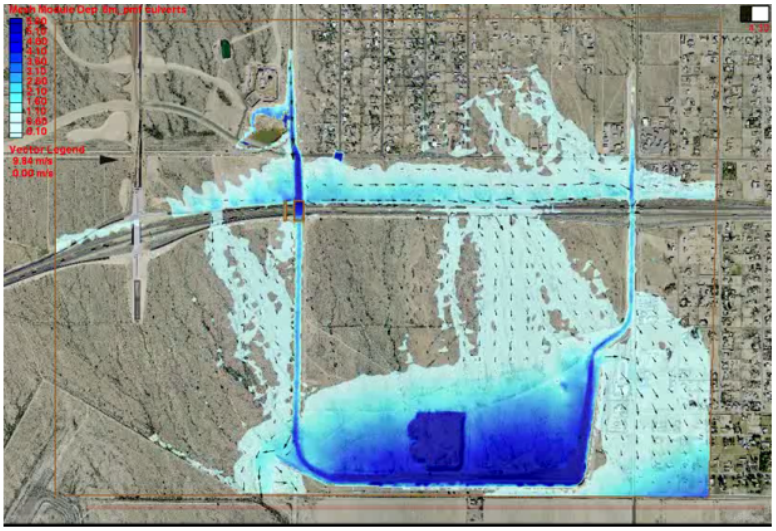


3D view of color filled contours

Vectors

- Visualize vector datasets as arrows
- Constant size or vary by magnitude
- Show just a range of magnitudes
- Color by magnitude

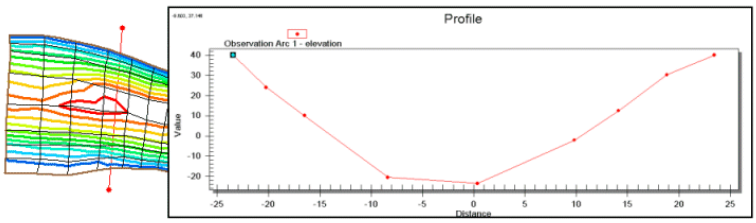
More Info...



Vectors display

Plots

- 2D plots to visualize results and compare to measured values
- Profile plots view scalar data along an arc
- Time-series plots view scalar, vector, or flux (flowrate) data at a point or across an arc



A plot showing a cross-section view of a river channel

- Several kinds of plots can be used to compare model results with measured data

More Info...

Functional Surfaces

- Surface with elevation based upon scalar dataset values
- Very useful for wave models and models with large change in water surface elevation
- Elevations can be exaggerated to better visualize dataset variations
- Surfaces can have a solid color or use color filled contours
- Transparency can be used to allow see through surfaces

More Info...

Animations

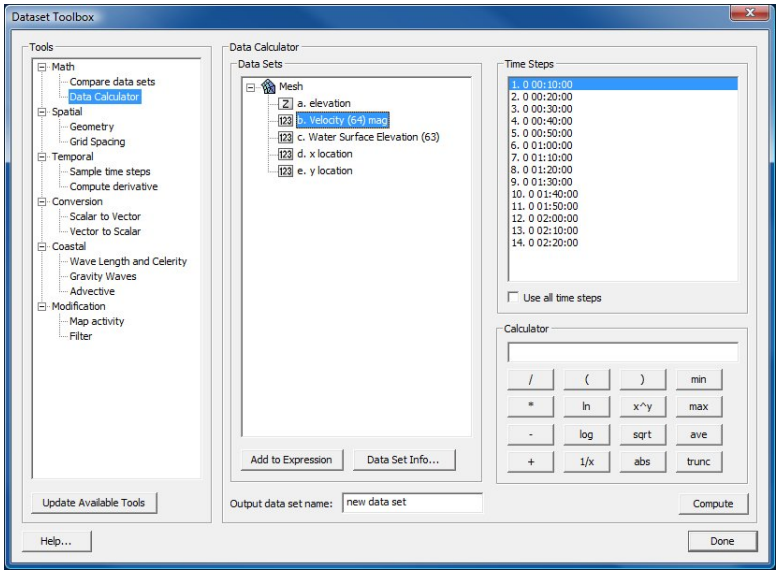
Flow trace animation

- Several types of AVI animations (film loops) can be generated by SMS
- Transient data animation shows model changes through time (contours, vectors, etc)
- Flow trace uses vector data to generate flow paths through the geometry
- Drogue plots use user specified starting locations and show how the particles would flow through a vector field
- Multiple view animations show the data while transitioning between different views
- Plot window animations show plots changing through time

More Info...

Data Tools

Data Calculator



SMS Data Calculator

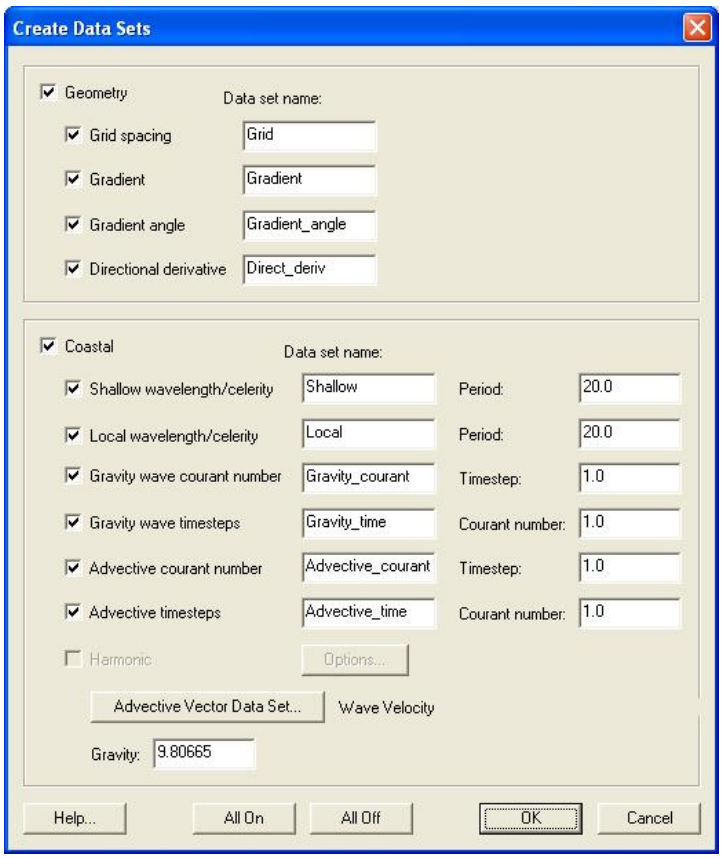
- Performs mathematical calculations on scalar datasets
- Calculations can include any number of scalar datasets and user supplied numbers
- Useful for computing derived values such as Froude numbers
- Useful for comparing scalar datasets

More Info...

Create Datasets

- Automatically generate commonly used datasets

More Info...

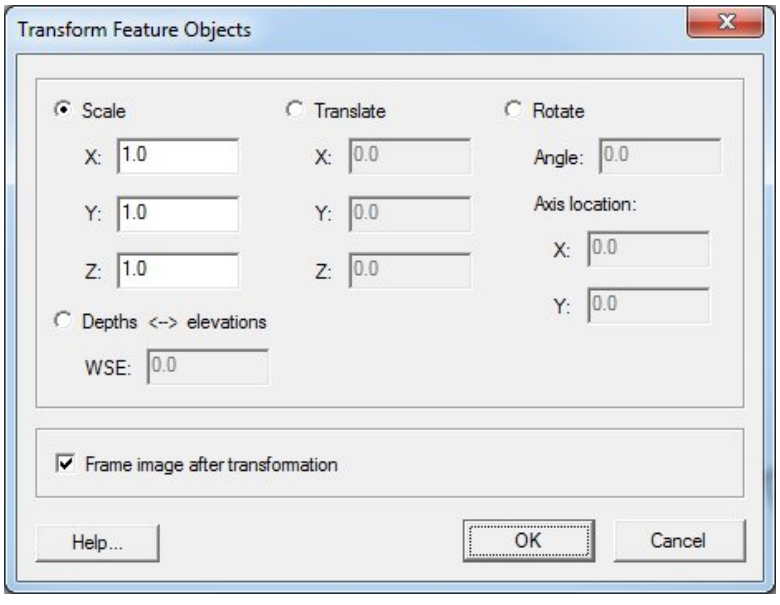


Create Datasets

Data Transform

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

More Info...

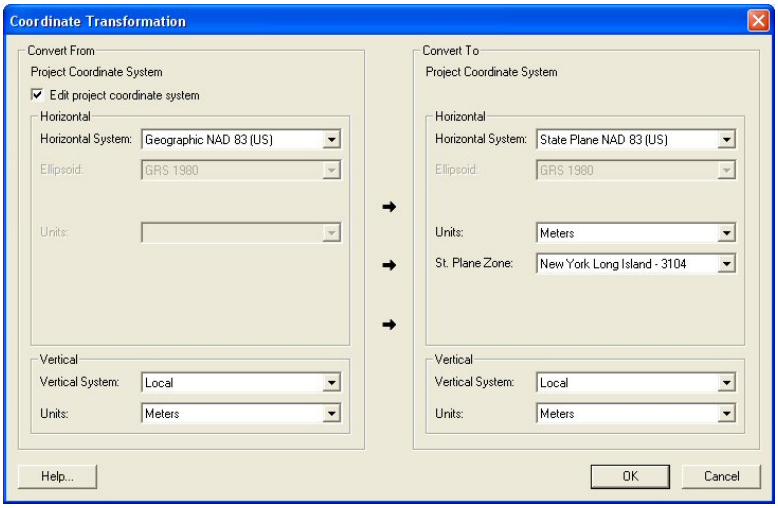


Transform Data

Coordinate Systems

- Associate a coordinate system with your data
- Convert data from one coordinate system to another
- Coordinate systems include Geographic, UTM, and State Plane coordinate systems

More Info...

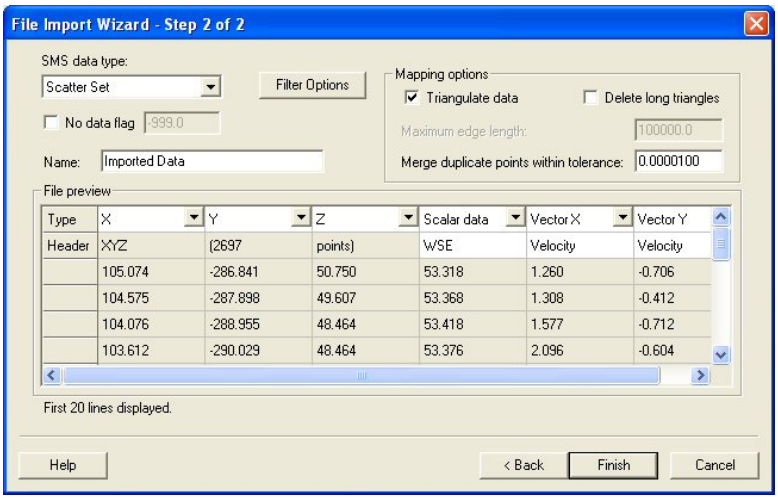


Coordinate Conversions

Import Wizard

- Read columnar ASCII data into SMS
- Columns can be fixed width or delimited by specific characters
- Data can be read as mesh, scatter, or map data

More Info...



Text file read in through import wizard

Miscellaneous Tools

Image Support

- Multiple images can be read/viewed at the same time
- Many image formats are supported including JPG, TIFF, PNG, MrSID, and ECW
- Images can be geo-referenced to view images along with other data
- Images can be draped over mesh or scatter data
- Image pyramids can be created which improves viewing at various zoom levels

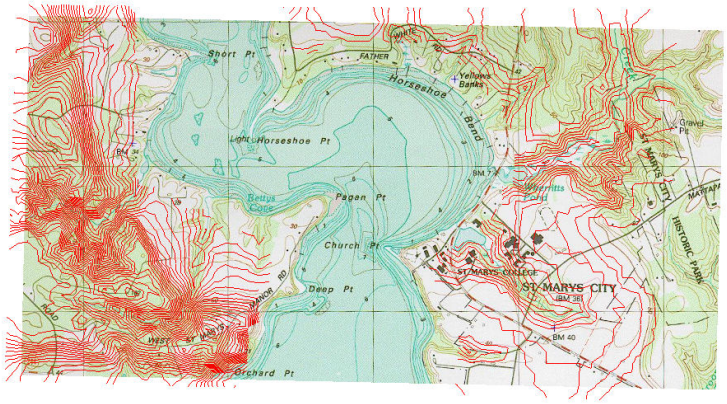
More Info...

A color aerial photo under a semi-transparent topographic map

CAD Support

- AutoCAD DXF and DWG files can be read into SMS (support of DGN format is under development)
- Supports up to AutoCAD version 2007
- CAD data is displayed in 3D
- CAD data can be converted to map or scatter data

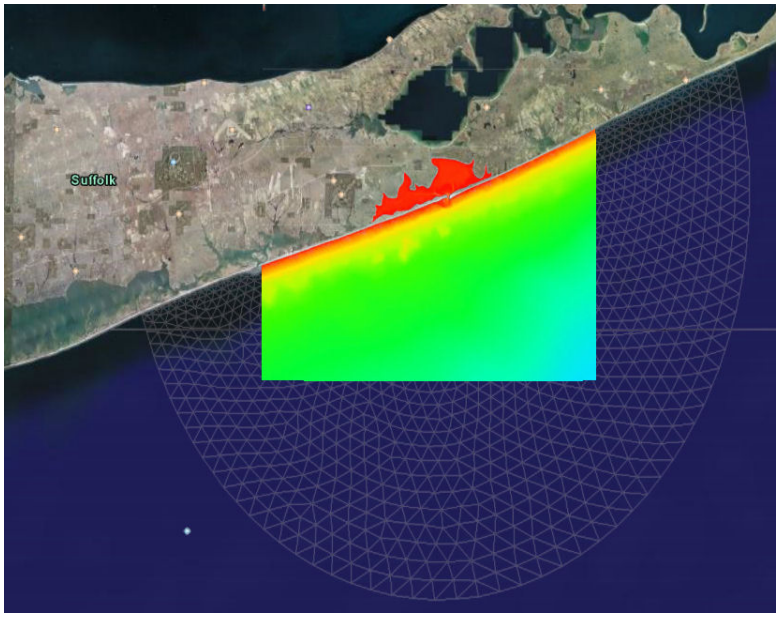
More Info...



Using CAD data in SMS

Export Options

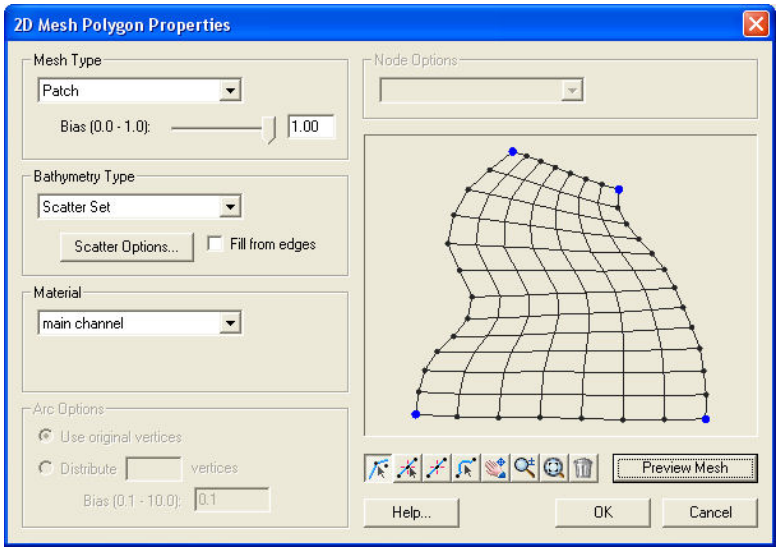
- Graphics window can be copied to the clipboard
- Current view can be exported in KML format for visualization in Google Earth



Model Data Visualized in Google Earth

Meshing Options

- 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)



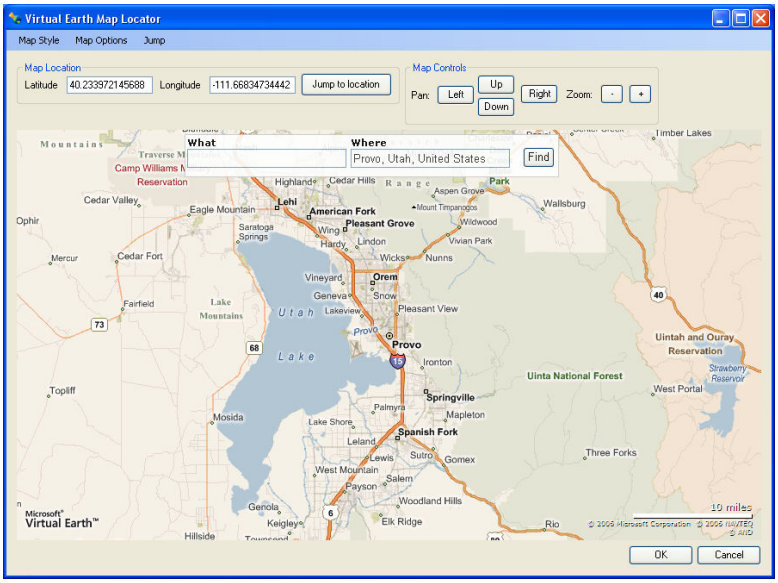
2D Mesh Polygon Properties dialog

More Info...

Import from Web

- Easy to use navigation tool allows user to choose model location
- Image data is downloaded from USGS terraserver
- Image options include aerial photos, topographic charts, and urban (higher resolution color)

More Info...

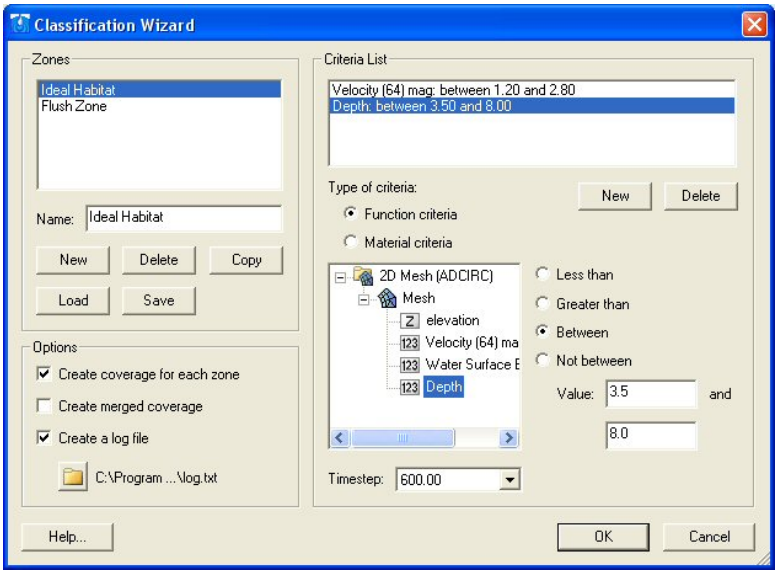


Virtual Earth Map Locator

Zonal Classification

- Generate a map coverage identifying areas that meet specific requirements
- Requirements can be based upon dataset values such as less than a specific value or based upon materials in an area property coverage

More Info...



Zonal Classification

What's New in SMS 11.2

Release Notes

General Features

Dataset Toolbox

- A *Merge Datasets* option is now available in the Dataset Toolbox.

Interface Components

- **Projection** commands have been moved to the *Display* menu.
- A **Reproject All** command has been added in the *Display* menu.
- **Help** buttons in dialogues will open the corresponding page in the XMSWiki.

Dynamic Model Interface

- We are excited to present a more flexible and powerful alternative to the generic model interface that has been supported by SMS for several years. This methodology allows a model developer to define all the attributes of an interface for a specific model in an XML file. SMS will read this file when launched, and then interact with the model. This schema is defined in Dynamic Model Interface Schema. The dynamic model interface includes:
  - Model specific menu commands.
  - Model specific parameters.
  - Model specific boundary conditions.
  - Model specific file formats (for both reading and writing).
  - Model specific execution procedures including as many different pre-run utilities or execution steps as are required by the model.

MIKE 21 (*.mesh) File Support

- MIKE 21 (*.mesh) files can now be loaded into SMS. See the article MIKE 21 *.mesh.

LIDAR File Support

- SMS can now recognizes a LIDAR file when a user asks the system to read such a file. The user can choose the import methodology to load selected layers from the file as either a scatter set or a raster. SMS also loads an image of the data in the specified format.

Saving Color Palettes

- Color palettes will now be saved when doing *File* | **Save Settings**. When SMS is loaded, any saved palettes will automatically be loaded as defaults.

NOAA HURDAT File Support

- SMS will now load storms for the standard NOAA HURDAT format. The file will need to have a *.hurdat2 extension to be recognized by SMS.

Contour Options

- The default values for contour options in a dataset will be that of the module.

Vector Display Options

- New display option feature for showing vector arrows at a constant elevation. See Vector Display Options for more information.
- New display option for vector arrows to follow flow path (vectors curve). This is done by selecting "Arrows follow flow path" under the *Vectors* tab in the *Display Options* dialog.

  Warning: this can be slow if displaying lots of vectors.'

Spectral Coverage

- Spectral coverages are now used to store all spectral data by location and time. These coverages are then used as spectral input for CMS-Wave and STWAVE, and are also used to view spectral output generated by the models in observation and nesting files.

Module Features

Curvilinear Grid

- New tool to split and merge row/column of a curvilinear grid. See the Curvilinear Grid Tools section for more information.
- Ability to merge two curvilinear grids. See the article Curvilinear Grid Module for more information

Mesh Modules

- SMS 11.2 allows for use of multiple meshes.

Model Features

SRH-2D

- There is now a custom interface for the SRH-2D model developed by the United States Bureau of Reclamation (http://www.usbr.gov/pmts/sediment/model/srh2d/index.html).This model is a finite volume engine that is very stable in wetting/drying conditions. It has been tested extensively by various agencies, academic and commercial users.

ADCIRC

- There is now support for time varying bathymetry in ADCIRC.
- You can now do a spatially average interpolate from a raster/DEM to an ADCIRC mesh.

ADH

- The CSTORM-MS coupler has been updated to now include AdH meshes in addition to ADCIRC. This is in preparation for linking to AdH simulations for sediment transport in the CSTORM-MS.

CMS-Flow

- The cards, values and comments are now displayed in color in the *CMS-Flow Advanced* tab
- Added projection cards

CMS-Wave

- All input spectra is now handled in a spectral coverage. This allows the user to utilize multiple observed spectra to drive the model in the same manner that a nested simulation has been used in the past. The timestamps assigned to the spectral data are matched up (or interpolated to match) with the times specified for each case. In order to correctly assign spectral data for each case, SMS now requires CMS-Wave simulations to have a reference time.

STWAVE

- The model control now uses a spectral coverage to specify the cases/wave states that will be included in a simulation. This allows a user to utilize multiple observed spectra to drive the model in the same manner that a nested simulation has been used in the past. The timestamps assigned to the spectral data are matched up (or interpolated to match) with the times specified for each case. In order to correctly assign spectral data for each case, SMS now requires STWAVE simulations to have a reference time.

1.1. General Information

General Interface Features

This article addresses features that do not belong to specific modules.

Display

Editing the View with the Mouse

The user can now navigate easily with mouse controls. The three controls include:

- If your mouse has a middle button (or a mouse wheel), you can scroll the wheel to zoom in and out.
- If your mouse has a middle button (or a mouse wheel), you can hold it down and drag to pan the view.
- You can also hold down both the right and left mouse buttons and drag to rotate the view.
- Display Options

Display Options in SMS refers to the control of what entities are displayed, and how (color and style) they are displayed. Each entity in each module has its own display options. The display options for the active module are shown when the *Display Options* dialog opens.

Visualization

Post-Processing tools inside SMS help to visualize the model solution created by running the simulation through the solver. Post-processing tools available on solution data vary based on the module but include:

- 2D Plots
- Animations
- Contours
- Calibration Targets
- Vector Visualization

Data that does not belong to a specific module

- CAD Data
- Annotation Layers
- Images
- LandXML Files

Related Topics

- Coordinates
- File Formats

Keyboard Shortcuts

Many commands in SMS are can be accessed using keyboard shortcuts.

Standard Menus

Keyboard Shortcuts

Modifier	Key	Command
	F	*Display* **| Frame Image**
	SPACE	*Display* **| Refresh**
CTRL	*C*	*Edit* **| Copy to Clipboard**
CTRL	*V*	*Edit* **| Paste Tabular Data**
CTRL	*A*	*Edit* **| Select All**
CTRL	*O*	*File* **| Open**
CTRL	*P*	*File* **| Print**
CTRL	*S*	*File* **| Save Project**
	F1	*Help* **| SMS Help**
	F2	**Pan**
	F3	**Zoom**
	F4	**Rotate**
SHIFT	*F*	*Display* **|** *View* **| Front**
SHIFT	*O*	*Display* **|** *View* **| Oblique**
SHIFT	*V*	*Display* **|** *View* **| View Options**
SHIFT	*P*	*Display* **|** *View* **| Plan**
SHIFT	*Z*	*Display* **|** *View* **| Previous**
SHIFT	*S*	*Display* **|** *View* **| Side**

Related Topics

- Right-Click Menus
- Layout of the Graphical Interface

Publications

SMS Related Publications and Reports

The following is a partial list of publications and reports related to the use of SMS. Please feel free to make additions that have not been listed.

WES reports are available through the Interlibrary Loan Service from the US Army Engineer Waterways Experiment Station (WES) Library, telephone number (601) 634-2355. National Technical Information Service (NTIS) report numbers may be requested from WES Librarians. To purchase a copy of a report, call NTIS at (703) 487-4780.

External Lists of Articles

- ADCIRC publications [1]
- Journal articles using SMS by Prof. Greg Pasternack, UC Davis [2]

2008

- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [3]
- Jul 2008 ERDC/CHL CHETN-IV-71 Particle Tracking Model (PTM) in the SMS 10: IV. Link to Coastal Modeling System [4]

2007

- SRH-2D Training Presentation [5]
- Aug 2007 ERDC/CHL CHETN-I-76 Modeling Nearshore Waves for Hurricane Katrina [6]
- Aug 2007 ERDC/CHL CHETN-I-75 Full-Plane STWAVE with Bottom Friction: II. Model Overview [7]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [8]
- May 2007 ERDC/CHL CHETN-I-73 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [9]
- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [9]
- May 2007 ERDC/CHL CHETN-I-74 WABED Model in the SMS: Part 2. Graphical Interface [10]
- Lai, Y.G. and Bountry, J.A. (2007). "Numerical modeling study of levee setback alternatives for lower Dungeness River, Washington" [11]

2006

- Sep 2006 9th International Workshop On Wave Hindcasting and Forecasting Jane McKee Smith Modeling Nearshore Waves For Hurricane Katrina [12]
- Sep 2006 ERDC/CHL TR-06-20 PTM: Particle Tracking Model [13]
- Aug 2006 ERDC/CHL TR-06-9 Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change [14]
- Jul 2006 ERDC/CHL CHETN-III-73 Wave-Action Balance Equation Diffraction (WABED) Model: Tests of Wave Diffraction and Reflection at Inlets [15]
- FISC 2006 Short Course Presentation [16]
- Mar 2006 ERDC/CHL CHETN-I-71 Full Plane STWAVE: SMS Graphical Interface [17]
- Feb 2006 ERDC/CHL CHETN-IV-67 Frequently-Asked Questions (FAQs) About Coastal Inlets and U.S. Army Corps of Engineers' Coastal Inlets Research Program (CIRP) [18]
- Lai, Y.G. and Bountry, J.A. (2006). "Numerical hydraulic modeling and assessment in support of Elwha Surface Diversion Project." [19]

- Lai, Y.G., Holburn, E.R., and Bauer, T.R. (2006)."Analysis of sediment transport following removal of the Sandy River Delta Dam." [20]
- Bountry J.A. and Lai, Y.G. (2006)."Numerical modeling of flow hydraulics in support of the Savage Rapids Dam removal." [21]

2005

- Jul 2005 ERDC TN-DOER-D4 Particle Tracking Model (PTM) in the SMS: I. Graphical Interface [22]
- Jul 2005 ERDC TN-DOER-D5 Particle Tracking Model (PTM): II. Overview of Features and Capabilities [23]
- Jul 2005 ERDC TN-DOER-D6 Particle Tracking Model (PTM) in the SMS: III. Tutorial with Examples [24]
- May 2005 ERDC/CHL CHETN-I-70 BOUSS-2D Wave Model in SMS: 2. Tutorial with Examples [25]
- Mar 2005 ERDC/CHL CHETN-I-69 BOUSS-2D Wave Model in the SMS: 1. Graphical Interface [26]
- May 2005 ERDC/CHL CHETN-IV-63 Representation of Nonerodible (Hard) Bottom in Two-Dimensional Morphology Change Models [27]
- 2005 US-China Workshop Paper [28][29]

2004

- May 2004 ERDC/CHL TR-04-2 Two-Dimensional Depth-Averaged Circulation Model M2D: Version 2.0, Report 1, Technical Documentation and User's Guide [30]
- Mar 2004 ERDC/CHL CHETN-I-68 How to Use CGWAVE with SMS:An Example for Tedious Creek Small Craft Harbor [31]

2003

- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D and STWAVE [32]
- Jun 2003 ERDC/CHL CHETN-I-67 Tedious Creek Small Craft Harbor:CGWAVE Model Comparisons Between Existing and Authorized Breakwater Configurations[33]

2002

- Jun 2002 ERDC/CHL CHETN-I-66 Grid Nesting with STWAVE [34]
- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [35]
- Mar 2002 ERDC/CHL CHETN-IV-40 Guidelines for Using Eastcoast 2001 Database of Tidal Constituents within Western North Atlantic Ocean, Gulf of Mexico and Caribbean Sea [36]
- Mar 2002 ERDC/CHL CHETN-II-45 Wave Transmission at Detached Breakwaters for Shoreline Response Modeling [37]

2001

- Sep 2001 ERDC/CHL CHETN-I-64 Modeling Nearshor Wave Transformation with STWAVE [34]
- Sep 2001 ERDC/CHL TR-1-25 BOUSS-2D: A Boussinesq Wave Model for Coastal Regions and Harbors [38]
- Jun 2001 ERDC/CHL CHETN-IV-32 Leaky Internal-Barrier Normal-Flow Boundaries in the ADCIRC Coastal Hydrodynamics Code [39]
- Mar 2001 Technical Report CHL-98-32 Shinnecock Inlet, New York, site Investigation Report 4, Evaluation of Flood and Ebb shoal Sediment Source Alternatives for the West of Shinnecock Interim Project, New York [40]

1999

- Dec 1999 Coastal Engineering Technical Note IV-21 Surface-Water Modeling System Tidal Constituents Toolbox for ADCIRC [41] [42]

1998

- Aug 1998 Technical Report CHL-98-xx CGWAVE: A Coastal Surface Water Wave Model of the Mild Slope Equation [43]

1990

- Mar 1990 CETN-II-21 Computer Program: Genesis Version 2 [44]

References

[1] http://adcirc.org/Related_publications.html
[2] http://shira.lawr.ucdavis.edu/publications.htm
[3] http://www.fsbpa.com/08Proceedings/07AlymovTruittPoffAnderson2008.pdf
[4] http://cirp.wes.army.mil/pubs/chetns/CHETN-IV-71.pdf
[5] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/SRH2D-v2-Training-Dec2007-Taiwan.pdf
[6] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-76.pdf
[7] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-75.pdf
[8] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-69.pdf
[9] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-73.pdf
[10] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-74.pdf
[11] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report%20Dungeness%202dmodel%20report%20May7_07.pdf
[12] http://www.waveworkshop.org/9thWaves/Papers/Smith.pdf
[13] http://www.dtic.mil/cgi-bin/GetTRDoc?AD=ADA455437
[14] http://cirp.usace.army.mil/Downloads/PDF/TR-06-9.pdf
[15] http://cirp.usace.army.mil/pubs/chetns/CHETN-III-73.pdf
[16] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/2006-April-GSTARW-Training.pdf
[17] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-71.pdf
[18] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-67.pdf
[19] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report-Elwha%20Nov%2014%202006.pdf
[20] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report%20Sandy%20River%20Report_031706.pdf
[21] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report%20Savage%20Rapids%20Jan%2023%202006.pdf
[22] http://el.erdc.usace.army.mil/elpubs/pdf/doerd4.pdf
[23] http://el.erdc.usace.army.mil/elpubs/pdf/doerd5.pdf
[24] http://el.erdc.usace.army.mil/elpubs/pdf/doerd6.pdf
[25] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-70.pdf
[26] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-69.pdf
[27] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-63.pdf
[28] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/2005-US-China-Workshop-paper.pdf
[29] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Paper%20US-China-Workshop%202005.pdf
[30] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=PUBLICATIONS!155
[31] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-68.pdf
[32] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-60.pdf
[33] http://chl.erdc.usace.army.mil/Media/4/9/5/chetn-i-67.pdf
[34] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-66.pdf
[35] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-41.pdf
[36] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-40.pdf
[37] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-ii-45.pdf
[38] http://chl.erdc.usace.army.mil/Media/5/6/9/BOUSS-2D.pdf
[39] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-32.pdf
[40] http://www.adcirc.org/publications/2001/2001_Militello03.pdf
[41] http://www.adcirc.org/publications/1999/1999_Militello.pdf

[42] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/cetn-iv-21.pdf

[43] http://champs.cecs.ucf.edu/Library/Research_Reports/pdfs/
CGWAVE,%20A%20coastal%20surface%20water%20wave%20model%20of%20the%20mild%20slope%20equation.pdf

[44] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/cetn-ii-21.pdf

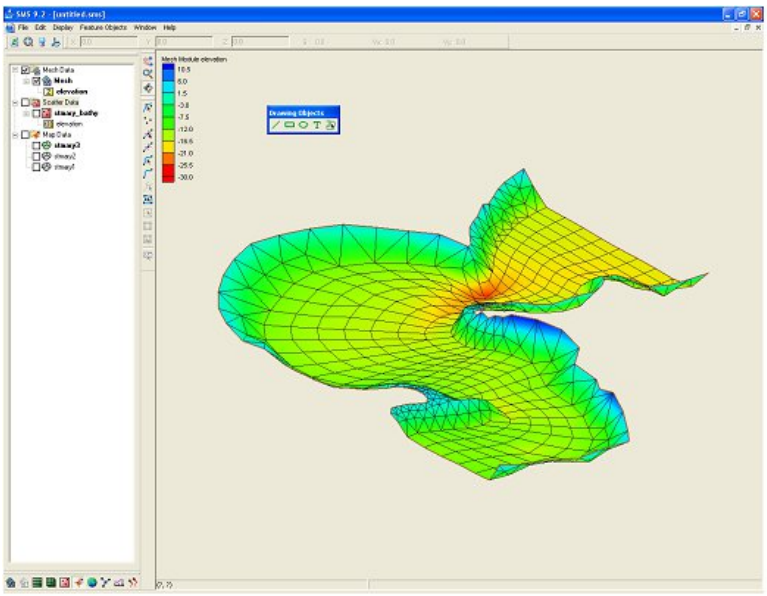
1.2. Layout

SMS Screen

The SMS screen is divided into six main sections:

1. Main Graphics Window
2. Project Explorer – this may also be referred to as the *Tree Window*
3. Toolbars
4. Edit Window
5. Menu Bar
6. Status Bars

Normally the *Main Graphics Window* fills the majority of the screen; however, plot windows can also be opened to display 2D plots of various data.



The SMS screen

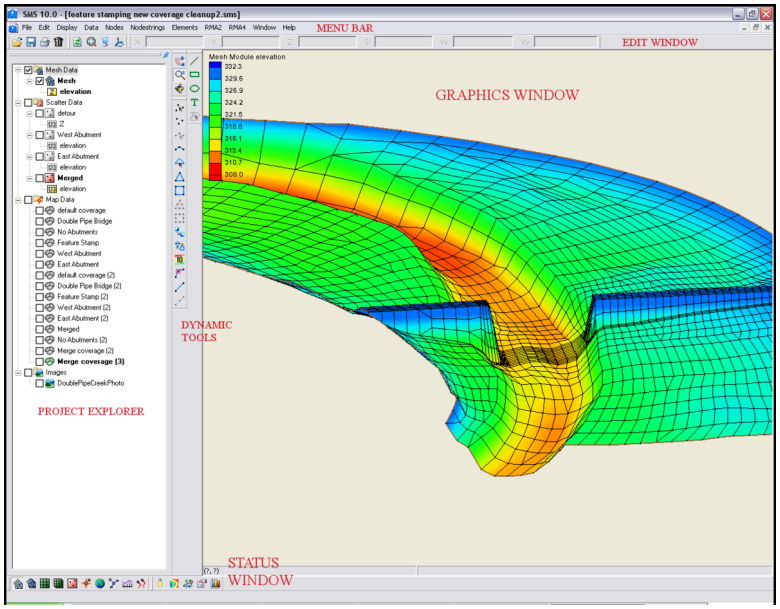
Related Topics

- Layout

Layout

At a glance

- The project explorer shows data currently loaded in project
- Menu bar depends upon the active module and model
- Edit window show x, y, z, scalar, and vector values
- Edit window values can be edited in some circumstances
- The status window on the bottom of the graphics window shows coordinates and selection information
- Help information is displayed at the bottom of the SMS screen
- Several toolbars are used in SMS. The dynamic tools change based upon the current module.



SMS Layout

The interface to SMS has been designed in a modular fashion. Separate modules are used for each data type. As the user switches from one module to another, the available menus and tools change. Inside the modules, the user associates a numeric model with a mesh or grid. When that grid is active, the tools and menus for the associated model are also enabled.

The SMS screen includes several toolbars, edit fields, and menus. Some of these change as the user switches modules or numerical models. The principal components include:

- Menu Bar – Menu commands to issue commands. These change as the module and model change.
- Edit Window – Fields directly below the menu bar showing the coordinates and function values for selected entities.
- Graphics Window – Display panel to show the data being manipulated.
- Project Explorer (Data Tree) – Tree representation of all the data currently referenced through SMS.
- Time Step Window – Appears when transient data is available.
- Toolbars – Several toolbars can be displayed. For more information on each toolbar, see the Toolbars article.
- Help or Status Window

The toolbars, project explorer, time steps window and edit window are dockable windows. Dockable windows may be positioned by the user.

Data Toolbar

The Data Toolbar contain tools to query or obtain data. Whether the Data Toolbar appears at startup is set in the Preferences Dialog. The following tools are available in the Data Toolbar:

Tool	Tool Name	Description	Right Click Menu
	Measure Tool	The **Measure Tool** is used to measure distances interactively. The units used to report the measured distance is specified on the *Toolbars* tab of the *Preferences* dialog.	N/A
	Get Data Tool	The **Get Data Tool** is used to specify the location of interest for obtaining data using the import from web feature.	N/A

Related Topics

- Layout of the Graphical Interface

Dynamic Tools

The Dynamic Toolbar contains tools that apply to the selected module and active numerical model. These tools are called dynamic because the available tools change whenever the module or numerical model is changed. These tools are used for creating and editing entities specific to the module. They appear between the Project Explorer and the Graphics Window below the Static Tools.

Selection Tools

The selection tools in SMS allow users to select entities displayed in the *Main Graphics Window*. It is necessary to first select objects before issuing many of the commands in SMS. For example, to delete a node, the node must be be selected and then the *Delete* command issued. Selections can be made using a box, polygon, arrow, or by clicking a single location. In addition, selections can be toggled, have new items added, or remove items. Below is a list of modifier keys and corresponding actions.

- **None –** This will clear the current selection and add the newly selected items. Dragging will create a selection box. All items contained in the box will be selected.
- **Ctrl –** Clicking while holding the *Ctrl* key will create a polygon. All items contained in the polygon will be selected. If control is held while dragging, an arrow will be created. All items which the arrow passes through will be selected. (*Control* will cause the same behavior with any combination of the *Alt* and *Shift* keys)
- **Shift –** Holding *Shift* causes all newly selected items to be toggled. If it was selected before it will be unselected, and if it was not selected it will be selected.
- **Alt –** Holding the *Alt* key causes all newly selected items to be added to the selection list regardless of previous state.
- **Alt + Shift –** Holding *Alt* and *Shift* causes all newly selected items to be removed from the selection list regardless of previous state.

The various selection types, polygon, arrow, and box, are available in all tools with the exception of the arrow. An arrow selection can only be performed when selecting line or polygon (e.g. mesh elements, scatter triangles, etc) elements. The arrow must cross a polygon or line edge to select it.

When selecting polygon features the rules for selection may vary slightly. In the map module all vertices of the polygon must be contained in the selection box or polygon. For mesh elements, scatter triangles, and Cartesian grid

cells only the centroid must be contained.

When clicking a single location, the element closest to the eye (i.e. drawn on top of other elements) will always be selected. All other forms of selection (box, polygon, and arrow) will select all elements meeting the required criteria.

Other commands for selecting multiple objects such as *Select With Poly*, *Select by Material Type*, and *Select by Data Value* can be found in the *Edit* menu.

Related Topics

- 2D Mesh Module Tools
- Cartesian Grid Module Tools
- Scatter Module Tools
- Map Module Tools
- GIS Module Tools
- 1D River Module Tools
- 1D Grid Module Tools
- Particle Module Tools

Edit Window

The *Edit Window* lies above the *Graphics Window* and below the *Menu Bar*. It includes a rows of edit fields and text strings. The edit fields are dim and the text strings blank if nothing is selected. When an entity, such as a mesh node, is selected, the controls display the attribute values of the selected entity. Some attribute values can be edited as shown in the table below. The attribute values are changed by typing in new values and hitting the *ENTER* or *TAB* key. If more than one entity is selected, only the Z edit field is available for editing. Entering a new value in the Z edit field will modify the bathymetry or depth of each of the selected entities. This allows the user to quickly model a feature such as a dredged channel or embankment.

Entity	X Edit Field	Y Edit Field	Z Edit Field	S Edit Field	Vx Edit Field	Vy Edit Field
Mesh Node	Editable for single selection	Editable for single selection	Editable	Editable if an ADCIRC Spatial Attributes dataset	Not editable	Not editable
Mesh Nodestring	N/A	N/A	Editable	N/A	N/A	N/A
Cartesian Grid Cell	Not editable	Not editable	Editable	Editable if an CMS-Flow hard bottom or roughness dataset	Not editable	Not editable
Cartesian Grid Cellstring	N/A	N/A	N/A	N/A	N/A	N/A
Scatter Point	Not editable	Not editable	Editable	Not editable	Not editable	Not editable
Feature Point	Editable for single selection	Editable for single selection	Editable	N/A	N/A	N/A
Feature Vertex	Editable for single selection	Editable for single selection	Editable	N/A	N/A	N/A
Feature Arc	N/A	N/A	Editable	N/A	N/A	N/A

Related Topics

- Layout of the Graphical Interface

Graphics Window

The *Main Graphics Window* is the biggest part of the SMS screen. The *Graphics Window* is where SMS displays two and three-dimensional data. It is also where the user interacts with that data in SMS. The selected tool in the determines the type of interaction that can be performed in the *Graphics Window*. For example, if the *Create Node* tool is currently selected any click in the *Graphics Window* will result in the creation of a node at the location of the click.

The user has control of what data appears in the *Graphics Window*, and how each data type is formatted. Each type of entity has an associated set of display attributes. These attributes include visibility, color, line thickness, and font type. Each data type is associated with a specific module and the attributes for that type are controlled via that modules *Display Options* dialog.

The *Graphics Window* is integral in the creation, editing and visualization of two-dimensional finite element meshes and two-dimensional finite difference grids. It is also the main means of interacting with a conceptual model and site maps.

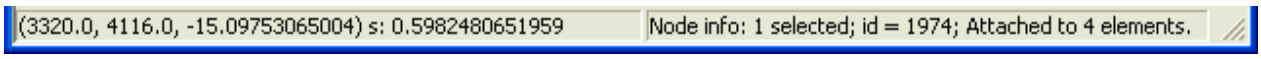
The row at the bottom of the graphics window tracks the coordinates and functional values of the location of the cursor. The z coordinate corresponds to an interpolated elevation value from either the mesh or grid, depending on which module is active.

Related Topics

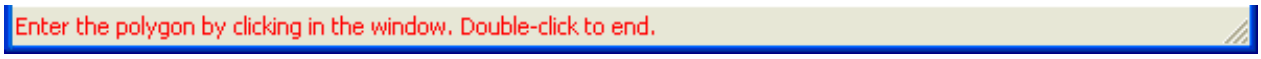
- Layout of the Graphical Interface

Help or Status Window

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



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



Status messages appear in bold red text here to inform the user on progress of a command.

Related Topics

- Layout of the Graphical Interface

Macros

The *Macro Toolbars* contain buttons to perform frequently used menu commands. All macros are shortcuts for menu commands. Which macro toolbars appear at startup is set in the Preferences Dialog. The macro toolbars include:

Optional Macro Toolbar

- **Lighting Options –** See Lighting Options.
- **Contour Options –** See Contour Options.
- **Vector Options –** See Vector Options.
- **Get Module Info –** See Get Info.
- **Plot Wizard –** See Plot Wizard.

File Toolbar

- **Open –** See Open.
- **Save Project –** See Save Project.
- **Print –** See Print.
- **Delete –** Delete the selected items. If none are selected, delete all items.

Display Toolbar

- **Refresh –** See Refresh.
- **Frame –** See Frame.
- **Display Options –** See Display Options.
- **Plan View –** See Plan View.

Related Topics

- Layout of the Graphical Interface

Menu Bar

Many commands in SMS are accessed through pull down menus located in the menu bar. Each menu can be accessed either with the mouse or by pressing the *ALT* key and the corresponding letter underlined in the menu title. Once a menu is visible the individual commands can be selected with the mouse or by again pressing the corresponding letter underlined in the menu command.

The menus available at any time are dependent on the active module and current numerical model. The first three menus, File, Edit, and Display, are always available. The remaining menus change with the module and the model. This is to partition the available commands into usable groups and avoid unnecessary complexity.

Standard Menus

- File Menu
- Edit Menu
- Display Menu
- Web Menu
- Window Menu
- Help Menu

Module Specific Menus

- 2D Mesh Module
- Cartesian Grid Module
- 3D Cartesian Grid Module
- Scatter Module
- Map Module
- GIS Module
- 1D River Module
- 1D Grid Module
- Particle Module

Model Specific Menus

- ADCIRC
- BOUSS-2D
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- Generic Model
- GenCade
- PTM
- STWAVE
- TABS
  - RMA2
  - RMA4
- TUFLOW

Related Topics

- Right-Click Menus
- Layout of the Graphical Interface
- Keyboard Shortcuts

Project Explorer

The *Project Explorer* (which could also be referred to as a "Data Tree") is a dockable window that appears by default on the left side of the SMS screen. This window displays a hierarchical tree structure representing all of the data currently being managed in an SMS simulation. The project explorer includes the following functionality:

Data Representation

The data tree includes one "Module type" folder for each type of data, including:

- Mesh 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

Each module type folder in the *Project Explorer* may contain several sub folders. For example, a simulation may includes several scattered datasets, each of which would consist of a folder inside the "Scatter Sets" folder. Further, all data associated with a specific scatter set, such as datasets of elevation or water level, are displayed as entities inside the scatter set folder. The user can create new folders and move datasets, solutions, and folders to other folders

anywhere on the *Project Explorer*. Folders can be created by right-clicking and selecting *New Folder* in the right-click menu. A dataset or folder can be deleted simply by selecting the folder and selecting the *Delete* key or by right-clicking on the item and selecting the *Delete* option in the right-click menu.

Datasets

The project explorer also includes a list of the datasets associated with each geomtric object (mesh, grid, scatter set). These are displayed below the object in the project explorer and can be arranged into folders.

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.) More Info...

Object Visibility Options

A toggle box appears to the left of each object in the project explorer. This toggle allows the display of all entities associated with the object to be turned on or off. When the toggle is turned on, only items turned on the the object's Display Options are shown.

Right-Click Menus

Right-click menus are used to interact with data in the *Project Explorer*. See the article Project Explorer Right-Click Menus for more information.

Related Topics

- Layout of the Graphical Interface

Project Explorer Right-Click Menus

The following Project Explorer mouse right-click menus are available based on where the mouse right-click is performed.

Project Explorer White Space Right-Click Menus

Right-clicking in the white space of the *Project Explorer* invokes an options menu with the following options:

- **Switch Module –** Use to change the active module (active menus and tools are based on the current module).
- **New Simulation –** Creates a new simulation for available models.
- **Convert to CAD –** Conversion of visible entities to CAD format. CAD layers are shown in a CAD Data folder in the *Project Explorer*.
- **Collapse all –** Collapses all items in the *Project Explorer*.
- **Expand all –** Expands all items in the *Project Explorer*.
- **Check all –** Checks all items in the *Project Explorer*. Checked items are displayed.
- **Uncheck all –** Unchecks all items in the *Project Explorer*. Unchecked items are not displayed.
- **Preferences –** Allows the user to set the Program Preferences in the *Preferences* dialog.

Module Right-Click Menus

General Options

The following are available for all Module Items:

- **New Folder –** Creates a new folder beneath the module item which can be used to organize datasets.
- **Delete –** Delete the module item.
- **Duplicate –** Duplicate the module item (cartesian grid, scatter set, etc.) including model parameters, boundary conditions, etc.
- **Rename –** Rename the module item.
- **Convert –** Convert the module item to another data type (e.g. Mesh → Scatter Set, Map → 2D Mesh, etc)
- **Reproject –** Reprojects the module item to a different projection.
- **Metadata –** View or modify the metadata associated with a module item.
- **Zoom To [Module Item] –** Reframe the image based on the module item extents.

Module Specific Options

See the module right-click menu article for more information on module specific right-click menus:

- 2D Mesh Module
- Cartesian Grid Module
- 3D Cartesian Grid Module
- Scatter Module
- Map Module
- GIS Module
- 1D River Module
- 1D Grid Module
- Particle Module
- CAD Data

Dataset Right-Click Menus

Right-clicking on a Dataset in the *Project Explorer* invokes an options menu with the following options:

- **Delete –** Deletes the selected dataset(s). This command may not be available for all datasets. If a dataset has been defined as an input dataset from a model parameter dialog, it must be deleted by changing the model parameter that requires the dataset as an input. It is not recommended that you delete datasets that are part of a single solution file since SMS reads these as a set from the single file.
- **Rename –** Rename the selected dataset. This is an option for datasets stored as part of the SMS project or created in the dataset toolbox. If the dataset comes from a numerical simulation solution, the name will revert to the name specified by the solution when it is read again.
- **Export –** Exports the selected dataset using the *Export Dataset* dialog.
- **Scalars to Vector –** Convert two scalar datasets to a single vector dataset. This command only appears on scalar datasets. This operation can also be accessed in the dataset toolbox.
- **Vector to Scalars –** Covert a single vector dataset into two scalar datasets (magnitude and direction or Vx and Vy). This command only appears on vector datasets. This operation can also be accessed in the dataset toolbox.
- **Dataset Contour Options –** Opens the *Dataset Contour Options* dialog.
- **Metadata –** Opens the *Dataset Metadata* dialog, used to add or view metadata associated with the project. SMS associates the specific data with the selected dataset.
- **Info –** Opens the *Dataset Info* dialog which displays characteristics of the dataset. These characteristics include statistics such as maximum, minimum, and range as well as mean and standard deviation.
- **Time Units and Reference –** For a transient dataset, the display of time values in the *Time Step Window* is controlled by the Time Settings.

Folder Right-Click Menus

- **New Folder –** Creates a new folder beneath the selected folder which can be used to organize datasets.
- **Delete –** Deletes the selected folder(s).
- **Rename –** Rename the selected folder.

Related Topics

- Menu Bar

SMS Menus

File Menu

The *File* menu is one of the standard menus available regardless of the current module and model. The *File* menu includes the following commands:

Open

The **Open** command is used to read any file used by SMS. This includes a large selection of file formats, both generic and model specific. This command opens a file browser from which one or more files can be selected. SMS attempts to recognize the file type based on the file extension. The available file formats (extenstions) varies based on the module and model being used. For example, there are several types of *.dat file that are used by different models. If a selected file does not match the anticipated type, a message is given to the user and the user may specify another format type to use to read the file. Data from the file is added to the current data base and SMS updates the display.

Save Project

The **Save Project** command is used to save an SMS Project File (file extension *.sms). The first time this command is invoked, the user is prompted for a file name (unless the user has already opened a project file into SMS). Every other time, SMS saves the project file using either the file name used to save the project or the filename of a project opened in SMS. To save a project with a new name, the **Save as...** command is used. The SMS Project File is saved as an XMDF file. The contents of the SMS Project File can be viewed using a HDF5 file browser or editor.

Save <Model>

The name of this command changes according to the active module and active model (i.e. Save RMA2 for the Mesh Module, RMA2 model). This command is similar to the **Save Project** command. The first time this command is invoked, the user is prompted for a file name (unless the user has already opened a model file into SMS). Every other time, SMS saves the model file using either the file name used to save the model or the filename of a model opened in SMS. To save a model with a new name, the **Save as...** command is used.

Save as...

The **Save as...** command allows the user to save data currently in the SMS database in a format not associated with the current model nor SMS or to save a model or project file with a new name. The user specifies the Save as type in the *Save* dialog to specify the file format. The Save as type available at any time depend on the data currently in SMS and on the current module and model (i.e. to save a map file, the user must be in the map module).

Delete All

The **Delete All** command deletes all the data associated with all modules. It resets the status of the program so that all display option and default values match the values in the "settings" file. This command should be selected when a new modeling problem is started.

View Data File

Since the process of numerical modeling often utilizes many input files and generates many output files, it is not uncommon to review an ASCII data file. When the **View Data File** command is selected, SMS asks the user to select a file. Based on the Preferences settings, the file will then open or SMS will ask which editor to open the file in. A separate process is created for editing/viewing the selected file using the selected editor. It should be remembered that this is now a separate process and the data in the file is not part of the SMS database. The data may be saved and incorporated into SMS using file read and import capabilities.

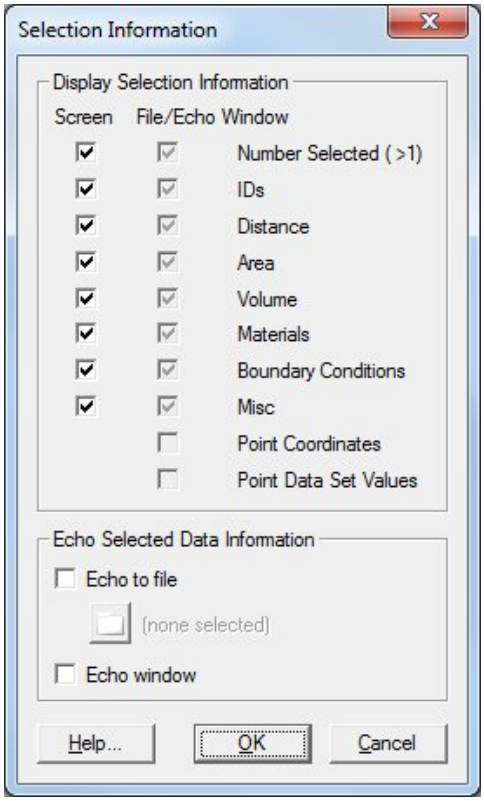
Get Info

The **Get Info** command reports basic information concerning the data type associated with the active module. For example, for meshes, the *Get Info* dialog reports the number of nodes, the number of elements, the number of linear elements, etc. For more information, see *Information* Dialog.

Info Options

When entities are selected, various information about the entities can be displayed or saved. For example, when two nodes are selected, the distance between them can be shown. The values are displayed by default in the *Status Window*. However, since space along the bottom of the window is limited, as due to the fact that the information being displayed be useful, the user has the option of displaying the information to a separate window and echoing the information into a file.

The *Selection Information* dialog allows you to turn on and off various data that can be displayed on the selected entities. The echo of the information to a file can be turned on and off with on *Echo to File* toggle. When this toggle is turned on, the user is prompted for a file name. The *Display Echo Window* option opens a window where information is also displayed.



Selection Information dialog

Save Settings

The **Save Settings** command is used to save the current settings of the program (display options, defaults, etc.) to a default settings file. SMS reads the "default settings" file each time it is launched or the new command is invoked.

Page Setup

Page Setup is used to set printing options. The dialog contains three tabs:

- **Margins**

This page allows you to set up the margins that will be used for printing to the selected printer. The right side of the display shows a gray region representing how the graphics window would be positioned on a printed page.

The scale of the printed image directly depends on the margins that are set on this page. The margins have a lower limit depending on the system's default printer. The *Maintain aspect ratio* option should usually be checked. When this option is turned on, the size of the printed image be constrained by one pair of margins, either the top and bottom or the left and right.

- **Paper Size**

This page allows you to define the size of the paper for the selected printer. The specific available options are dependent on the system's default printer. Both the paper size and paper source can be specified, as well as the image orientation on the paper.

The preview window shows a sample of what the printed image will look like.

- **Options**

This page allows you to define a scale to be added to the bottom of the printed image. If you change the scale value, the margins are updated to match. The scale is defined as either one inch or one centimeter, and will equal the specified number of units that your data are in. This could be feet, meters, lat/lon, etc. If, after the scale is set, you zoom in or zoom out, the scale will change to match the new world boundaries and the page margins.

The preview window shows a sample of what the printed image will look like.

The data displayed in the Graphics Window is then printed through the **Print** menu item.

Print

The **Print** command opens the *Windows Print* dialog. Pages are printed with the data displayed in the *Graphics Window* using the settings set through the **Page Setup** menu command.

Demo Mode

Since some users may not require all of the modules or model interfaces provided in SMS, modules and model interfaces can be licensed individually. The icons for the unlicensed modules or the menus for unlicensed model interfaces are dim and cannot be accessed. The Demo Mode command provides a way of evaluating additional modules you may wish to consider licensing in the future. This is particularly useful when using the tutorials provided with SMS.

When the **Demo Mode** command is selected, all modules of the program will be enabled. The only exceptions are that the **Print** and **Save** options will be disabled. It is important to note that when the mode is changed all current data will be deleted. When the program is in demo mode, a check mark appears next to the menu item. To return to normal operating mode, select the Demo Mode command again. If an evaluation copy of the software is being used, or if all modules are enabled, this menu item is unavailable.

Register

For more information on the **Register** command, see the article Registering SMS.

Recent Files

SMS remembers the last five files opened during operation. These files are added to the *File* Menu. A file can be reopened by choosing it from the list.

Exit

The **Exit** command is used to exit the program. If your data has not been saved, SMS warns you before it exits.

Edit Menu

The *Edit* menu is one of the standard menus and is available in all of the modules. The commands in the *Edit* menu are used to select objects, delete objects, and set basic object and material attributes.

Delete

The **Delete** command is used to delete the selected objects. This command is equivalent to hitting the *DELETE* or *BACKSPACE* keys on the keyboard. If no objects are selected when the **Delete** command is executed, then all of the objects of the tool selection type will be deleted. Unless the Confirm Deletions option is turned on, you will not be asked to confirm the deletion of selected entities.

Select All

The **Select All** command selects all items associated with the current selection tool.

Select With Poly

The **Select With Poly** command selects items associated with the current selection tool which are inside a user defined polygon. Create the polygon after selecting the command by clicking in the *Graphics Window*. The polygon is closed with a double-click. A similar feature called **Select with Feature Polygon** is available from the Map module. If a feature polygon is defined, the user can select nodes or elements in the mesh module or vertices in the data module that are inside or outside of the feature polygon.

Select By

Select By brings up a submenu with the following options:

- **Material Type**

  The *Select By*|**Material Type...** command selects all items of the current selection tool of a specified material. This command opens a dialog with a list of the defined materials and waits for the user to select a material type. This enables all nodes or elements that reference a specific material to be selected together.

- **Dataset Value**

  The *Select By*|**Dataset Value...** command opens a dialog to get a range from the user. All entities (nodes, elements, scatter points, etc.) of the current selection tool type whose scalar dataset value lies inside that range are selected. This enables all entities above or below threshold to be selected together for quick editing.

- **Area**

  The *Select*|**By Area...** command opens a dialog to get a range from the user. All polygons whose area lies inside that range are selected. This enables all entities above or below threshold to be selected together for quick editing.

- **Length**

  The *Select By*|**Length...** command opens a dialog to get a range from the user. All arcs whose length lies inside that range are selected. This enables all entities above or below threshold to be selected together for quick editing.

Confirm Deletions

By default, whenever a set of selected objects is about to be deleted, the user is prompted to confirm the deletion. This helps ensure that objects are not deleted accidentally. Selecting the **Confirm Deletions** command toggles this request for confirmation. When the option is off, the check mark next to the **Confirm Deletions** line in the menu disappears.

Current Coordinates

This command allows the user to tell SMS what coordinate system the data is to reference. SMS supports several different global systems as well as a user defined local system.

Coordinate Conversions

The user converts the current data from on coordinate system to another. For more information, see the Coordinate Conversions article.

Single Point Conversion

This command opens a dialog which acts as a stand-alone coordinate converter. The user specifies a to and from coordinate system and a location. The location is converted to the new system within the dialog.

Projection...

Brings up the *Current Projection* dialog. See Projections for more information.

Reproject...

Reprojecting means to convert data from one coordinate system to another. See Reproject for more information.

Single Point Projection...

Single Point Projection allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection. See Projections for more information.

Time Settings...

This command opens the *Time Settings* dialog. For more information, see the Time Settings article.

Materials Data

See the Materials Data article.

Project Metadata

This command allows the user to define metadata for the project. This documents a history of the project.

Copy to Clipboard

Copies the contents of the graphics window to the windows clipboard. This allows graphics to be easily transferred to documents and presentations.

Paste

Opens the Import Wizard with the contents of the windows clipboard. This requires that the contents be text values This allows graphics to be easily transferred to documents and presentations.

Preferences

This command allows the user to set program preferences. For more information, see the article Preferences.

Display Menu

The *Display* menu is the third standard menu available in all modules. The commands in the *Display* menu are used to control what entities are displayed and the attributes of those entities. The commands include:

Display Options

See the Display Options article.

Lighting Options

See the Lighting Options article.

Refresh

When editing the image in the *Graphics Window* it occasionally becomes necessary to refresh the screen by redrawing the image. By default, SMS automatically updates the display when it is required (see **Automatic Refresh** below). To force the display to update, select the **Refresh** command from the *Display* menu or click the **Refresh** button. The process of redrawing can be aborted by pressing the *ESC* key.

Frame Image

Selecting the *Frame Image* command centers displayed data. This command adjusts the window boundaries so that all visible objects fit in the *Graphics Window*.

View

The view command brings up a sub menu. Items in the *View* submenu include:

- **View Angle –** set the bearing and dip of the look from direction. The bearing and dip values correspond to a rotation about the z and x axes. The bearing affects the horizontal angle (rotating the object in the xy plane), and the dip changes the vertical angle (shifting the viewing angle on the object to a higher or lower perspective). The object cannot be tilted sideways. Using only two viewing angles rather than three limits the viewing angles, but it is simpler and more intuitive.Plan view is a bearing of 0 degrees and a dip of 90 degrees.
- **Window Bounds –** The numerical model resides in a virtual world. The extents of that world displayed in the *Graphics Window* are the window boundaries. These boundaries can be altered using the Pan and Zoom tools. Alternatively, it is possible to precisely control the visible region by using the Set Window Boundaries command.

The Set Window Boundaries dialog box appears, and the x and y limits of the viewing area can be set.

- **Plan –** Change the view in the *Graphics Window* to a plan view.
- **Front –** Change the view in the *Graphics Window* to a front view.
- **Oblique –** Change the view in the *Graphics Window* to a oblique view.
- **Side –** Change the view in the *Graphics Window* to a side view.
- **Previous –** Change the view to the previous view (the view before zooming and framing).

Plot Wizard

Opens the plot wizard. Details of how plots are generated and controlled are defined in the visualization tools. See the Plot Window article.

Plot Data

Edit the data plotted in the active plot. Make a plot active by clicking on it.

Plot Display Options

Change the display options for the active plot. Make a plot active by clicking on it.

Projection...

Brings up the *Current Projection* dialog. See Projections for more information.

Reproject...

Reprojecting means to convert data from one coordinate system to another. See Reproject for more information.

Single Point Projection...

Single Point Projection allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection. See Projections for more information.

Web Menu

The *Web* menu is one of the standard menus available regardless of the current module and model. The *Web* menu includes the following commands:

Import from Web...

Opens the web services utility which allows for the automated download and import of certain data types from the internet.

Add Online Map

This command will bring up the *Get Online Maps* dialogue allowing selection of various online data. See Get Online Maps article for details.

Find Data

This includes options to open to the Geo-Spatial Data Acquisition page on the XMS wiki. This page provides links to websites which provide various types of data which can be used in SMS.

Tidal Data

- NOAA Hourly Verified...
- NOAA 6-Minute Raw...

Window Menu

The *Window* Menu is one of the standard menus available regardless of the current module and model. The *Window* Menu includes the following commands:

Cascade

Arranges all windows in an overlapping fashion within the SMS *Graphics window*.

Tile

Arranges all windows as non-overlapping vertical tiles within the SMS *Graphics window*.

Tile Horizontally

Arranges all windows non-overlapping horizontal tiles within the SMS *Graphics window*.

Active Window

A list of the currently open graphics and plot windows is shown at the bottom of the *Window* Menu. A check mark appears in front of the active window. Choose a window from the list to make it active.

Help Menu

The *Help* Menu is one of the standard menus available regardless of the current module and model. The *Help* Menu includes the following commands:

Help

Launches the Help File.

About

Displays the Version, Build Date, Contact Information, etc.

Right-Click Menus

Many commands in SMS are accessed through mouse right-click menus. The mouse right-click menu available at any time is dependent on the active module, current numerical model, active tool, and where the right-click is performed.

Standard Right-Click Menus

- Plot Window
- Project Explorer
- Time Step Window

Module / Model Specific Right-Click Menus

Module and model specific right-click menus are documented for each individual tool. See the Dynamic Tools Article for more information.

Related Topics

Menu Bar

Static Tools

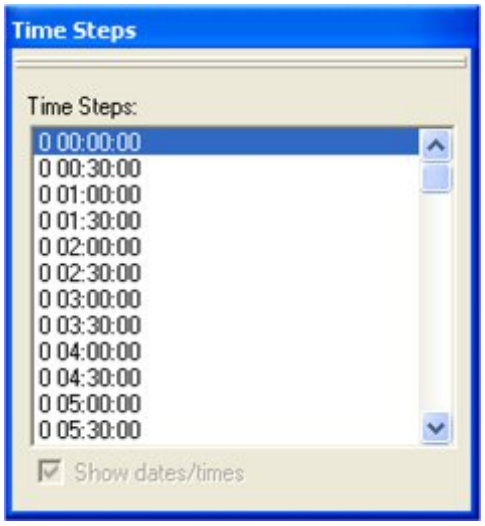
The *Static Toolbar* contains tools which are available in every module. These tools are tools for basic operations such as panning and zooming. Only one tool is active at any given time. The action that takes place when the user clicks in the *Graphics Window* depends on the current tool. The following table describes the tools in the *Static Tool* palette.

Tool	Tool Name	Description
	Pan	The **Pan** tool is used to pan the viewing area of the *Graphics Window*. Panning can be done in 3 ways: • When the **Pan** tool is active, holding down the main mouse button while dragging moves the view. • If another tool is active and you don't want to switch tools, you can pan by holding down the *F2* key and clicking and dragging with the mouse. • If your mouse has a middle button (or a mouse wheel), you can hold it down and drag to pan the view.
	Zoom	The viewing area can be magnified/shrunk using the **Zoom** tool. Zooming can be done in the following ways: • With the **zoom** tool selected, clicking on the screen zooms the display in around the point by a factor of two. Holding down the *SHIFT* key zooms out. • With the zoom tool selected, a rectangle can be dragged around a portion of the display to zoom in on that region. Holding down the *SHIFT* key zooms out. • If another tool is active and you don't want to switch tools, you can zoom by holding down the *F3* key and clicking and dragging with the mouse. • If your mouse has a middle button (or a mouse wheel), you can scroll the wheel to zoom in and out.
	Rotate	The **Rotate** tool provides a quick way to rotate the viewing location. Rotating can be done in the following ways: • With the **rotate** tool selected, holding down the mouse button and dragging the cursor in the *Graphics Window* rotates the object in the direction specified. A horizontal movement rotates the image about the z axis. A vertical movement rotates the image about the x and y axis. The amount of rotation depends on the distance the cursor moves while the mouse button is down. • If another tool is active and you don't want to switch tools, you can rotate by holding down the *F4* key and clicking and dragging with the mouse. • The viewing angle can also be entered directly through the Display Options Dialog (General Options, View Tab).

Related Topics

- Layout of the Graphical Interface

Time Step Window



The *Time Steps Window* is used to select a time step to be active and is only visible if a transient dataset has been loaded into the project.

The *Time Step Window* is located below the Project Explorer by default, but it can be moved to anywhere on the window since it is a "dockable" toolbar. The *Time Step Window* can be resized by clicking on the window borders and dragging them. The *Time Step Window* only appears when a transient dataset is selected in the *Project Explorer*.

The display of time values in the *Time Step Window* is controlled by the settings in the *Time Settings* and *Preferences* dialogs.

Time Step Window Right-Click Menu

Right-clicking on the *Time Step Window* will give bring up the following options:

- Time Settings – Allows the user to change the how time is displayed.
- Time Preferences – Opens the *SMS Preferences* dialog.

Related Topics

- Layout of the Graphical Interface

Toolbars

There are several toolbars that can be displayed.

Macros

Many of the more frequently used menu commands can be accessed through the macro buttons. These buttons essentially serve as shortcuts to menu commands. *Macro Toolbars* include:

- Optional Macro Toolbar
- File Toolbar
- Display Toolbar

For more information, see the Macros article.

Static Toolbar

The *Static Toolbar* contains the tools which are available in every module. These tools are tools for basic operations such as panning and zooming. 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. For more information, see the Static Tools article.

Dynamic Toolbar

When the active module is changed, the tools in the *Dynamic Tool Palette* change to the set of tools associated with the selected object/module. Each module has a separate set of tools. For more information, see the Dynamic Tools article.

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.

See Modules for more Information.

Data Toolbar

The *Data Toolbar* is used to query objects displayed in the graphics window. For more information, see the Data Toolbar article.

Related Links

- Layout of the Graphical Interface

1.3. Support

Support

Support for the current version of SMS is provided by Aquaveo. For contact information please go to Aquaveo Technical Support [1].

Support Forum

For news, updates or to post questions and participate in discussion topics for GMS, SMS, and WMS visit the Aquaveo support forum [2]. A weekly email summary can be requested by forum subscribers.

Related Topics

- GMS [3]
- SMS [4]
- WMS [5]

External Links

- Aquaveo Technical Support [6]

References

[1] http://www.aquaveo.com/technical-support
[2] http://forum.aquaveo.com/
[3] http://www.aquaveo.com/gms/
[4] http://www.aquaveo.com/sms/
[5] http://www.aquaveo.com/wms/
[6] http://www.aquaveo.com/technical-support/

Downloads

By going to the Download Center at www.aquaveo.com [3], user will find updates for SMS, tutorials, release notes, beta versions, and more.

Directions to the Aquaveo Download Page

To access the online SMS Downloads page follow these steps:

1. Go to www.aquaveo.com [3]
2. Click on the *Support* menu at the top of the page.
3. Select **Downloads** from the menu options. This will bring up the Aquaveo download page.
4. Select the *SMS* tab to find SMS materials available for download.

Aquaveo Download Page

The following link goes directly to the Aquaveo Download page:

- Aquaveo download page [1]

Related Topics

- Installing and Setting up SMS
- System Requirements
- License Agreement

References

[1] http://www.aquaveo.com/downloads

FTP Site Info

To upload files:

1. Zip up the files to upload.
2. Go to this link:

   | Upload Link [1]
3. Paste files in window.

References

[1] ftp://pubftp.ems-i.com/upload/

License Agreement

AQUAVEO SOFTWARE LICENSE AGREEMENT

We understand that neither Aquaveo LLC nor its employees makes any warranty express or implied, or assumes any legal responsibility for the accuracy, completeness, or usefulness of the computer programs and documents herein ordered. We acknowledge that Aquaveo's liability for damages associated with these programs is limited to the amounts paid to Aquaveo. We understand that we may register the executable on only one machine per license purchased. We also agree to not make the program available to more persons, at any given time, than the number of licenses purchased. We agree to not distribute this program in unmodified or modified form, outside our organization without the written permission of Aquaveo.

License agreements for software used by Aquaveo software

FTGL LICENSE AGREEMENT

http://ftgl.wiki.sourceforge.net/states: "FTGL is free software. You may use it, modify it and redistribute it under the terms of the MIT license or the GNU LGPL, at your option."

The MIT license link is to http://en.wikipedia.org/wiki/MIT_License, which states:

Copyright (c) <year> <copyright holders> Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Registering SMS

After installing SMS, it will need to be registered. Registration can be done with a password or with a hardware lock. When SMS is first launched, a dialogue box appears that has two options. The first button, **Demo Mode**, allows you to run SMS in demo mode. The second, **Enable** is used to enable the program. This is described below.

Password

If you are using a password to enable SMS, you will need to send information to your vendor about your machine to get your password. There are several ways you can send this information.

Register SMS with a Password [1]

1. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
2. Select *License code* for the Licensing method and enter the 7 digit alpha-numeric code that begins with the letter P. Click the **Next >** button.
3. If the registration is successful, click **Finish** to exit the *Registration Wizard*.
4. The *Register SMS* dialog displays the registered components, licensing method, and license expiration dates.

Hardware Locks

Follow the instructions you received with the hardware lock to install the hardware lock and accompanying drivers. If you did not receive hardware lock instructions, or they have been misplaced, they can be found in the \Utils\Hwlock\Instructions directory on the CD. There are separate files for single user and network hardware locks. These files can be read using your web browser.

Demo Mode

If no valid license is detected, SMS runs in Demo Mode. All features of the software are enabled except printing, saving, and running models. This mode is intended to allow users to evaluate the software before making a purchase. Datasets, grids, or meshes can be read in, manipulated and viewed.

Evaluation Version

An evaluation version that is valid for 30-60 days may be requested by selecting the Evaluation button. You will be connected to a web utility, and a valid registration code will be sent to you via email. Once you receive the registration code, enter it into the dialogue box, and select register. Once you have evaluated SMS, please contact your vendor with any questions or to purchase.

Related Topics

- SMS System Requirements
- Hardware Locks

References

[1] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_with_a_Password_I.pdf

Hardware Locks

Single User USB Hardware Lock [1]

1. Install the hardware lock drivers. If hardware lock drivers have already been installed, skip to the next step. The drivers can be installed by running the **Sentinel System Driver Installer.exe** program found in the SMS installation directory.
2. After installing the hardware lock drivers, plug the Aquaveo hardware lock into an available USB port.
3. Start SMS and the program should automatically detect the hardware lock. If the Welcome dialog appears, click the **Register…** button.
4. Select **Hardware lock** for the Licensing method and click the **Next >** button.
5. In the Hardware lock options, select **Get license from a single user lock** and click the **Next >** button.
6. If the registration is successful, click **Finish** to exit the *Registration Wizard*.
7. The *Register SMS* dialog displays the registered components, licensing method, and license expiration dates.

Update a Single User USB Hardware Lock [2]

1. Plug the Aquaveo hardware lock into a computer with hardware lock drivers and SMS installed.
2. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
3. Select **Hardware lock** for the Licensing method and click the **Next >** button.
4. In the *Hardware lock* options, select **Modify lock on this computer with the following code** and click the **Next >** button.
5. In the *Hardware Lock* dialog, click the **Next >** button to burn the hardware lock.
6. If the registration is successful, click **Finish** to exit the *Registration Wizard*.
7. The *Register SMS* dialog indicates the registered components, the licensing method, and the license expiration dates.

Setup a Network License Server [3]

1. Install the Sentinel installation program that includes hardware lock drivers and the Sentinel Protection Server software. The installation program can be downloaded at www.aquaveo.com/downloads [1].
2. In the Sentinel installation wizard, select "Complete" for the setup type.
3. After installing the Sentinel lock drivers and server software, plug the Aquaveo hardware lock into an available USB port.
4. Ensure that the computer with the network hardware lock can be seen by other computers on the local network. Client machines can connect to the server by hostname or IP address.
5. The License Server is now ready to provide SMS licenses to client machines. Refer to the instructions for registering SMS with a Network USB Hardware Lock for more information.

Network USB Hardware Lock [4]

1. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
2. Select **Hardware lock** for the Licensing method and click the **Next >** button.
3. In the *Hardware lock* options, select **Get license from a network lock** and click the **Next >** button.
4. Enter the *IP address* or *Host name* of the server hosting the network hardware lock.
5. Click the **Browse Lock Setting...** button. This opens a web browser and tests the a connection to the hardware lock over a local network.
6. Click the **Apply Lock Setting...** button.
7. Once the "Lock license acquired" message appears, click the **Finish** button.
8. The *Register SMS* dialog displays the registered components, licensing method, hardware lock serial number, and license expiration dates.

Update a Network USB Hardware Lock [5]

1. Plug the Aquaveo hardware lock into a computer with hardware lock drivers and SMS installed.
2. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
3. Select **Hardware lock** for the Licensing method and click the **Next >** button.
4. In the *Hardware lock* options, select **Modify lock on this computer with the following code** and click the **Next >** button.
5. In the *Hardware Lock* dialog, enter the number of licenses to burn on the lock. This number is typically the same as the number of licenses purchased and available. Click the **Next >** button to burn the hardware lock.
6. If the registration is successful, click **Finish** to exit the *Registration Wizard*. Please note that the *Registration* dialog may not show the enabled components. To verify the enabled components on the network lock, refer to the instructions for registering SMS with a Network Lock.
7. Return the network hardware lock to the computer serving as the license server if necessary.

Related Topics

- Registering SMS

References

[1] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_with_a_SUHWL_I.pdf
[2] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_Burn_SUHWL_I.pdf
[3] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_NW_License_Server_Setup_I.pdf
[4] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_with_a_NWHWL_Client_I.pdf
[5] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_Burn_NWHWL_I.pdf

Graphics Card Troubleshooting

XMS (WMS, GMS, or SMS) use OpenGL for rendering graphics. OpenGL is a graphics standard, but each implementation is maintained by individual graphics card companies. Different graphics cards and drivers support different versions of the OpenGL standard. XMS currently uses features up to version 1.5 of OpenGL (as of April 2009 version 3.1 was most recent version).

Some graphics cards, as well as remote desktop, do not support functionality through OpenGL version 1.5. This is mostly a problem with older integrated graphics cards, in particular those manufactured by Intel. This page will give you some ideas on troubleshooting these problems. The best solution is to get a graphics card that supports later versions of OpenGL. You will see improved performance as well as be able to access all the features of XMS.

Remote Desktop

XMS (WMS, GMS, or SMS) will have reduced capability when running remote desktop.

Since remote desktop only supports OpenGL version 1.1 not all of the features of XMS may be available.

1. One solution is to use a different remote control software that utilizes the graphics card of the computer you are controlling. www.logmein.com [1] has free and paid versions of remote desktop that behave better with XMS. RealVNC is a program that does this and can be purchased at a reasonable cost. There is a free version but it has not been tested with the XMS software. See VNC Homepage [2] for more information.
2. Another solution is to use the Mesa software rendering option available in the application's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

Parallels Desktop for Mac

XMS has reduced capability when running in a pure virtual PC through Parallels Desktop for Mac. Although Parallels version 6.0 provides OpenGL version 2.1 support (instead of OpenGL version 1.1) when "Enable 3D acceleration" is selected in the virtual machine's hardware configuration, the Parallels virtual video card adapter does not render all XMS graphics correctly. The solution is to use the Mesa software rendering option available in XMS's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

If you are running XMS in a virtual PC utilizing a Boot Camp partition then Parallels uses the actual graphics card installed in the Mac. See sections below regarding graphics card issues.

OpenGL Graphics Dialogs

XMS (post WMS 8.2, GMS 7.0 onward, and SMS 10.1 onward) have dialogs that allow the selection of OpenGL support. The choice is between the system default library and the Mesa software library. The system default can change based upon current conditions such as a remote login. Not all system defaults support all needed graphics functionality. Therefore Mesa is provided for better functionality at a potential reduction in speed. However, Mesa may produce poor images when printing. The user can make this tradeoff in the graphics dialog found in preferences. The dialog provides 4 options so that on subsequent runs XMS will:

1. Ask which graphics library to use if the system does not support all OpenGL functionality needed by XMS. This option is initially set and gives the following options:
    1. Autoselect the Mesa software library for this run if the system default does not support all functionality. XMS will not prompt on subsequent runs. It will just check support and select a library.
    2. Use the system default library on this run (and on future runs if the "Do not ask again box" is checked).
    3. Use the Mesa software library on this run (and on future runs if the "Do not ask again box" is checked).
2. Autoselect the Mesa software library if the system default does not support all functionality.

3. Always use the system default library.
4. Always use the Mesa software library.

Determining Graphics Card Manufacturer

Always download and install the latest drivers from your graphics card vendor. Graphics card problems are often due to using the wrong or outdated drivers. You can use a simple diagnostic program called dxdiag [3] to determine your computer's hardware, operating system, and graphics card. To use the dxdiag [3] program:

1. Select **Start**
2. Choose **Run.**
3. Type "dxdiag" in the box and click *OK*.'
4. Click **Yes** to the prompt, and the program will begin running.
5. Select the *Display* tab and the Name listed under the "Device" section is the name of your graphics card.

You can also:

1. Right-click on the desktop and select **Properties**
2. In the *Display Properties* dialog, click on the *Settings* tab
3. Your video card manufacturer and chipset is shown below the "Display:" line
4. Look for the names NVIDIA, ATI, Intel, Matrox, SiS, S3, etc.

Updating Laptop Graphics Card Drivers

If you have a laptop, visit the laptop manufacturer's website (Dell [4], HP or Compaq [5], Toshiba [6], Sony [7], etc.) to get the most recent driver.

Updating Desktop Graphics Card Drivers

If you are using a desktop computer, visit the graphics card manufacturer's website to download the latest driver. Listed below are a few common graphics cards and links to download their drivers:

- 3DLabs [8]
- ATI [9]
- Diamond [10]
- Elsa [11]
- Intel [12]
- Matrox [13]
- nVidia [14]
- S3 [15] – Not all S3 card support OpenGL 1.5 which is required for all display options to be enabled.
- SIS [16] – Not all SIS card support OpenGL 1.5 which is required for all display options to be enabled.
- VIA [17] – Not all VIA card support OpenGL 1.5 which is required for all display options to be enabled.

Updating Windows Operating System

Many problems are resolved by keeping the windows operating system and hardware drivers up to date using the windows update site [18]. Hardware updates are often only installed if the "Custom" or "Optional" updates are included.

Updating XMS Software

Many problems are resolved by installing the latest version of XMS. Bugfixes and updates are released frequently. The updates can be downloaded at the Aquaveo Download Center [1].

Known Graphics Issues

- Issue: Graphic symbols are not displayed correctly and sometimes corrupt text lines located next to them.

  Hardware: Make: ATI Technologies Inc. Model: RADEON X600 PRO (0x5B62) Name: ATI Radeon X300/X550/X1050 Series

  Solution: Updating the driver will allow the symbols to display correctly, but the text corruption still remains.

Switch from Hardware to Software Rendering

THE FOLLOWING SHOULD BE ATTEMPTED ONLY IF THE OTHER SOLUTIONS PRESENTED DO NOT RESOLVE THE DISPLAY ISSUES

If you have updated your graphics driver and are still having problems, you can download this opengl32.dll ZIP file [19] and unzip the "OpenGL32.dll" and the "Glu32.dll" file to the directory where XMS is installed. Close and re-open XMS so this DLL is used for displaying XMS objects. Placing these DLL's in your XMS directory will fix most graphics-related issues, such as problems with displaying triangles on large TIN or DTM datasets and other problems with displaying large amounts of data. The following are known disadvantages to using this DLL for displaying:

- Displaying graphics using this DLL will likely be slower since software is used to display your graphics instead of your computer's graphics hardware. Panning, zooming, and rotating operations will be significantly slower.
- Some entities, such as symbols, are currently not displayed correctly when using this DLL. Only squares and circles will be displayed. Changing all symbol display options to squares or symbols will allow you to work around this problem. We are currently working on trying to fix this problem of symbols not displaying when using this DLL. (THIS PROBLEM HAS NOW BEEN FIXED IN SOME BETA VERSIONS OF XMS COMPILED AFTER March 31, 2009) In general, you will not want to use this DLL unless you are working with large datasets that have display issues where XMS closes unexpectedly.

Contacting Support

If you continue to experience problems after updating your graphics card drivers, contact support [6].

External Links

- Aquaveo Technical Support [6]

References

[1] http://www.logmein.com/
[2] http://www.realvnc.com/
[3] http://en.wikipedia.org/wiki/DxDiag
[4] http://www.dell.com/
[5] http://welcome.hp.com/country/us/en/support.html
[6] http://www.toshiba.com
[7] http://www.sony.com
[8] http://www.3dlabs.com/support/drivers/
[9] http://ati.amd.com/support/driver.html
[10] http://www.diamondmm.com/
[11] http://www.elsa.com/supports/download.asp
[12] http://support.intel.com/support/graphics
[13] http://www.matrox.com/mga/support/drivers/latest/home.cfm
[14] http://www.nvidia.com/content/drivers/drivers.asp
[15] http://www.s3graphics.com/drivers.jsp
[16] http://www.sis.com/support/support_prodid.htm
[17] http://www.viaarena.com/default.aspx?PageID=2
[18] http://update.microsoft.com
[19] http://wms.aquaveo.com/OPENGL32.zip

Introduction to Setting up SMS

Installing

The installation wizard will guide you through the installation process. You will have the option to install different parts of the SMS program including the executable files, tutorial files, documentation files, etc. If you are missing a part of the installation, reinstall and verify that all parts are selected to be installed.

Registering

See Registering SMS for information regarding registering SMS using a hardware lock or password.

Program Defaults

When you start SMS there are default values set for directories, display options, etc. Each project also saves settings associated with the project. As a project is read, the values are set to the project settings. The default settings only appear when you are creating new projects. You can modify the default settings for a new project by choosing Save Settings from the File menu. This will replace the default settings with your current settings.

Frequently Asked Questions

Q: My hardware lock doesn't work.

A: Visit the SMS:Hardware Locks page for Hardware lock troubleshooting. Review the hardware lock troubleshooting guides: Single User Locks [1] and Network Locks [2].

Q: I encountered an error when trying to install SMS.

A: The most common cause for installation errors is running the installation program without "Administrator" priviledges. Also check that the installation diectory is a valid location and that it isn't "read-only."

Q: Where can I get the latest build updates of SMS?

A: See Downloads or visit the Aquaveo download page [1].

Q: What is "Demo Mode?"

A: SMS runs in Demo Mode if a valid license is not present. In Demo Mode, printing and saving are disabled.

Q: I purchased SMS. How do I enable the software and get out of Demo Mode?

A: SMS can be enabled with a password or hardware lock. To obtain a password or hardware lock, contact your software vendor. Passwords enable a single version of SMS on a single machine. Passwords are machine specific. When obtaining a password, you will need to provide your vendor with your computer's register string. The register string is listed in the Register dialog (File | Register.) Hardware locks enable a roaming license of SMS. In order to enable SMS with a hardware lock, the lock must be attached to the machine when running SMS. See Registering SMS for more information.

Q: In the graphics window, letters and numbers appear instead of points and nodes. How can that be fixed?

A: If letters and numbers appear in the graphics window instead of regular points and nodes, then there was an error installing the SMS font or the font was corrupted. Usually this problem clears up when the computer is restarted after the SMS installation process is finished.

Q: My password doesn't work.

A: Passwords are not case sensitive but register strings **ARE** case sensitive. Double check the password data sent to you by your vendor and make sure the register is correct and you have entered the correct password.

Q: What does the error "This application has failed to start because MSVCR71.dll was not found" mean?

A: Your machine is missing the file MSVCR71.dll. This file should have been installed on your computer with the Windows Operating System. To fix this, download and reinstall MSVCR71.dll. MSVCR71.dll is available for download on many websites. Use a search engine, such as Google, to find it.

Related Topics

- Downloads
- System Requirements
- License Agreement

References

[1] http://ems-i.com/Support/single_user_hardware_locks.html
[2] http://ems-i.com/Support/network_hardware_locks.html

System Requirements

System requirements for GMS, SMS and WMS.

Windows 7[1]

Windows 7 is supported in GMS 7.0, SMS 10.1, WMS 8.3 and greater versions only.

Component	Minimum Required	Recommended
RAM	1 GB	4 GB or greater
CPU	XMS software is CPU intensive. We recommend the fastest CPU your budget allows.	
Hard Disk Free Space	300 MB	300 MB or greater
Graphics Card	**For all display features to be enabled, OpenGL 1.5 must be supported.**	The use of a dedicated graphics card is strongly recommended. Integrated graphics cards are often problematic.
Minimum Resolution	1024x768	1024x768 or greater

Windows Vista[2]

Windows Vista is supported in GMS 7.0, SMS 10.0, WMS 8.1 and greater versions only.

Component	Minimum Required	Recommended
RAM	1 GB	4 GB or greater
CPU	XMS software is CPU intensive. We recommend the fastest CPU your budget allows.	
Hard Disk Free Space	300 MB	300 MB or greater
Graphics Card	**For all display features to be enabled, OpenGL 1.5 must be supported.**	The use of a dedicated graphics card is strongly recommended. Integrated graphics cards are often problematic.
Minimum Resolution	1024x768	1024x768 or greater

Windows XP

Windows XP is recommended for GMS 7.0, SMS 10.0, WMS 8.1 and greater versions.

Component	Minimum Required	Recommended
RAM	512 MB	2 GB or greater
CPU	XMS software is CPU intensive. We recommend the fastest CPU your budget allows.	
Hard Disk Free Space	300 MB	300 MB or greater
Graphics Card	**For all display features to be enabled, OpenGL 1.5 must be supported.**	The use of a dedicated graphics card is strongly recommended. Integrated graphics cards are often problematic.
Minimum Resolution	1024x768	1024x768 or greater

[1] Windows 7 is supported in GMS 7.0, SMS 10.1 and greater versions only.
[2] Windows Vista is supported in GMS 7.0, SMS 10.0, WMS 8.1 and greater versions only.

Notes

- You may have display problems when running over remote desktop. This can usually be fixed by restarting the software after beginning/ending a remote desktop session. Remote Desktop cannot be used with single-user/standalone locks, only with network locks.
- Always download and install the latest drivers from your graphics card vendor. Graphics card problems are often due to using the wrong or outdated drivers. See Graphics Card Troubleshooting for instructions on how to download and install graphics card drivers. If you continue to experience problems after updating your graphics card drivers, contact support (http://www.aquaveo.com/technical-support/).

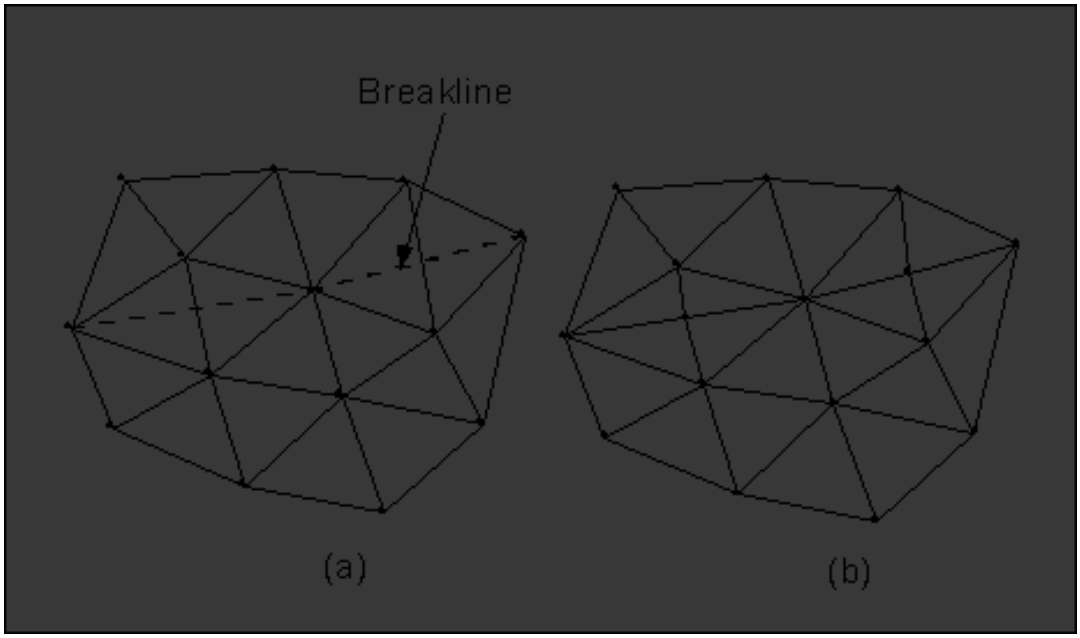
External Links

- Description of the DirectX Diagnostic Tool (http://support.microsoft.com/kb/190900)

2. Functionalities

Breaklines

A breakline is a feature or polyline representing a ridge, thalweg, or other shape that the user wishes to preserve in a surface made up of triangular elements or scatter set. In other words, a breakline is a series of edges to which the mesh or scatter triangles should conform to, i.e., not intersect.



Breaklines (a) Original Triangulation and Breakline. (b) Triangulation After the Breakline has been Processed by inserting new mesh nodes along the breakline.

Mesh Module Breaklines

Breaklines are processed using the **Force Breaklines** command from the *Nodestrings* menu. How breaklines are processed is controlled by the breakline options in the *Nodestring Options* Dialog.

Scatter Module Breaklines

Breaklines are processed using the **Force Breaklines** command from the *Breaklines* menu. Scatter breaklines are always processed by swapping triangle edges to ensure that the edges of the triangles will conform to the breakline.

Importing Scatter Breaklines

Scatter Breaklines can be imported along with scatter data using the *File Import Wizard*. In order to import breaklines, the tabular file must be prepared in one of two supported formats. They are illustrated below. In either case, an additional column of data defines the breakline information. In the import wizard, this column should be mapped as "Breakline". This brings up the *Scatter Breakline Options* Dialog

Note: Scatter breaklines must be imported at the same time as their corresponding scatter vertices.

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
```

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

- Editing 2D Meshes

Object Info

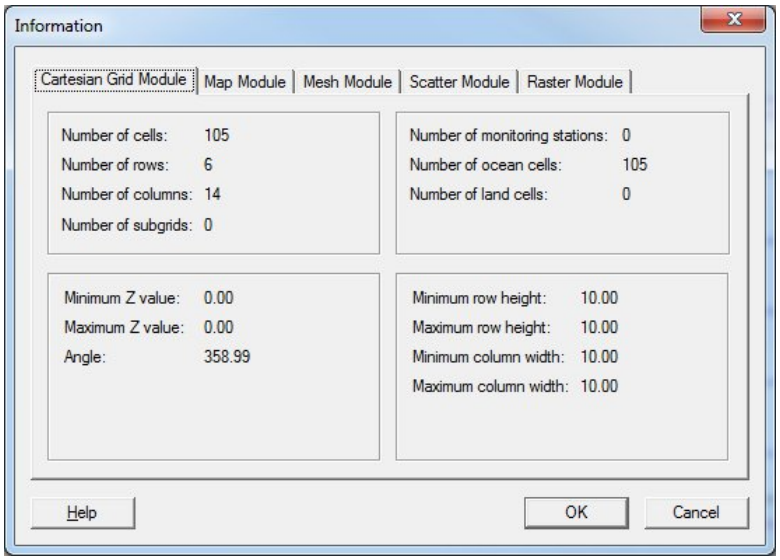
Information Dialog

The **Get Info** command reports basic information concerning the data type associated with the active module. Information is available for the following modules:

Cartesian Grid Module Information

The following information is shown on the *Cartesian Grid Module* tab of the *Information* dialog:

- Number of cells
- Number of rows
- Number of columns
- Minimum Z value
- Maximum Z value
- Angle
- Cell size
- Number of monitoring stations
- Number of ocean cells
- Number of land cells

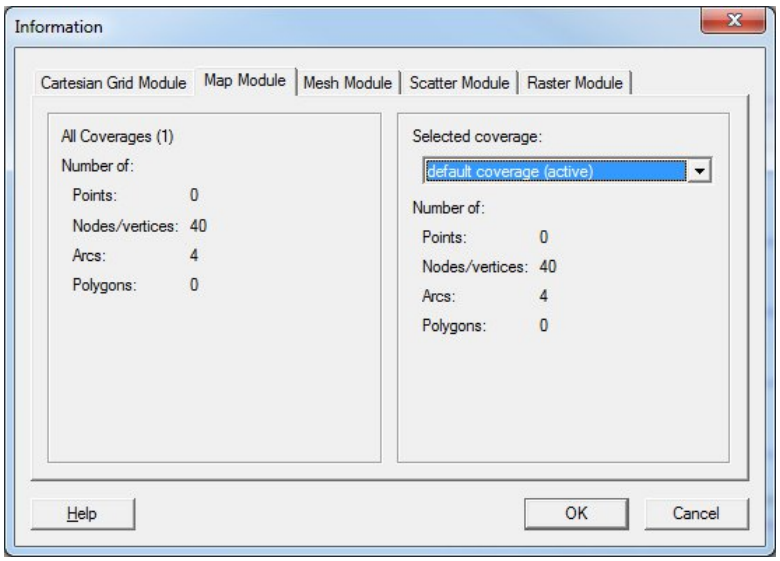


Information dialog showing the Cartesian Grid Module *tab*

Map Module Information

The following information is shown on the *Map Module* tab of the *Information* dialog:

- For all coverages:
  - Number of points
  - Number of nodes / vertices
  - Number of arcs
  - Number of polygons
- For selected coverage:
  - Number of points
  - Number of nodes / vertices
  - Number of arcs
  - Number of polygons

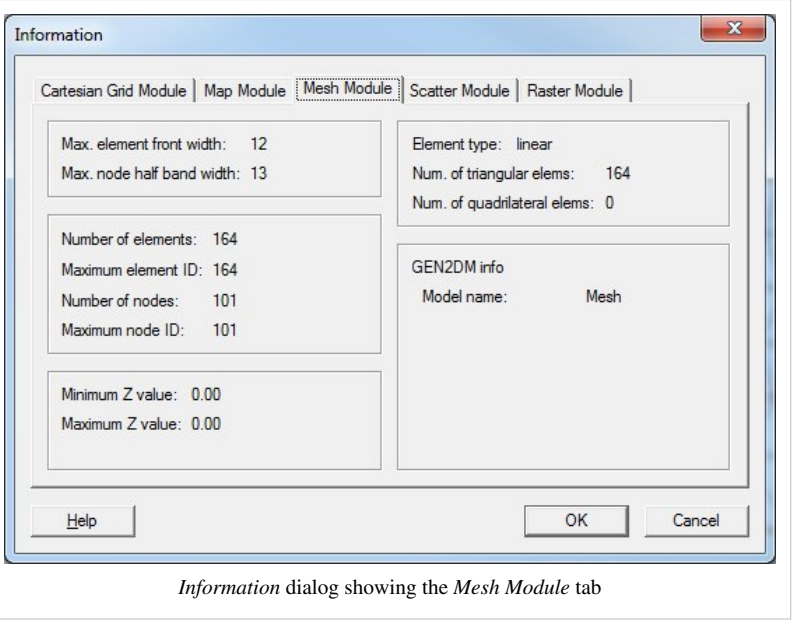


Information dialog showing the *Map Module* tab

Mesh Module Information

The following information is shown on the *Mesh Module* tab of the *Information* dialog:

- Maximum element front width
- Maximum node half band width
- Number of elements
- Maximum element ID
- Number of nodes
- Maximum node ID
- Minimum Z value
- Maximum Z value
- Element type
- Number of triangular elements
- Number of quadrilateral elements
- Model specific Info
  - RMA2
    - Transition elements
    - Junction elements
    - Control elements
    - Linear elements
    - 1D nodes without 1D
  - FESWMS
    - Number of culverts
    - Number of piers
    - Number of weirs
    - Number of drop inlets
    - Max ceiling value
    - Min ceiling value
  - Generic Mesh Model
    - Model name

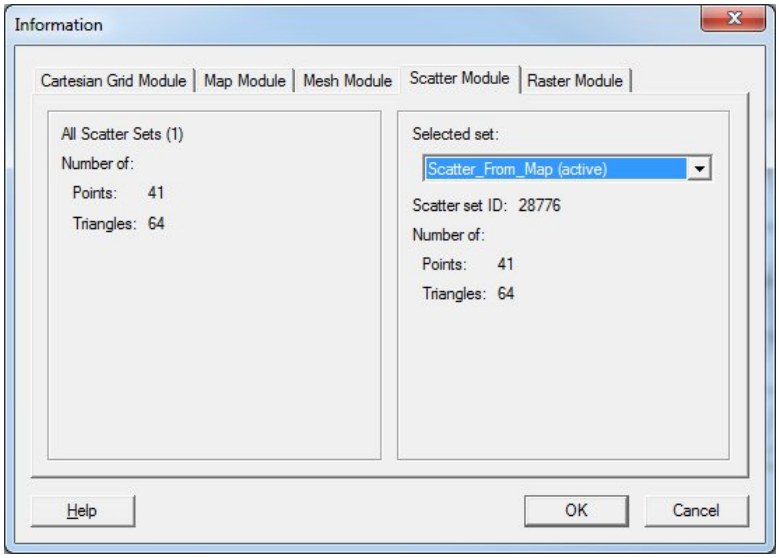


Information dialog showing the *Mesh Module* tab

Scatter Module Information

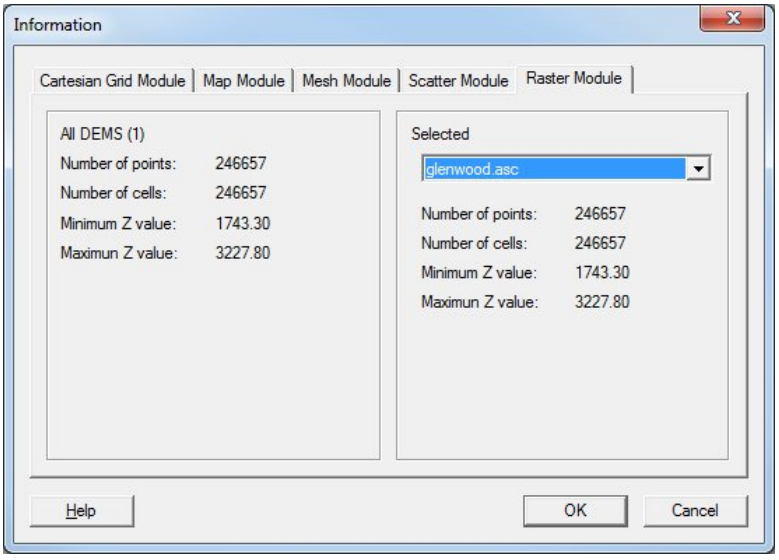
The following information is shown on the *Scatter Module* tab of the *Information* dialog:

- For all scatter sets
  - Number of points
  - Number of triangles
- For selected scatter sets
  - Scatter set ID
  - Number of points
  - Number of triangles



Information dialog showing the *Scatter Module* page

Raster Module Information



Information dialog showing the *Raster Module* tab

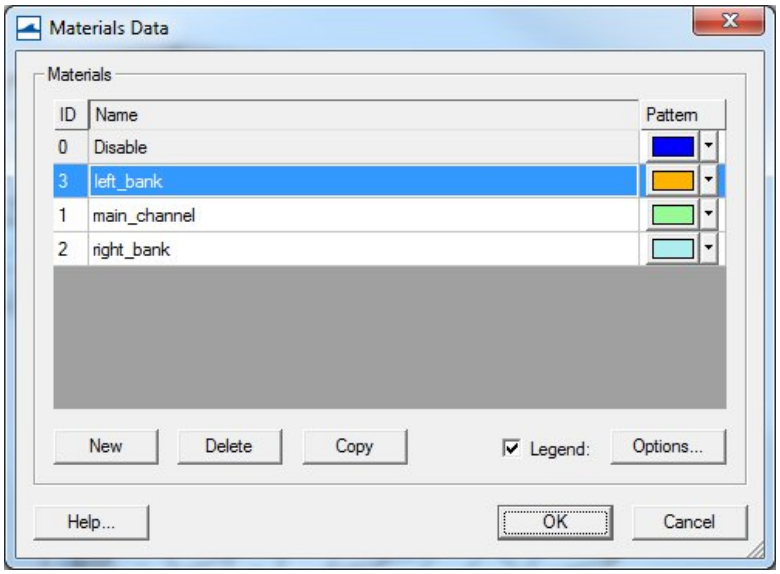
The following information is shown on the *Raster Module* tab of the *Information* dialog:

- All DEMS (1)
  - Number of points:
  - Number of cells:
  - Minimum Z value
  - Maximum Z value
- Selected
  - Number of points
  - Number of cells
  - Minimum Z value
  - Maximum Z value

Related Topics

- File Menu

Materials Data



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.

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.

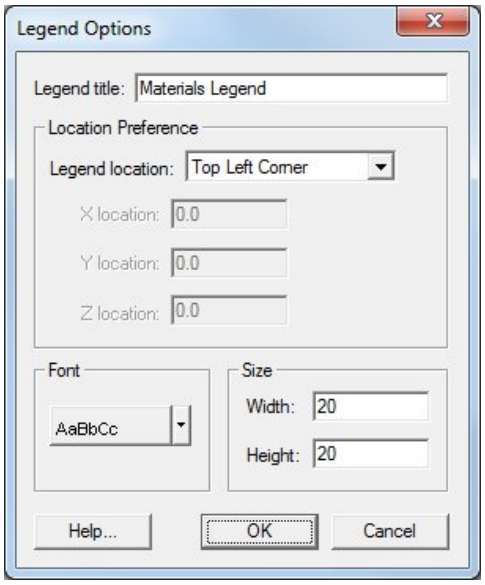
Buttons

- *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.



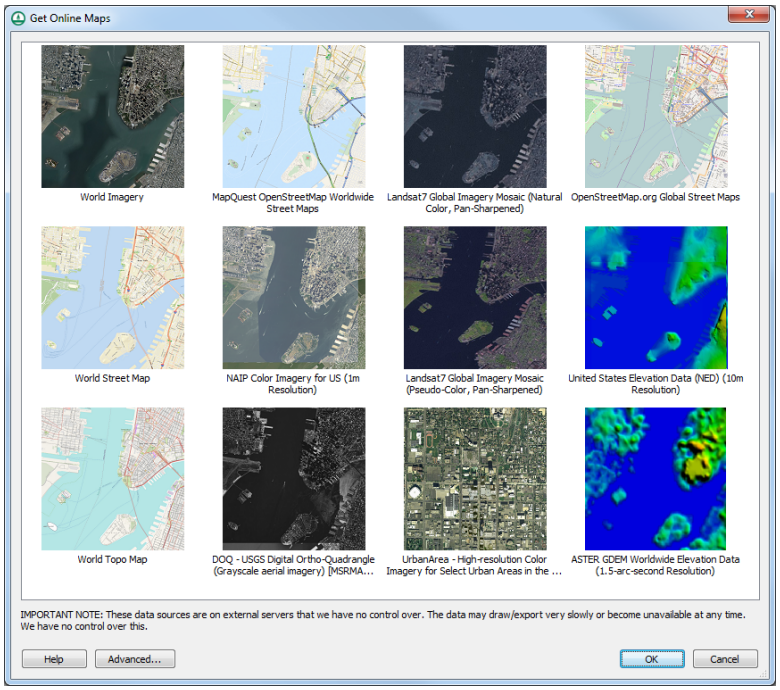
Materials Display Options *Legend Options* dialog

Related Topics

- Area Property Coverage
- Edit Menu

Get Online Maps

The **Get Online Maps** button allows you to select online data from a variety of different sources. Once online maps has been selected, the data resolution will be automatically adjusted based on your zoom parameters. Online maps can only be viewed in plan view.



Get Online Maps Dialog

Online maps are raster datasets that can contain imagery, elevation, or land use information. If you have online maps in your program, you can right-click on each of the maps to convert them to static images that can be saved to your local hard drive. You can convert or interpolate online maps containing elevation data to various elevation formats.

Note that online data sources are on external servers that we have no control over. The data may draw/export very slowly or become unavailable at any time. We have no control over this.

The **Advanced** button allows you to select from other data sources and to use other online data query functions that may not be fully supported. In the *Advanced* dialog, the **Add Sources From File** button allows you to add new Web Map Service (WMS) sources from an external text file.

More information about the various types of online data can be found by visiting the following links:

- NED data – USGS [1]
- ASTER and SRTM data – USGS & NASA [2]
- NLCD and CORINE (European) Land Cover data [3]
- World Imagery [4] More Info [5]
- World Street Maps [6] More Info [7]
- World Topo Maps [8] More Info [9]
- MapQuest OpenStreetMap Worldwide Street Maps [10]
- USA Topo Maps [11] More Info [12]
- Other data sources-Geologic data, land cover, etc. (use the advanced button)

Exporting to a File

An online map can be exported to a file and loaded into the project. You may want to do this to save a local copy and not be dependent on internet access. Also, there may be more commands and options available with a local file, such as interpolation or conversion to other object types, than with online maps.

References

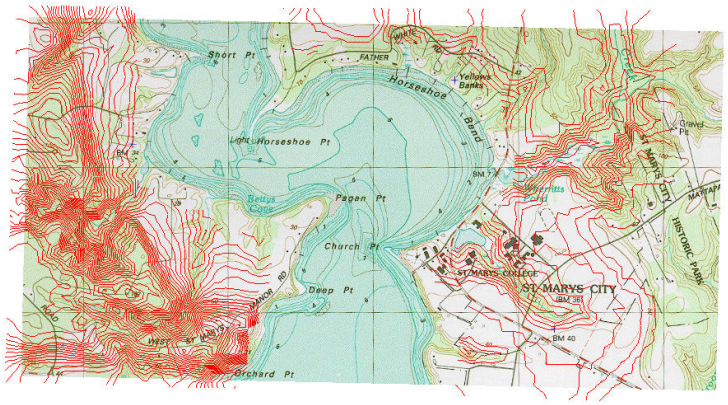
[1] http://ned.usgs.gov/
[2] http://srtm.usgs.gov/
[3] http://seamless.usgs.gov/nlcd.php
[4] http://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer
[5] http://www.arcgis.com/home/item.html?id=10df2279f9684e4a9f6a7f08febac2a9
[6] http://services.arcgisonline.com/ArcGIS/rest/services/World_Street_Map/MapServer
[7] http://www.arcgis.com/home/item.html?id=3b93337983e9436f8db950e38a8629af
[8] http://services.arcgisonline.com/ArcGIS/rest/services/World_Topo_Map/MapServer
[9] http://www.arcgis.com/home/item.html?id=30e5fe3149c34df1ba922e6f5bbf808f
[10] http://developer.mapquest.com/web/products/open
[11] http://services.arcgisonline.com/ArcGIS/rest/services/USA_Topo_Maps/MapServer
[12] http://www.arcgis.com/home/item.html?id=99cd5fbd98934028802b4f797c4b1732

CAD Data

CAD Support

- AutoCAD DXF and DWG files can be read into SMS (support of DGN format is under development)
- Supports up to AutoCAD version 2007
- CAD data is displayed in 3D
- CAD data can be converted to map or scatter data

SMS can import CAD data from AutoCAD formats (DWG/DXF). CAD data in SMS can be converted and use in other modules. SMS data can be converted into CAD layers and saved in a supported format.



Using CAD data in SMS

Importing

SMS can import DWG or DXF files via the *File* | **Open** command. If there is already CAD data in memory, SMS will replace the existing data with the data being imported. Currently, SMS cannot merge the incoming data with the data in memory.

Working with CAD data

The objects in a DWG or DXF file are organized into layers. The display of layers in a CAD drawing is controlled using the check boxes in the *Project Explorer*. Individual layers can be turned off/on or if you would like to turn off the display of all CAD data then uncheck the box next to the CAD folder.

Creating CAD data from SMS data

You can select either the DWG or DXF file types to save the CAD data. SMS objects must first be converted to CAD data before CAD data can be exported. To convert SMS data to CAD data, right-click in the empty space at the bottom of the project explorer and choose **Convert To CAD**.

Delete Data

To Delete the CAD Data right-click on the CAD data folder in the tree and select **Delete** from the pop-up menu. If the CAD data was imported from a file, the file is not deleted from disk.

CAD → Map

CAD data can be converted to SMS feature objects by right-clicking on the CAD data folder in the *Project Explorer* and selecting **CAD → Map** command. CAD points are turned into points, CAD lines and polylines are turned into arcs, and CAD polygons are turned into polygons. The feature objects are added to the active coverage. Once converted, the feature objects can be used to build conceptual models.

CAD → 2D Scatter

A set of CAD 3D faces which have been imported to SMS can be converted to a 2D Scatter Set by right-clicking on the CAD data folder in the *Project Explorer* and selecting the **CAD → 2D Scatter** command.

Exporting

SMS data can be exported to a DWG or DXF file that can then be read into a CAD package. If there is CAD data in memory when a SMS project is saved, SMS creates a new DWG file from the CAD data. The file is put in the same folder with the other project files and named using the project prefix.

Alternatively, CAD data in memory can also be saved using the **Save As** command in the *File* menu.

CAD Data Right-Click Menus

The following *Project Explorer* mouse right-click menus are available when the mouse right-click is performed on a CAD Data item.

CAD Data 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

CAD Data 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:

- **Convert**
  - CAD → Map – Converts CAD data to Map Module data
  - CAD Faces → 2D Scatter Triangles – Converts CAD Faces to Scatter Module triangles
  - CAD Points → 2D Scatter – Converts CAD Points to Scatter Module vertices

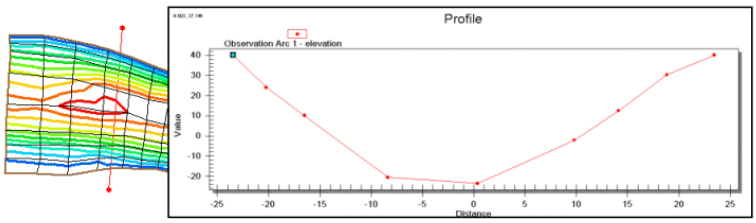
Related Topics

- Project Explorer Right-Click Menus

2.1. 2D Plots

Plot Window

At a glance



A plot showing a cross-section view of a river channel

- 2D plots to visualize results and compare to measured values
- Profile plots view scalar data along an arc
- Time-series plots view scalar, vector, or flux (flowrate) data at a point or across an arc
- Several kinds of plots can be used to compare model results with measured data

In SMS, the *Plot Window* can be used to display various plots. Plots aid the user in extracting data from two or three dimensional objects, model verification, and defining one-dimensional river models. By selecting *Display Options* | **Plot Wizard**, a user is given help in a step by step process to create a variety of plots. The list of currently available plots in the *Plot Wizard* is shown below. All of the plots listed below are associated with Observation Coverages, except the 1-dimensional river plots.

Available Plot Types

1. Computed vs. Observed Data
2. Residual vs. Observed Data
3. Error vs. Simulation
4. Error vs. Time Step
5. Error Summary
6. Time Series
7. Observation Profile
8. TUFLOW Cross-sections
9. PTM Gages
10. Runup/Overtopping Transect
11. Runup/Overtopping Solution
12. GenCade Inlet Time-series
13. GenCade Shoreline Change and Transport
14. Angle Representation Region (ARR) mesh quality assessment plot

Plot Wizard

The Plot Wizard is used to bring up a plot window which will display the specified plot. There are two steps that a user is directed through to create a plot.

Step 1 – User selects the desired plot type from the list. Some plot types are hidden unless certain requirements are met.

- The "TUFLOW Cross Section" plot requires a TUFLOW Cross-Section coverage.
- The "Runup/Overtopping Transect" and "Runup/Overtopping Solution" plots require a Runup/Overtopping simulation.
- The plot "PTM Gages" requires a PTM Gage coverage.
- The "GenCade Inlet TS" and "GenCade Shoreline" plots require a GenCade 1D grid.

Each plot is described as they are highlighted from the list. The Plot Wizard will also tell the user if they have appropriate data to be able to make certain plots.

Step 2 – The plot is defined by selecting what data will compared, which time step will be shown, and other pertinent information. Each plot's options are described in more detail by click on the Plot Type links above.

By default, a plot is displayed after being created in a separate display window than the simulation data. A plot window can be minimized, moved, and resized just like any other window.

Plot Options

Right-clicking on a plot will bring up a menu of commands for formatting the data in the plot as well as giving access to tools for exporting the plot data for use in spread sheets or other plotting utilities.

Plot Window Right-Click Menu

The following *Plot Window* mouse right-click menus are available:

- Plot Data – Opens step 2 of the plot wizard
- Display Options – Opens the *Profile Customization* dialog
- Axis Titles – Opens the *Axis Titles* dialog. The *Axis Titles* dialog allows you to edit the X and Y axis titles of plots.
- Set as Display Defaults
- Legend – Set the legend location (Top, Bottom, Left, Right)
- Symbol Size – Set the symbol size (Micro, Small, Medium, Large)
- Frame Plot – If the view is zoomed in to a portion of the plot, resets zoom extents
- Maximize Plot – Makes the plot appear full screen
- View Values – Opens the *View Values* dialog
- Export/Print – Opens the *Exporting Profil*e dialog

Time Settings Options

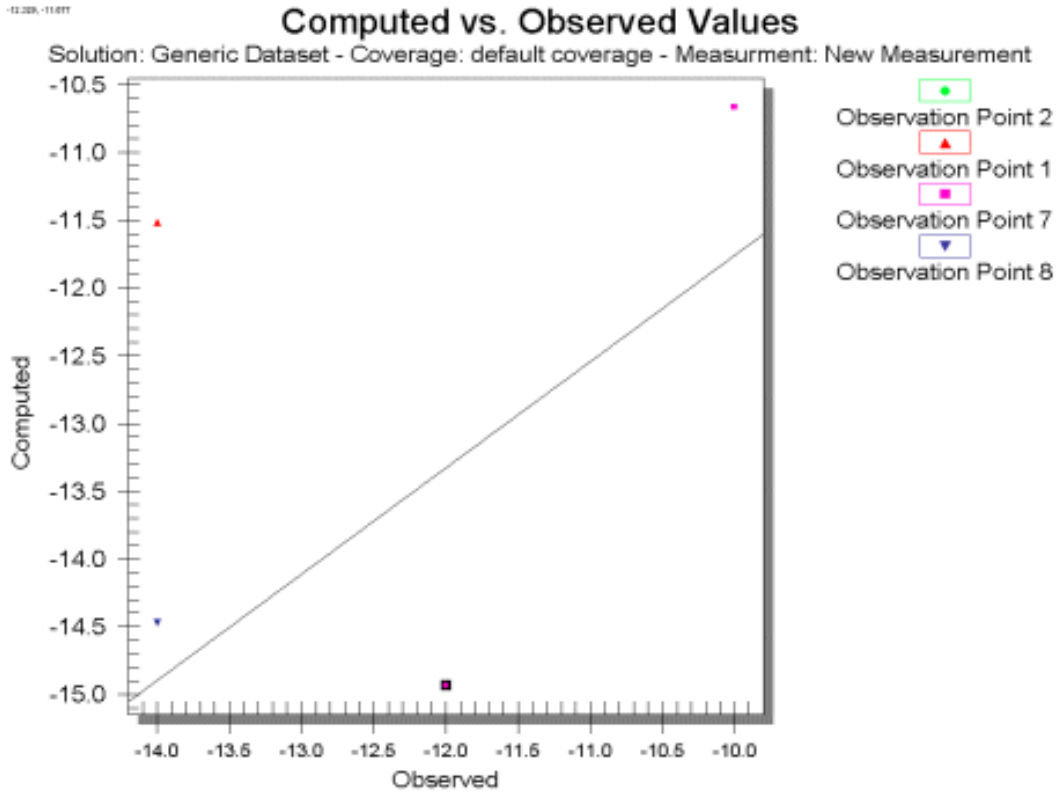
The formatting of Date / Times displayed in plots is controlled by the global time settings. See Time Settings for information on how to change the format times are displayed in.

Related Topics

- Visualization

Computed vs. Observed Data

A *Computed vs. Observed* plot is used to display how well the entire set of observed values for observation points matches the solution data. On this plot is drawn a 45 degree line, representing what would be a perfect correspondence between observed data and solution values. Then, one symbol is drawn for each observation point at the intersection of the observed and computed values for the point. This plot can show the trend of the solution values with regards to matching the observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. These plots are created in the *Plot Wizard* by setting the plot type to *Computed vs. Observed*. A sample plot is shown in the figure.



Computed vs. Observed Plot Options

After the plot type is set in Step 1 of the *Plot Wizard*, the **Next** button is clicked to go to Step 2, which displays the following items.

Coverage – Displays the name of the coverage where the current data for the plot is coming from.

Measurement – This is the name of the current measurement, created in the *Feature Objects* | **Attributes** dialog, being plotted.

Feature Objects – Displays which feature object is utilized in the current plot, points or arcs.

Related Topics

- Plot Window

Error Summary

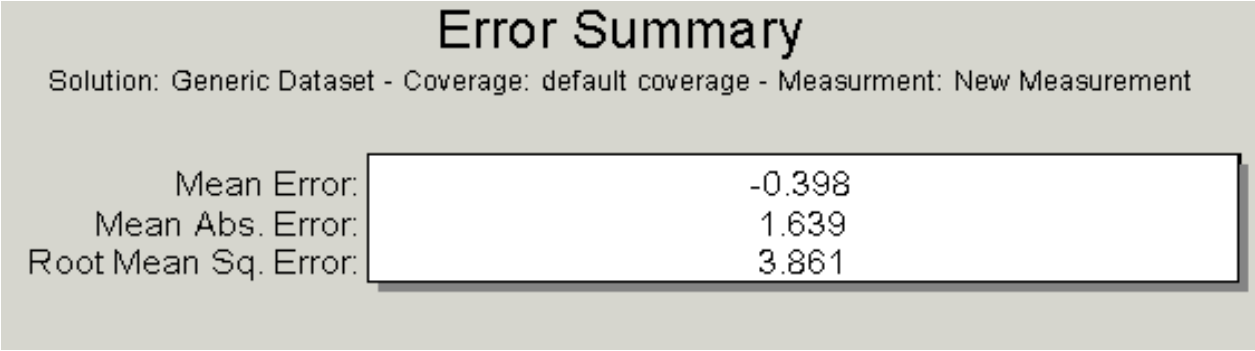
An *Error Summary* plot is used to display a text listing of the mean error, mean absolute error, and root mean squared error for a dataset and the observed values associated with a mesh or grid on observation points in the Observation Coverage. The errors shown are the mean errors for all observation points with computed data.

Mean Error – This is the average error for the points. This value can be misleading since positive and negative errors can cancel.

Mean Absolute Average – This is the mean of the absolute values of the errors. It is a true mean, not allowing positive and negative errors to cancel.

Root Mean Square – This takes the sum of the square of the errors and then takes its square root. This norm tends to give more weight to cases where a few extreme error values exist.

Error Summary plots are created in the *Plot Wizard* by setting the plot type to *Error Summary*. A sample plot is shown in the figure.



Error Summary Plot Options

After the plot type is set in the first step of the *Plot Wizard*, the **Next** button is clicked to move to the second step of the *Plot Wizard*.

Coverage – Displays the name of the coverage where the current data for the plot is coming from.

Measurement – This is the name of the current measurement, created in the *Feature Objects* | **Attributes** dialog, being plotted.

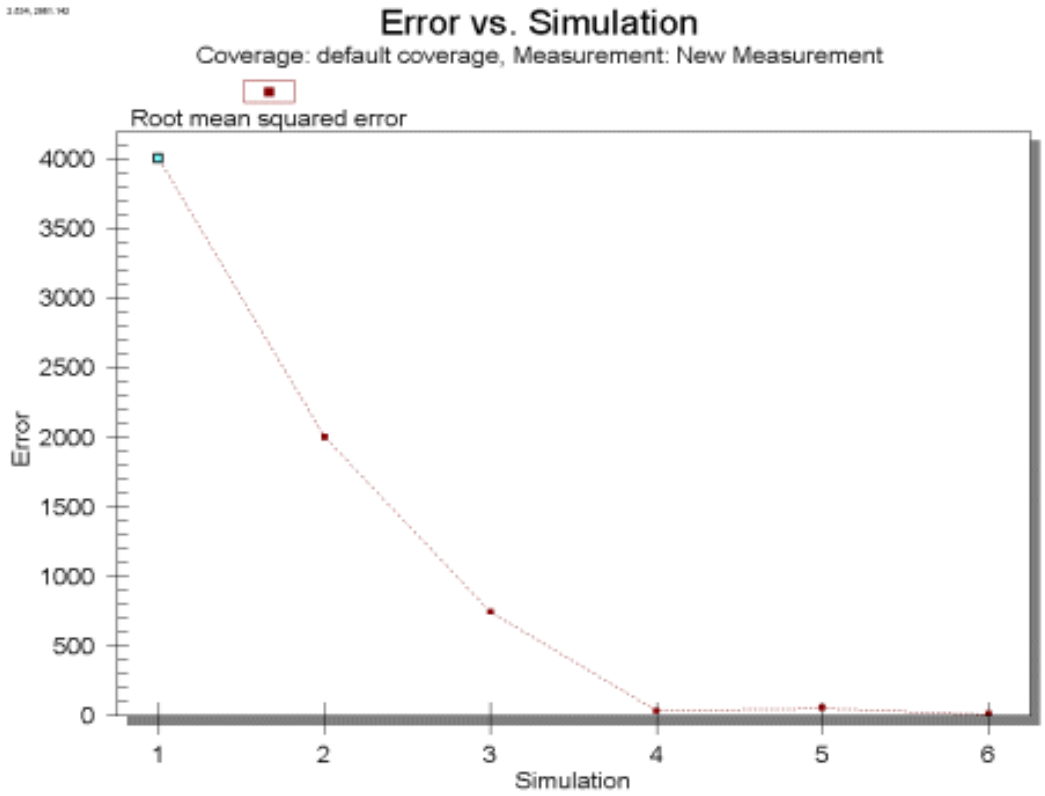
Feature Objects – Displays which feature object is utilized in the current plot, points or arcs.

Related Topics

- Plot Window

Error vs. Simulation Plot

An Error vs. Simulation plot is generally used with constant simulations and measurement types, although it may be used in transient simulations. This plot can display the mean error, mean absolute error, and root mean squared error between successive solutions and a set of observed data. Various simulations would be run after changing model parameters, such as material roughness values and/or eddy viscosities. The plot will show trends in the solution to see if model parameter changes are causing better calibration with measured field data. Error vs. Simulation plots are created in the Plot Wizard by setting the plot type to Error vs. Simulation. A sample plot is shown in the figure.



Error vs. Simulation Plot

After the plot type is set in the first step of the Plot Wizard, the second step of the Plot Wizard shows the options for the Error vs. Simulation plot.

Solutions – This box lists all available solutions for the simulation. Select the desired solution to use its datasets in the plot.

Move Up/Move Down – SMS initially shows the solutions in the order they were opened. However, this is not necessarily the order in which they were run. To change the order, highlight a solution and move it up or down to rearrange their order.

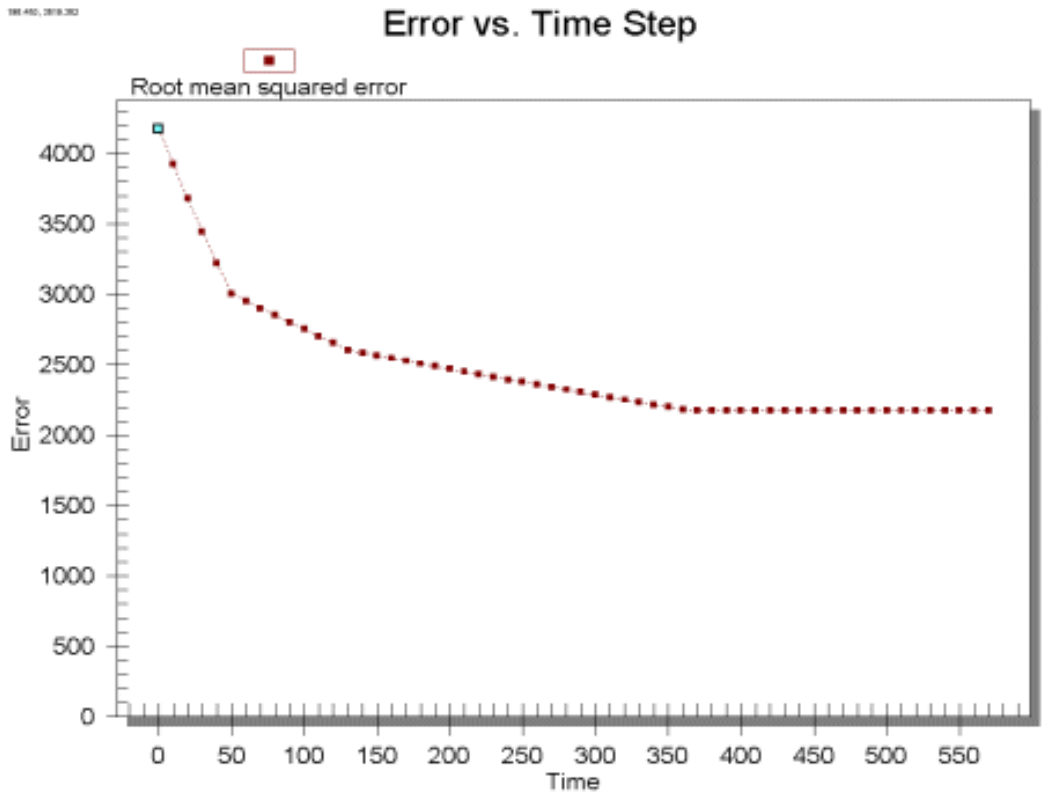
Check Box Options – There are three options that can be turned on or off. The three options determine whether the mean error, mean absolute error, and root mean squared error plots should be shown. Because these values are an average of all observation points, their line and symbol styles are not linked to any one observation point, but can be defined by clicking on the appropriate canvas window in the dialog.

Related Topics

- Plot Window

Error vs. Time Step Plot

An *Error vs. Time Step* plot is used with transient simulations to display the mean error, mean absolute error, and root mean squared error between a solution and observed data as a function of time. This plot is shown for a single dataset of a mesh or grid as an average of all observation points assigned to the specified measurement type in the Observation Coverage. The measurement type should be defined as a transient measurement. Although this plot can be used for constant measurement types, only a single point will be shown in the plot, and you would be better off using the *Error Summary Plot*. Transient measurement types will show the average errors at each time step of the data set. *Error vs. Time Step* plots are created in the *Plot Wizard* by setting the plot type to *Error vs. Time Step*. A sample plot is shown in the figure.



Error vs. Time Step Plot Options dialog

After the plot type is set in the first step of the *Plot Wizard*, the plot options are shown in Step 2 of the *Plot Wizard*, contains the following options.

Computed – This lists all available datasets. The dataset to be analyzed should be chosen.

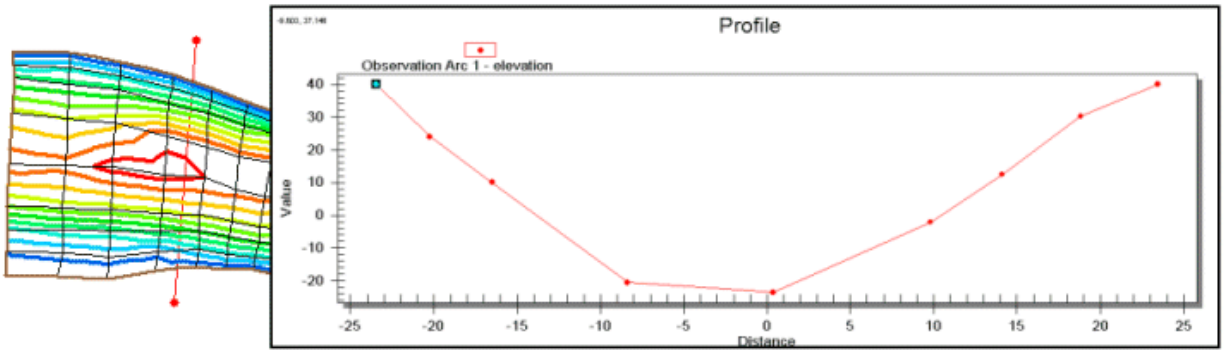
Check Box Options – There are three options that can be turned on or off. The three options determine whether the mean error, mean absolute error, and root mean squared error plots should be shown. Because these values are an average of all observation points, their line and symbol styles are not linked to any one observation point, but can be defined by clicking on the appropriate canvas window in the dialog.

Related Topics

- Plot Window

Observation Profile

A *Profile plot* is used to display the variation of one or more scalar datasets associated with a mesh or grid along observation arcs in the Observation Coverage. Profile plots are created in the *Plot Wizard* dialog by selecting Observation Profile from the plot type list. When an arc is selected two small arrows appear at either end of the arc. These arrows indicate the viewing direction for the plots. To change the viewing direction select the arc and execute the *Feature Objects* | **Reverse Arc Direction** command. A sample plot is shown in the figure.



Profile Plot Options

After the plot type is set in Step 1 of the *Plot Wizard*, the profile plot options need to be defined. The following options must be set for a profile plot:

Coverage

A profile plot operates on a single observation type coverage. The following coverage related options are available:

- **Coverage –** If multiple observation type coverages exist, the coverage to use for the profile plot must be selected.
- **Extraction method**
  - **Model Intersections –** Profile plot points are based on intersections of the specified feature arcs and element, cell, or triangle edges.
  - **Points and Vertices –** Profile plot points are interpolated at the location of points and vertices on the specified feature arcs.

Dataset

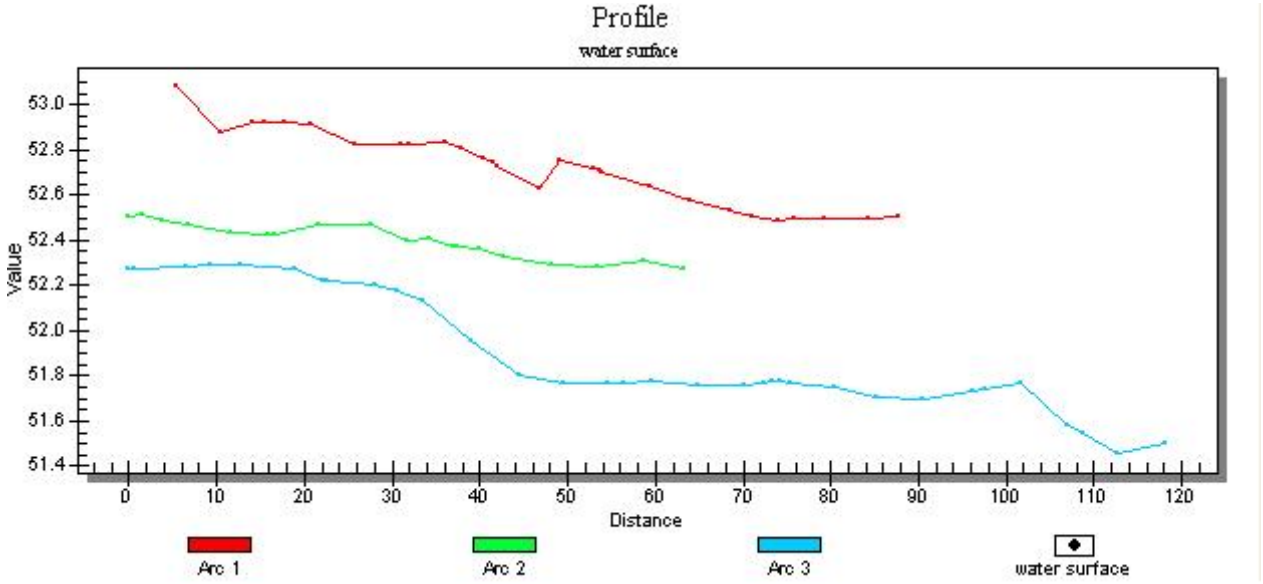
- **Active dataset –** Profile plot points are based on the active dataset. The profile plot will update when the active dataset is changed.
  - **Module** - Since each module contains an active dataset, when using the active dataset option, the module must be specified.
- **Specified dataset(s) –** Profile plot points are based on the specified dataset(s). Datasets from different modules can be specified.

Time step

- **Active time step –** Profile plot points are based on the active time step. The profile plot will update when the active time step is changed.
- **Specified time step –** Profile plot points are based on the specified time step.
- **Use active dataset and time step –** This option causes the plot to display the values of the active dataset and time step for each arc being plotted. When the active dataset changes, the plot is recomputed and updated.
- **Use selected dataset and time step –** This option causes the plot to display the values of one or more specified datasets or time steps for each arc being plotted. Changing the active dataset does not affect the plot. Check the check-box of the dataset that will be viewed from the list box.

Plotting With Multiple Arcs Selected

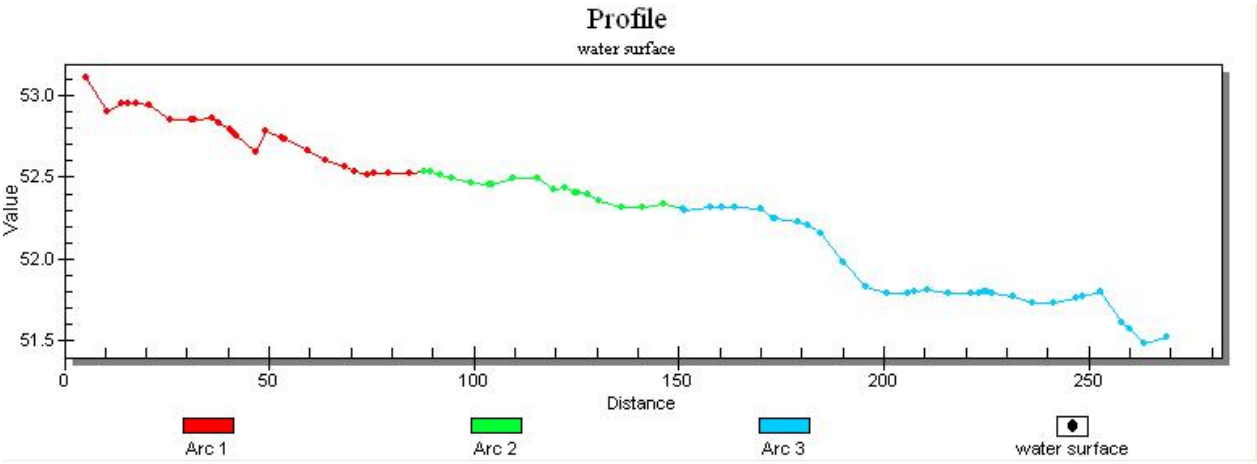
There are two ways in which an observation profile can be created when multiple arcs will be graphed. Multiple arcs can be graphed on a single plot was so they appear in separate segments as shown below.



Multiple observation arcs can also be plotted to look continuous if they are part of an arc group by following these steps:

- Create a profile arc that composed of more than one arc (this means the arcs must be connected at the end points and start points)
- Choose the menu command Feature Objects | Create Arc Group
- Create the observation profile plot like before by using the Plot Wizard

When the plot is generated, it will look as shown below where the line is continuous. The different arcs are drawn in their respective colors, but are linked together end to end.

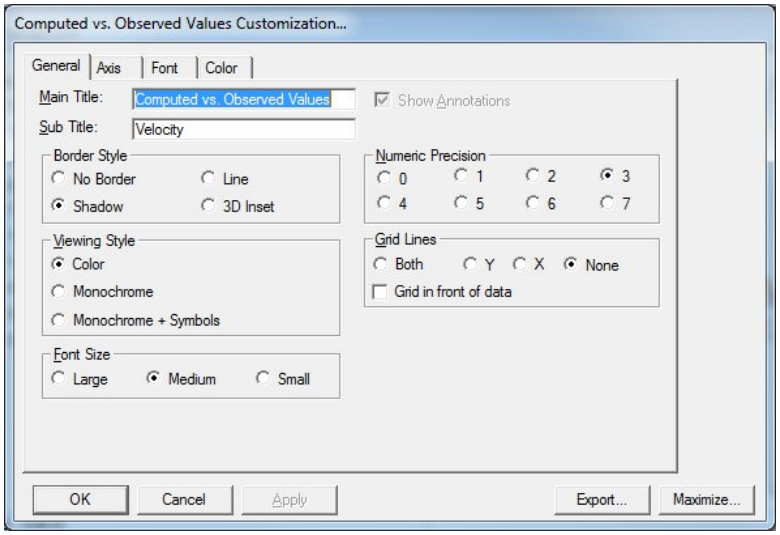


Related Topics

- Plot Window

Profile Customization Dialog

The *Profile Customization* dialog allows you to edit many plot properties. The plot options are organized onto the following tabs in the dialog:



Profile Customization dialog for *Computed vs. Observed Values Customization*

- **General Tab –** plot title, border style, viewing style, font size, numeric precision, and grid line style can be changed
  - Main Title
  - Sub Title
  - Show Annotations
  - Border Style – has options for No Border, Line, Shadow, and 3D Inset
  - Viewing Style – has options for Color, Monochrome, and Monochrome + Symbols
  - Font Size – has options for Large, Medium, and Small
  - Numeric Precision – has options for 1–7
  - Grid Lines – has options for Both, Y, X, None, and Grid in front of data
- **Axis Tab –** contains x and y axis information
  - Y Axis

Linear Auto

Log Min

Max

Min/Max

- X Axis

Linear Auto

Log Min

Max

Min/Max

- **Font Tab –** plot font style can be edited. Users can select the font style for each of the following:
  - Main Title
  - Sub-Title
  - Subset / Point / Axis Labels
- **Color Tab –** any color option can be changed here. It has the following options:
  - Graph Attributes
    - Desk Foreground
    - Desk Background
    - Shadow Color
    - Graph Foreground
    - Graph Background
    - Table Foreground
    - Table Background
  - Quick Styles
    - Bitmap/Gradient Styles
- **Export –** allows the user to export the plot and plot data in different file formats, to a printer, or the clipboard
- **Maximize –** cause the plot to fill the computer screen. Pressing the *ESC* key will return the plot window to its normal size.

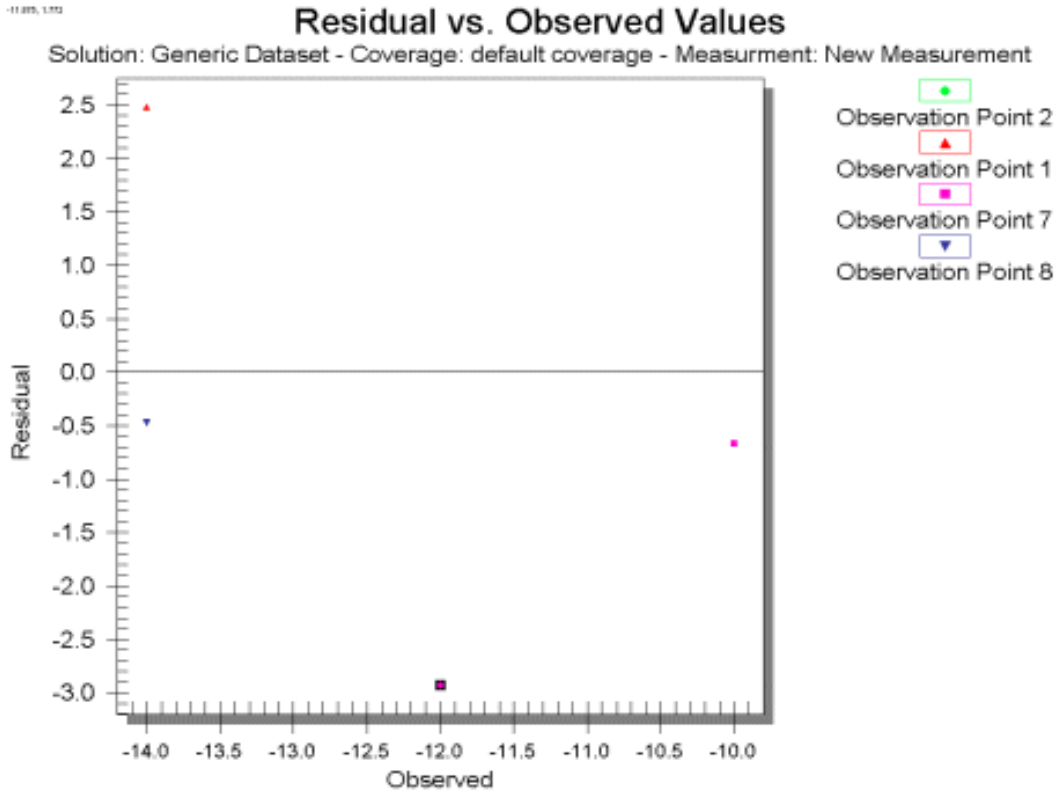
To open the *Profile Customization* dialog, select *Display* | **Plot Display Options** or right-click on the plot and select **Display Options**.

Related Topics

- Plot Window

Residual vs. Observed Data

A Residual vs. Observed plot is used to display how well the entire set of observed values for observation points matches the solution data. On this plot is drawn a horizontal line along an error of zero, representing what would be a perfect correspondence between observed data and solution values. Then, one symbol is drawn for each observation point at the intersection of the observed and residual (computed-observed) values for the point. This plot can show the trend of the solution values with regards to matching the observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. These plots are created in the *Plot Wizard* setting the Plot Type to Residual vs. Observed. A sample plot is shown in the figure below.



Residual vs. Observed Plots

After the plot type is set in Step 1 of the *Plot Wizard*, next to move to Step 2 where the plot options will be available:

Coverage – Displays the name of the coverage where the current data for the plot is coming from.

Measurement – This is the name of the current measurement, created in the *Feature Objects* | **Attributes** dialog, being plotted.

Feature Objects – Displays which feature object is utilized in the current plot, points or arcs.

Related Topics

- Plot Window

Time Series

The time series is similar to the XY series with a few differences.

1. Times series can support more than 2 columns of data.
2. The time series group can be specified (i.e. velocity, xy), when available.
3. When using times, the reference time can be specified.
4. The units of each column can be specified (from a list) and the column data will be converted when switching between units.

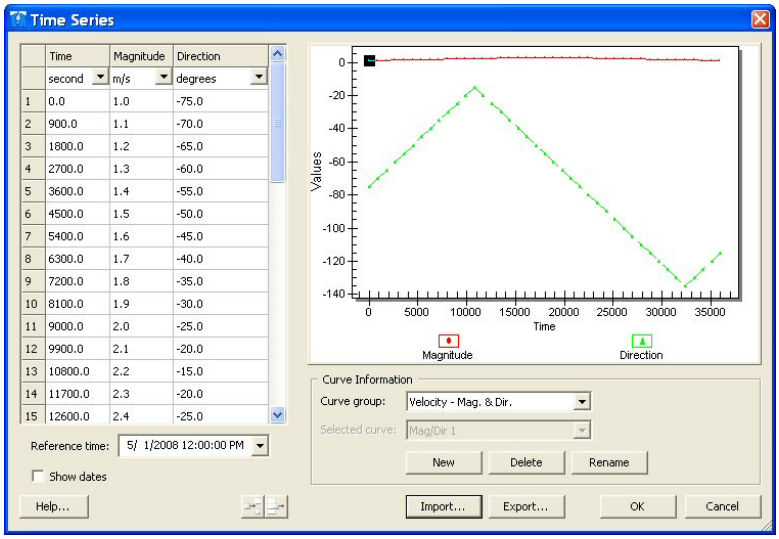
The time series editor can import xy series.

The time series assumes that all angles used are specified in the cartesian system.

For information about ADH model specific curve groups, see ADH Time Series.

Dialog Description

Curve Information



Time Series Dialog

Curve group field specifies the current group. This may be a combo box (depending on how the window is accessed) which allows the other groups to be selected.

Selected curve field specifies the current curve loaded. This may also be a combo box containing multiple curves. If this field is empty, then no curves exist in the current curve group.

New... button adds a new curve to the current curve group and opens a window in which the name of the new curve is specified. The name will appended if necessary to ensure uniqueness within the curve group. The new curve will be selected and appear in the *Selected curve* field.

Delete button removes the currently selected curve from the curve group. This button is only enabled if a selected curve exists.

Rename... button opens a window to specify a new name for the selected curve. This button is only enabled if a selected curve exists.

Plot type field specifies the current plot format. The available plot types are determined by the format of the current time series. The options for displaying the plots are as follows:

1. *Scientific* – This displays the data on a traditional XY plot.
2. *Multi axes* – This displays each column of the dataset on a separate Y axis.
3. *Rose* – This filters the data into vector data and then displays a rose plot of the binned data.

Various plot options, such as legend style and numerical precision, are accessed by right-clicking on the plot. It is important to note than when plotting data, the first column of the time series is always assumed to be the X values. This field is only enabled if a selected curve exists.

Curve Data

Spreadsheet lists the data of the selected curve. The column types are determined by the current curve group.

Attributes... button opens the selected curve's attribute window. This button is only enabled if the current curve group includes attributes. For information about ADH model specific curve group attributes, see ADH Time Series Attributes.

Insert New Row Above and ***Delete Row(s)*** toolbar assists in editing the spreadsheet rows. The tools are only available if there is a valid cell selection that excludes the title, units, and empty rows.

Reference time field specifies the date and time the selected curve begins out. This field is only visible if one of the curve groups available (listed in the ***Curve group*** field) allows time referencing and the field is enabled if the current curve group allows it.

Show dates check box specifies whether the time values of the curves are displayed in the spreadsheet as date and times instead of offsets from the reference time. This control is visible and enabled based on the same requirements as the '**Reference time**' field.

Miscellaneous (Outside of any group)

Import... and ***Export...*** buttons read and save, respectively, Time series (*.tsd) and XY series (*.xys) files.

Related Topics

- Compass Plot
- Spatial Data Coverage
- Coverages

Time Series Data File

The Time Series Data file format provides a means of transferring data to and from SMS. It is a simple ASCII file format that defines the type of data and its time reference.

Sample Format

The file follows the following format:

TIME_SERIES
Series Type Curve Name NCols NVals Reference Date

```
          Date 1         Value 1A         Value 1B

          Date 2         Value 2A         Value 2B

          Date 3         Value 3A         Value 3B

          Date 4         Value 4A         Value 4B

          .

          .

          .
```

```
         Date NVals     Value NVals      Value NVals
```

Sample File

The following illustrates a sample file:

TIME_SERIES "Mag/Dir 1" "Velocity - Mag. & Dir." 3 41 "05/01/2008 12:00:00"

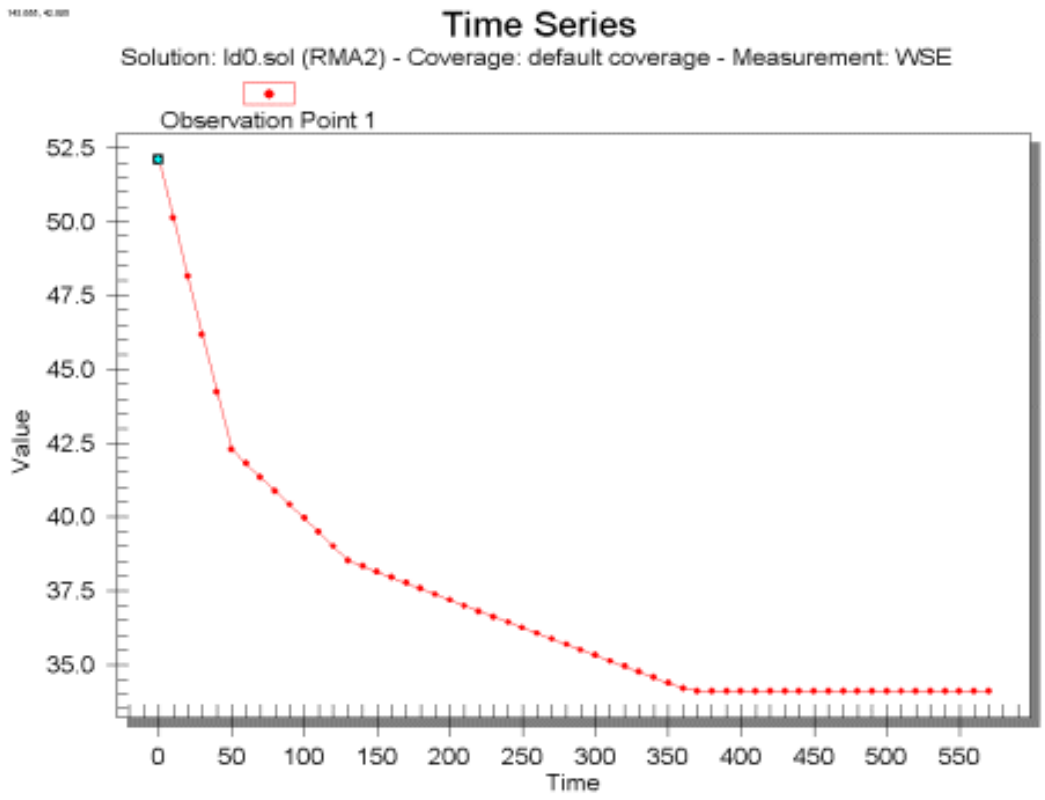
```
             0.0              1.0            -75.0
           900.0              1.1            -70.0
          1800.0              1.2            -65.0
          2700.0              1.3            -60.0
          3600.0              1.4            -55.0
          4500.0              1.5            -50.0
          5400.0              1.6            -45.0
          6300.0              1.7            -40.0
          7200.0              1.8            -35.0
          8100.0              1.9            -30.0
          9000.0              2.0            -25.0
          9900.0              2.1            -20.0
         10800.0              2.2            -15.0
         11700.0              2.3            -20.0
         12600.0              2.4            -25.0
```

Related Topics

- Time Series
- Spatial Data Coverage
- Coverages

Time Series Plot

A Time Series plot is used to display the time variation of one or more scalar datasets associated with a mesh or grid at observation points in an Observation Coverage. In addition, if transient calibration data has been defined, a band can be shown which represents a time variant Calibration Target. Only transient data sets may be used in these plots. Time Series plots are created by using the *Plot Wizard*, found in the *Display* menu, and selecting **Time Series** from the plot type list in Step 1 of the *Plot Wizard*. A sample plot, with calibration target band, is shown in the figure below.



Time Series Plot Options

After the plot type is set in Step 1 of the *Plot Wizard*, the user defines the time interval and the scalar dataset desired for the plot in Step 2. When this is completed, click **Finish** and the plot will be generated.

Use active dataset – This option causes the plot to display the values of the active dataset for each observation point being plotted. When the active dataset changes, the plot is recomputed and updated.

Use selected dataset – This option causes the plot to display the values of one or more specified datasets for each point being plotted. Changing the active dataset does not affect the plot. Select the dataset from the list box by putting a check in the dataset's check box.

Use calibration data – This allows the user to display the calibration curve defined for each point. If you do not have calibration data for the entity, leave the box unchecked and the calibration data will not be displayed.

For more information concerning how to edit the *Time Settings*, see Plot Window.

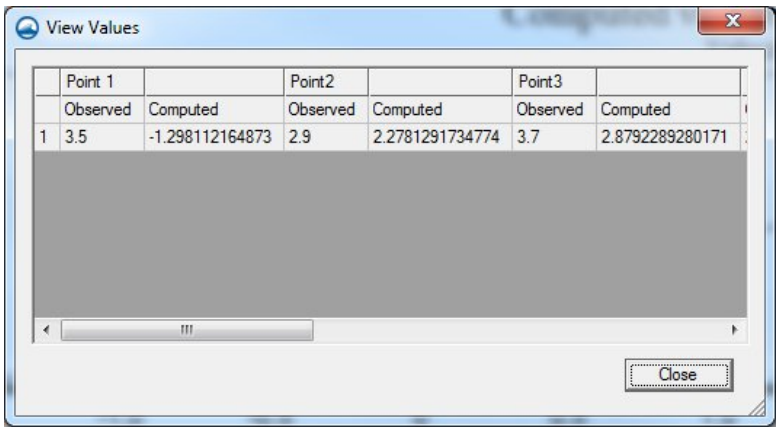
Related Topics

- Plot Window

View Values Dialog

The *View Values* dialog displays the values used to create the plot. This dialog is accessed by right-clicking in the *Plot Window* and selecting the **View Values** command. The values can easily be copied from the dialog and pasted into a spreadsheet program or document using the following steps:

1. Select the cells of interest from the spreadsheet
2. Press *CTRL + C* – cell contents are now in the clipboard
3. Select the paste destination
4. Press *CTRL + V*



View Values dialog

Related Topics

- Plot Window

ARR Mesh Quality Assessment Plot

The Angle Representation Region (ARR) plot is used to assess the overall quality of a triangular mesh such as those used by ADH, ADCIRC and other numerical engines. When the plot wizard is selected, this option appears if the mesh module is enabled. Clicking finish in the plot wizard results in an ARR plot for the current unstructured mesh, loaded in SMS.

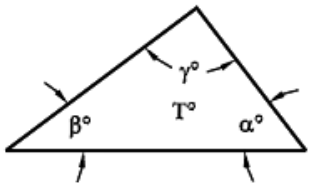
The plot includes the ARR region (defined below), a point for each element in the mesh, and three contour lines (0.3 in red, 0.45 in yellow and 0.6 in green) of the currently selected element quality measure (also defined below). As a general rule, elements with quality lower than 0.3 should be reviewed and improved (mesh editing) if possible.

The user can click on any point in the plot to see the element ID associated with that point and the six quality measure values for that element.

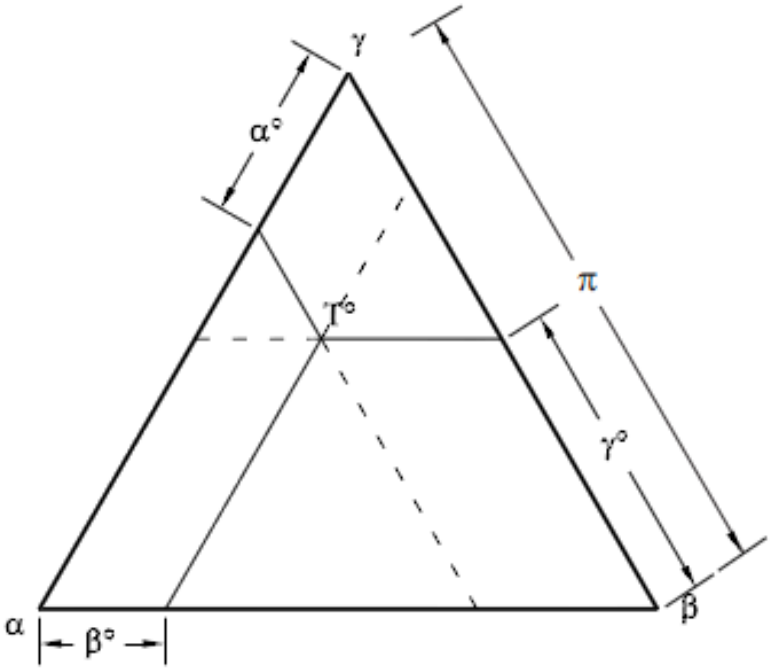
Once the mesh is edited in any way, the user can update the ARR plot by right-clicking in the plot and selecting **Refresh**. Until this is done, the plot will continue to reflect the mesh that existed when it was generated (or most recently refreshed).

This plot is based on a the publication in Communications in Numerical Methods in Engineerings, Volume 19 (2003) pp 551-561 by J. Sarrate, et. al. entitled "Numerical representation of the quality measures of triangles and triangular meshes". Several of the figures below are derived from this paper.

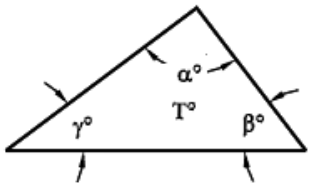
To assess the quality of a triangular mesh, such as those used by ADH or ADCIRC, the quality of each element is represented as a point, based on the interior angles of that element. These interior angles are labeled α, β, and γ as shown:



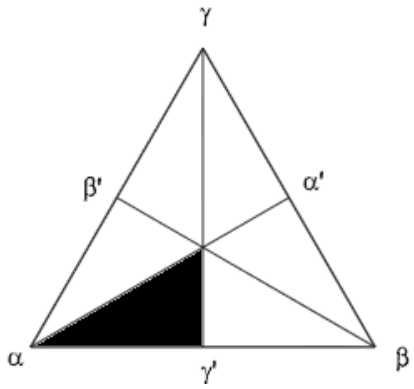
We can plot these three angles into an equilateral triangle



If we order the three angles so that α > β > γ as shown:



All of the points, will fall into the shaded portion of the equilateral triangle. This is referred to as the ARR region.



The quality of the elements is further assessed by computing a quality measure from attributes of the triangle. These attributes include:

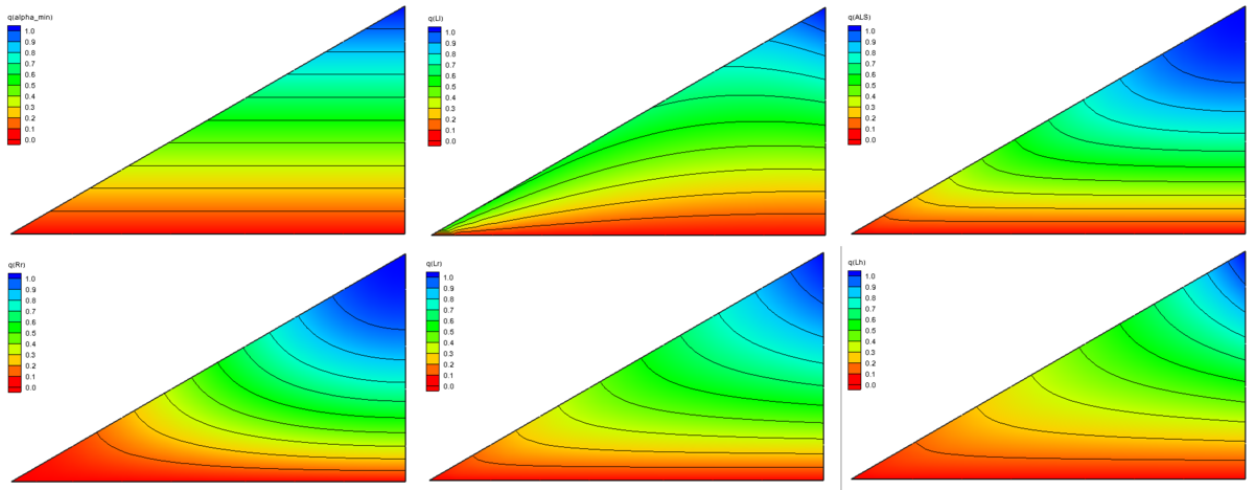
- The minimum interior angle α_{min} (γ from previous figure).
- The lengths of edges.
- The triangle area.
- The inner and outer radius.
- The minimum distance through the triangle (h_{min}).

These measures vary from 0.0 at the edges of the equilateral triangle to 1.0 at the center. The measures supported by SMS include:

$$q_{x_{min}} = \frac{3x_{min}}{\pi},\ q_{Ll} = \frac{l_{min}}{l_{max}},\ q_{ALS} = \frac{4\sqrt{3}A}{l_1^2 + 1_2^2 + 1_3^2}$$

$$q_{Rr} = \frac{2r}{R},\ q_{Lr} = \frac{2\sqrt{3}r}{l_{max}},\ q_{Lh} = \frac{2h_{min}}{\sqrt{3}l_{max}}$$

The following figures show how each of these quality measures cover the ARR



Related Topics

- Plot Window

2.2. Animation(Film Loop)

Animations

At a glance

Flow trace animation

- Several types of AVI animations (film loops) can be generated by SMS
- Transient data animation shows model changes through time (contours, vectors, etc)
- Flow trace uses vector data to generate flow paths through the geometry
- Drogue plots use user specified starting locations and show how the particles would flow through a vector field
- Multiple view animations show the data while transitioning between different views
- Plot window animations show plots changing through time

Animations in SMS provide a powerful tool for visualizing solution data.

Film Loop Setup Wizard

To create an animation select *Data* | **Film Loop** to open the *Film Loop Setup* wizard. The pages in the *Film Loop Setup* wizard include:

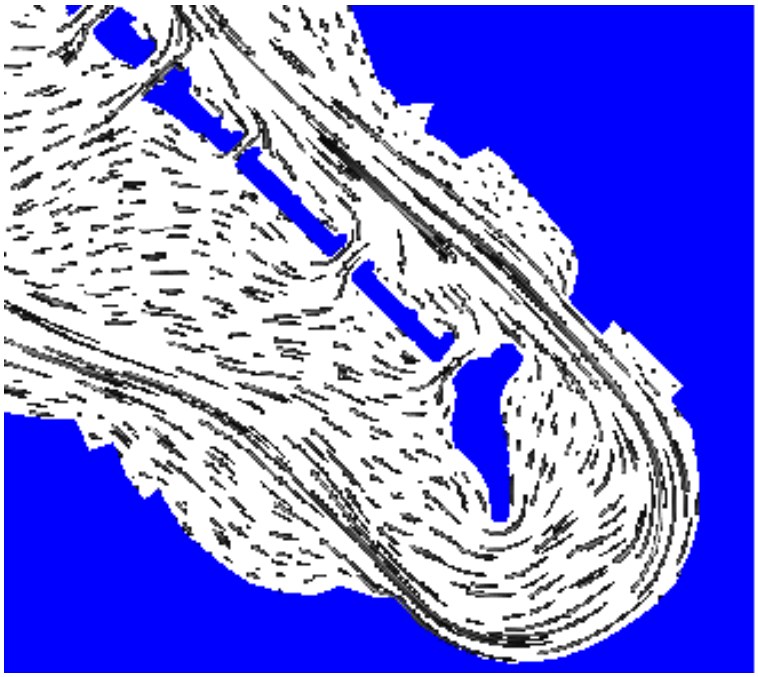
- General Options
- Display Options
- Time Step Options
- Multiple Views
- Drogue Plot Options
- Flow Trace Options

Animation Types

Flow Trace Animations

Flow trace animation is a technique used to visualize vector fields in SMS. It can be thought of as dropping tiny drops of dye into a fluid field in a random distribution and watching the flow pattern created. The process can also be thought of as creating particles of zero mass, and letting the vectors in the vector field be forces pushing the particles around. The *Flow Trace* portion of the *Film Loop Setup* dialog allows the user to control the flow trace. This entire

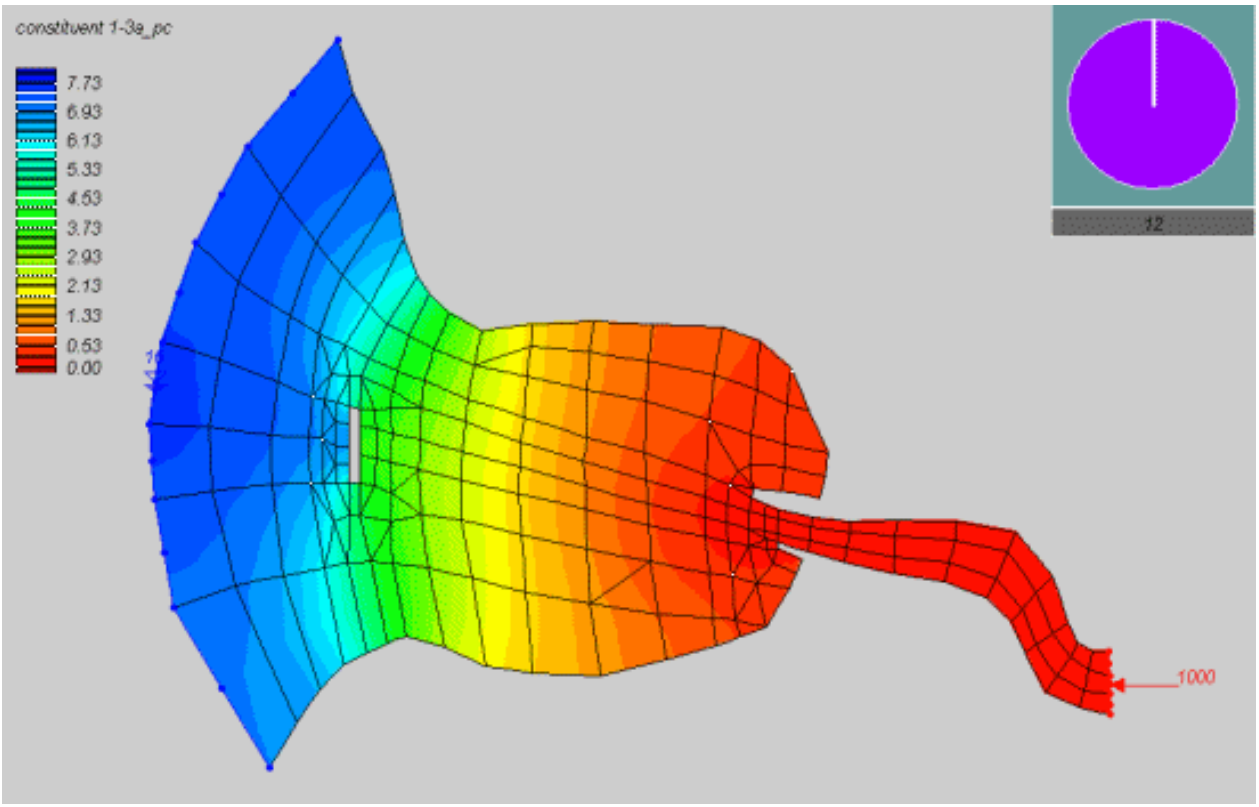
portion of the dialog is disabled if no vector data exists for the current data set. The top radio group allows the user to specify whether the flow trace should be created for a steady state or dynamic system. Below this the user can specify the density of particles or dye droplets by specifying the average number of particles for each cell or element. The number of frames required for a droplet to become dispersed is represented as a portion of the animation in the *Decay ratio* field.



The path of each particle is defined by tracing the particle. A starting position is defined randomly in the mesh or grid. Successive particle locations are computed by applying the forces of the vector field to the current location. At the new point, the velocity and direction are sampled. If the particle has traveled farther than the Flow trace length limit, or the velocity has changed more than the Velocity difference limit, the step is broken into two steps of half the step size. This process is repeated, until a sequence of valid points within the limits are defined for each frame. Therefore, the smaller the values of the Flow trace length limit and Velocity difference limit, the more precisely the particles will imitate the vector field. Generally, the default values are sufficient.

The Average particle speed is used to scale the vector field, thus changing the distance each particle or droplet travels. This is useful for vector fields with extreme magnitudes. For a low magnitude data set, the particles may not move very far. While this sluggish motion is accurate for the data, scaling the vector field up, and exaggerating the motion causes the flow patterns to be more visible. Similarly, in high magnitude fields the particles may become long streaks and scaling the values down may result in a clearer picture of the flow patterns.

Transient Data Animations



Animation Clock

Since animations are simulating the passage of time, it is natural to display a clock, which indicates the time reference for each frame of the animation. The *Display Clock* toggle controls whether a clock will be displayed. The **Options** button brings up the *Legend Options* dialog with a control to specify a digital clock face or analog.

Animation Time Control

Animation can be applied to any object with a dynamic dataset. The user defines the beginning and ending time for the animation sequence and the time step between subsequent frames. As each frame is generated, data values corresponding to the current time are loaded into memory and the image is redrawn using the current display options. The display options may be modified while setting up an animation using the display options button in the *Display Options* portion of the dialog.

The strip in the center of the *Data Options* portion of the *Film Loop Setup* dialog displays the allowable time values for the current data function(s) and the selected range to be animated. The user can select a time range to animate graphically on this scale, or explicitly in the edit fields below the time step strip. The legal time range displayed in the strip is based on the current scalar and vector data set(s). SMS allows animation of only scalar or vector data while the other remains constant. This normally is only used when a static field such as elevation is displayed with a varying velocity field or a static velocity field is displayed over a changing scalar field such as constituent dispersion or sediment deposition.

The total number of frames generated in the film loop can be defined by either matching the time steps (one frame per time step) or by using a constant interval (e.g., one frame for every two-hour interval). If the *Match Time Steps* option is chosen, extra frames can be created between each time step using linear interpolation of the data values at the specified time steps.

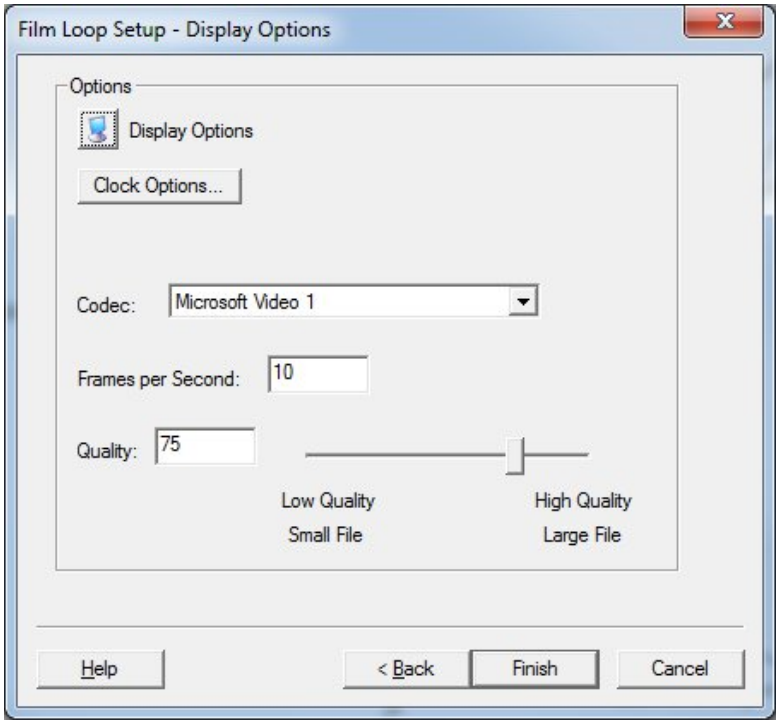
Related Topics

- Visualization

Film Loop Display Options

This page of the *Film Loop Setup* wizard allows you to set up the film loop clock options. You can place the clock on any corner of the screen, set its size, and set the font that is used for the digital clock. In SMS version 9.0, you can set additional options for the clock position, progress bar and clock style.

This page also gives you access to the SMS *Display Options* property sheet. These options only affect the display of *Scalar/Vector Animations*.



Film Loop Setup – Display Options dialog in the *Film Loop Wizard*

AVI Codecs

Starting in SMS 10.0 you will be able to choose what Codec you use to create your AVI movie. SMS will search your computer for all compatible codecs and they will be available in the pull down menu. The choice of codec will determine both the quality and size of the resulting avi file.

FFDSHOW Video Codec

Ffdshow Video Codecs create a much sharper, smoother image than the SMS default codec (Microsoft Video 1), and is therefore more desirable to use when making film loops. ffdshow has an extensive list of codecs from which you can choose. We have had good success with the Divx codec. The H.263+ codec creates nice animations but they will not play within PAVIA (default for playing vidoes from SMS). See below for some ideas of alternate video players. The H.264 codec is popular but doesn't always work.

Installation

In order to use an ffdshow codec, you must first download ffdshow. If you installed the 32-bit version of SMS, ffdshow was installed unless you turned off the option during installation.

If you don't have it installed, to download ffdshow, go to http://ffdshow-tryout.sourceforge.net [1].

- Make sure that the program will be saved on your local disc (C:) program files.
- During installation, be sure to specify that you want the VFW interface.

Using FFDSHOW

There are two steps to using the ffdshow codecs for animations generated in SMS. First you have to use the ffdshow program to set the options for the video encoder. Secondly, you choose the ffdshow codec in the filmloop wizard from inside SMS. Detailed steps are given below:

- Once installed, click on the windows *Start* button and search for *VFW Configuration*. Click on it.
- The *ffdshow video encoder configuration* dialog will appear.
- Set the Encoder box to which ever codec you would like to use. (Some codecs will not function because of incompatible requirements with the SMS filmloop generation code). Options of the corresponding Fourcc will appear in the Fourcc box. Pick one and Click the 'Apply' button. Click 'OK' to close the dialog.
- Once the Codec type is set, go to the *Film Loop Setup - Display Options* dialog in SMS (Data → Filmloop) and change the Codec type to 'ffdshow Video Codec.'

  Note: Any codec that you use needs to be on the computer that will be playing the animations. If you want to move your avi's to a different computer, make sure that the computer you are moving them to has ffdshow. Otherwise, you will have to download ffdshow to play the animations.

Alternate Video Players

- KMPlayer [2]
- VLC [3]
- Windows Media Player [4] – Doesn't have much to control playback of videos so the others might be better options.

Related Topics

- Animations

References

[1] http://ffdshow-tryout.sourceforge.net/
[2] http://kmplayer.kde.org/
[3] http://www.vlcmediaplayer2011.com/
[4] http://windows.microsoft.com/en-US/windows/products/windows-media-player

Film Loop Drogue Plot Options

This page of the *Film Loop Setup* wizard allows you to set up color options that pertain only to drogue plots. A color ramp can be set up to display points in a color based on either its current velocity or the total distance it has traveled. The minimum value is always zero so the maximum value defines the range to be used for the specified color ramp. You may need to experiment with this maximum value before you get something that you like for a specific model. The head of each particle can be from one (1) to six (6) pixels in size. The maximum tail length is specified in hours and can fade to black or remain solid. If you don't want to have a tail, then set the fade time to zero (0.0).

You can specify the background to be either a solid color or an image that you have already opened and registered. If you don't have an image open, then that option is not available. The background of the model domain is always black.

The final option on this page allows a statistical report to be written while the particles are computed.

Related Topics

- Animations
- Particle Report

Film Loop Flow Trace Options

This page of the *Film Loop Setup* wizard allows you to set up options for the particles in the flow trace. The following options are available:

- **Particles per object –** Increasing this value increases the total number of random particles that get created and distributed throughout the domain. For a finite element mesh, the number specified is multiplied by the number of elements to determine the number of particles to be distributed over the domain.
- **Decay Ratio –** This defines how quickly the particle's tail decays and should be a value between zero (0.0) and one (1.0). A larger value produces particles with longer tails. A value of 1.0 indicates that it will take 100 percent of the film loop time for a end of a tail to fade away.
- **Average particle speed –** This provides a means of magnifying or reducing the activity in the domain. The particles will be traced through the domain with the velocity of the current vector functional dataset. The velocity is assumed as pixel space units. No time match between distance on the screen and velocity is attempted because the time between flow trace frames is not explicit. If an exact velocity in distance is desired, the user must compute the scale value and specify it here. Normally, for visualization purposes, experimentation with this parameter will generate the desired results. Another option would be to use the drogue animation tools.
- **Flow trace length limit –** This specifies a maximum distance the particle can travel in a single numerical integration step. When computing the numerical integration, if a particle travels more than this distance, the integration step is reduced to produce a more accurate particle path. Decreasing this number causes slower integration, but a more accurate path.
- **Velocity difference limit –** This specifies a maximum change in speed the particle can experience in a single numerical integration step. When computing the numerical integration, if a particle speed changes more than this limit, the integration step is reduced to produce a more accurate particle path. Decreasing this number causes slower integration, but a more accurate path.

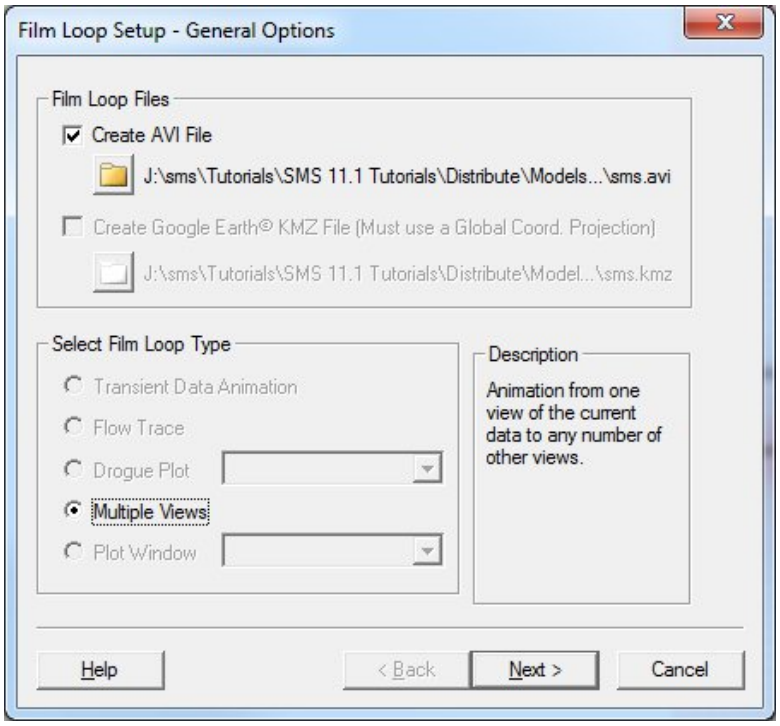
SMS will use the current background as the background for the drogue plots. The model domain is always black and particles are always white. These two options cannot be changed.

Related Topics

- Animations

Film Loop General Options

Overview



Film Loop Setup – General Options dialog in the *Film Loop Wizard*

The *Film Loop Setup* wizard is used to create the following types of video animation files:

- AVI, or audio video interleave [1] (*.avi)
- Google Earth© KMZ (*.kmz) – See Google Earth© KMZ file export requirements section below.

The wizard is invoked by choosing the **Film Loop** option from the *Data* menu. When the wizard is successfully completed, the animation is generated according to the specified options. The animations are then opened and displayed. AVI files are displayed in the Play AVI Application (PAVIA). Google Earth© KMZ files are displayed in Gooogle Earth©. the Play AVI Application is included with SMS. Google Earth© must be downloaded and installed separately. See http://earth.google.com/ [2] for information on obtaining Google Earth©.

The *General Options* page allows you to specify the following:

- File name for each video animation file type being exported
- Film loop type
  - **Transient Data Animation –** To use this option, you must have opened a dynamic solution file. This will show how contours and/or vectors change with time by displaying a sequence of images, one for each time step.
  - **Flow Trace –** To use this option, you must have an available vector data set, such as velocity. This animation randomly distributes particles throughout the domain and shows their path through time.
  - **Drogue Plot –** To use this option, you must have an available vector data set, such as velocity, and you must have created points and/or arcs in a Particle/Drogue coverage with the Map module. This option is similar to the Flow Trace, except that particles are initially placed at feature points and at each vertex of feature arcs in the selected coverage.
  - **Multiple Views –** This option creates an animation of a single time step from one rotated view to another. A viewing path is created with any number of bearing/dip pairs. Multiple View film loops can not be exported to a Google Earth© KMZ file.
  - **Plot Window –** This option allows you to animate a plot window, such as how a functional value across an observation arc changes through time. Plot Window film loops can not be exported to a Google Earth© KMZ

file.

Google Earth© KMZ file export requirements

The following requirements must be met to export Google Earth© KMZ file

- Must be in plan view
- Must use a Global Coordinate Projection (not local)
- Film loop types which can be exported to KMZ
  - Transient data animation
  - Flow trace
  - Drogue plot
- Film loop types which cannot be exported to KMZ
  - Multiple Views
  - Plot window

Play AVI Application (Pavia)

Controls exist within the application to play, stop, and step the animation. When an animation is stopped on a frame, the frame's image can be copied to the clipboard using the Copy button. The pavia.exe file is a separate application and can be copied to other computers to display existing animations on computers that do not have SMS installed. This is useful for displaying animations during a presentation or sending the animation to another individual or entity. You do not have to close the pavia application before creating another animation in SMS, but you will not be able to create an animation with the same file name as an opened animation.

Related Topics

- Animations

External Links

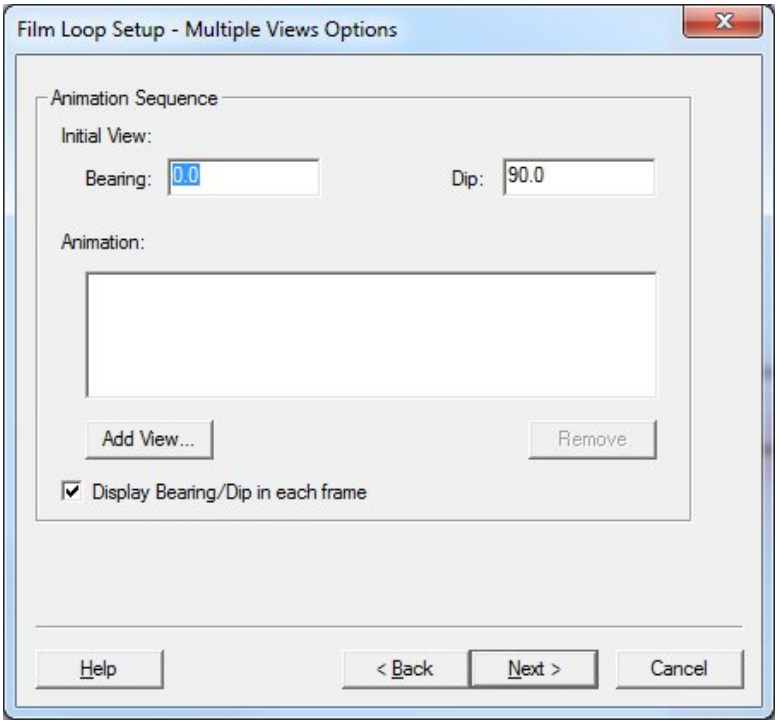
- Google Earth© Application [2]

References

[1] http://en.wikipedia.org/wiki/Audio_Video_Interleave
[2] http://earth.google.com/

Film Loop Multiple Views

This page of the *Film Loop Setup* wizard allows you to define the view path to be traversed for the animation. By default, the initial view is set to the window's current bearing and dip. You can add any number of views with any number of steps to the view. For your convenience, the bearing/dip pair can be displayed inside each frame of the animation.



Film Loop Setup – Multiple Views Options dialog in the *Film Loop Wizard*

Related Topics

- Animations

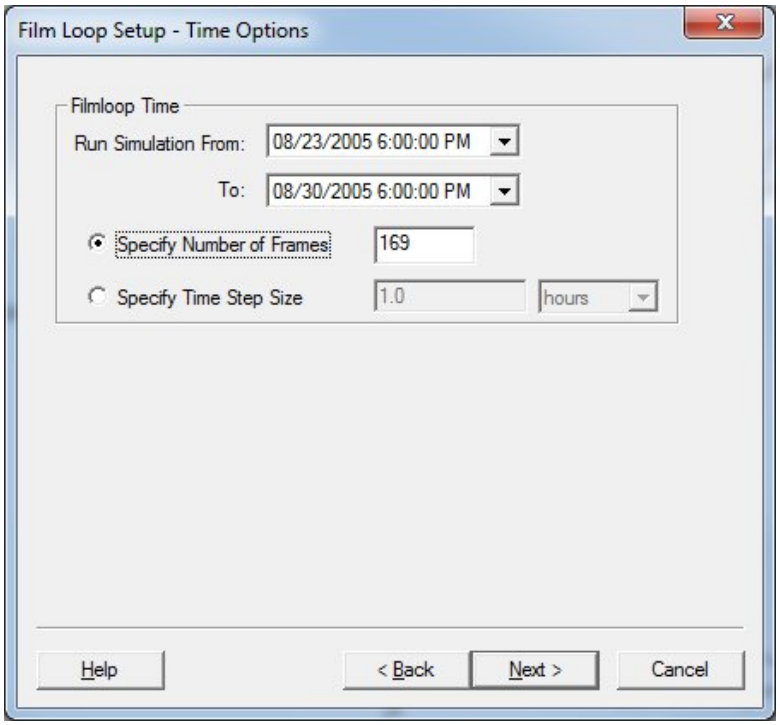
Film Loop Time Step Options

The *Time Options* dialog is used to specify the time range and time step to use in the animation. It is the second step of the *Film Loop Setup* wizard when exporting the following film loop types:

- Transient Data Animation
- Flow Trace – if using transient data
- Drogue Plot – if using transient data

The following time options are available:

- Film loop start time
- Film loop duration
- Number of Frames or Time Step Size



Film Loop Setup – Time Step Options dialog in the *Film Loop Wizard*

If the Time Settings are set to display as *Relative Time*, the zero time can be changed. If exporting to a KMZ file, the time zone can also be specified. Specifying the time zone is required if the model time is in a local time zone (as opposed to UTC) for Google Earth to display the correct times for the associated temporal data.

Related Topics

- Animations

2.3. Coordinate Systems

Coordinate Systems

XMS programs work in a single coordinate system. The user selects what system will be used using the *Edit* | *Coordinate System* menu command.

Both a horizontal and vertical system are specified. Many numerical models work in a local system, so there may not be a problem if the user does not know what coordinate system the data is referenced to. If the desired model requires a global system, or the base data is referenced to more than one system, the modeler must determine the coordinate systems involved.

Since data can be gathered and referenced to various coordinate systems, XMS allows the user to convert data from one coordinate system into another. The Coordinate Conversions page describes how to do this. The available systems and units include:

Horizontal System

The options to the right of the dialog change as the horizontal system is changed. For example, the hemisphere is required for a Geographic system and a UTM zone is required for a UTM system.

- Local
- Geographic
- UTM
- State Plane
- CPP

Local is the default horizontal and vertical system. Unless the user desires to convert data to another system, it is not necessary to change the system from *Local*.

Horizontal Units

The horizontal units can be specified for all systems except Geographic, which uses decimal degrees. **The units set here are the units used by the models such as RMA2**.

- U.S. Survey Feet
- International Feet
- Meters

Ellipsoid

The Ellipsoid options are only available for non-NAD/HPGN/CPP systems. The ellipsoid can be selected for a region of the world. Changing the ellipsoid changes the minor and major radii of the earth, measurements used in performing conversions. To set the radii for an ellipsoid not included in XMS, select the *User Defined* ellipsoid and a button appears titled *Define*. Clicking on the button brings up a dialog where the radii are defined. The Major and Minor Radii (a and b, respectively) are defined in meters only. The Minor Radius can be input directly or defined by specifying the ellipsoid flattening (1/f) or eccentricity squared (e2) variables, where

$$\frac{1}{f} = \frac{(a-b)}{a} \quad \text{and} \quad e^2 = 1 - \frac{b^2}{a^2}$$

Vertical System

The supported vertical options are for North America. If a non-NAD/HPGN horizontal system is selected, this option is disabled.

- Local
- NGVD 29
- NAVD 88

Vertical Units

The vertical unit is only used for conversions. Area and volume calculations performed by XMS use the horizontal units.

- U.S. Survey Feet
- International Feet
- Meters

External Links

- Introduction to Tidal and Geodetic Vertical Datums and Datum Computations [1] – Presentations available here [2]

References

[1] http://www.ngs.noaa.gov/corbin/class_description/Tidal_Geodetic_Datums.shtml
[2] ftp://ftp.ngs.noaa.gov/pub/corbin/datum_training/

Coordinate Conversions

Converting data from one coordinate system to another can be done using the *Edit*|**Coordinate Conversions** menu command. The command may also be accessed by right-clicking on a single entity (grid, mesh, scattered data set ...) in the Project Explorer. This allows the user to convert just that entity from one system into another (typically to the current project coordinate system).

All data will be converted from the system on the left of the dialog to the system on the right.

"Convert From" System

The "Convert From" system defines the coordinate system the data is currently referenced to. When working from the *Edit* menu, this is the system XMS is working in and is dimmed by default because it is assumed you have already specified this system. When working from the Project Explorer, this is the coordinate system of the selected entity and must be selected.

"Convert To" System

The "Convert To" system defines the system you will be working in after the conversion. When the dialog is invoked from the *Edit* menu, this is selected by the user and all data is converted from the current system to this new system. When the dialog is invoked from the Project Explorer, the "Convert To" system is dimmed because this is assumed to be the system XMS is working in and all other data is already in this system.

Restrictions

Some conversions are not allowed, such as converting between a NAD and non-NAD system. A warning is issued when conversions are not allowed.

CPP Coordinate System

A CPP (Carte Parallelo-Grammatique Projection) system is a local system. The origin of the system must be defined in latitude/longitude decimal degrees.

The conversion from of a point from latitude/longitude to CPP is:

$$\text{newpoint}_x = R * \left(\text{point}_{\text{longitude}} - \text{origin}_{\text{longitude}}\right) * \cos\left(\text{origin}_{\text{latitude}}\right)$$

$$\text{newpoint}_y = \text{point}_{\text{longitude}} * R$$

The conversion of a point from CPP to latitude/longitude is:

$$\text{newpoint}_{\text{longitude}} = \frac{\text{origin}_{\text{longitude}} + \text{point}_x}{R * \cos\left(\text{origin}_{\text{latitude}}\right)}$$

$$\text{newpoint}_{\text{latitude}} = \frac{\text{point}_y}{R}$$

$R = 6378206.4m.$(Clarke 1866 major spheroid radius)

Geographic Coordinate System

A Geographic system is a latitude/longitude system defined in decimal degrees. Supported Geographic systems include:

- NAD (North American Datum) 1927 and NAD 1988
- 33 world ellipsoids and a user defined ellipsoid (i.e., Clarke 1866, WGS 1984, etc.)

The hemispheres are defined for non-NAD systems. The hemisphere cannot be changed for NAD systems (Northern, Western hemispheres).

Local Coordinate System

A local coordinate system is a system defined for a survey. Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). If a portion of the data for a project is referenced to a global coordinate system, and a portion is in a local system, a transformation to convert all the data to a single system must be defined. This can be done by defining the location and orientation of the origin of the local system with reference to the global system. This allows the data to be referenced back to the global system or to another local system.

Three scenarios are possible when dealing with local coordinate systems:

1. Global to Local – When converting from a global to a local coordinate sytem, the global coordinates of the origin of the local coordinate system must be defined using the Local Origin button in the lower portion of the dialog. This dialog can also be used to enter the angle of rotation of the local coordinate axes relative to the global coordinate axes. The angle is measured ccw from the positive x-axis.
2. Local to Local – When converting from one local coordinate system to another local coordinate system, the same approach is used as when converting from a global to local coordinate sytem. The Local Origin dialog is used to define the coordinates of the origin of the new coordinate system relative to the old coordinate system.
3. Local to Global – When converting from a local to a global coordinate sytem, the Local Origin dialog is used to coordinates of the local coordinate system (which is the old system in this case) relative to the new global coordinate sytem.

Single Point Conversion

The **Single Point Conversion** dialog allows a user to convert a coordinate (xyz point) between to coordinate systems. This dialog can be accessed from two locations within XMS programs:

1. *Edit* menu, *Convert Single Coordinate* item
2. Register Image Dialog

The dialog consists of two sides. On the left side, the original (or Convert from) coordinate system is entered. On the right side, the final (or Convert to) coordinate system is entered. The options for the two sides are described on the Coordinate Systems page.

The other items in the dialog include:

- **Enter coordinates –** Enter the original (or from) coordinates.
- **New Coordinates –** View the final (or to) coordinates.
- **Convert –** Perform the conversion.
- **Create Mesh Node/Feature Point –** Create a mesh node (if in the mesh module) or a feature point (if in the map module) at the final coordinates when OK is pushed.

Related Topics

- Coordinate Systems
- Coordinate Conversions

2.3.a. Projections

Projections

Related Versions	
GMS	v7.0
SMS	v10.1
version note	

"Projection" refers to a map projection like UTM [1]. In XMS software, a projection can be associated with a project, and data can be reprojected from one projection to another. XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

Previous XMS software versions referred to projections as "coordinate systems" and reprojection as "coordinate conversion".

Project Projection

The current projection, or the projection currently associated with the project, can be specified via the *Edit|Projection* menu command. Changing the projection does not alter the XYZ coordinates of the project data.

Local Projection

Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). XMS software allows for local projections that are unrelated to any standard projection.

Reproject

Reprojecting means to convert data from one coordinate system to another. For example, a 2D mesh representing the ground surface may have XYZ coordinates in a UTM system and they need to be converted to a State Plane system to be consistent with other data. Reprojecting usually results in the XYZ coordinates of the data changing, although conceptually the data is in the same place with respect to the Earth, just in a different coordinate system.

There are three basic reprojection tasks that you can do:

- Reprojecting the entire project from one system to another
- Reprojecting one object from one coordinate system to the project coordinate system
- Single point reprojection, which allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection.

Either operation brings up a dialog with two projections specified. On the left, the "Convert From" projection defines the projection the data is currently in. On the right, the "Convert To" projection defines the projection the data will be in after the operation.

Reproject everything

Reprojecting everything can be done by selecting the *Edit|Reproject* menu command. This will convert all the data loaded into the XMS application from one projection to another. In this operation, the "Convert From" projection (left side of the dialog) is dimmed by default. The user selects a new projection on the right side that all data in the application will be converted to. This operation changes the "current" projection that is being used by XMS to the newly specified "Convert To" projection.

Reproject object

This command is done on a specific geometric object (grid, mesh, scatter set, ...) by right-clicking on the entity in the Project Explorer. The object is reprojected from some projection into the current projection being used by the XMS application. In this operation, the "Convert To" projection (right side of the dialog) is dimmed by default because it represents the current projection used by the project. It is assumed that the desire is to reproject data from another projection into the current projection. The current projection is not changed in this operation.

Single Point Reprojection

Single Point Reprojection allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection. It also lets you create a feature point at the new location. This operation is accessed via the *Edit|Single Point Reprojection* menu command. It's also available in the Register Image dialog.

Restrictions

Some reprojections are not allowed, such as reprojecting between a NAD and non-NAD system. A warning is issued when the reprojection is not allowed.

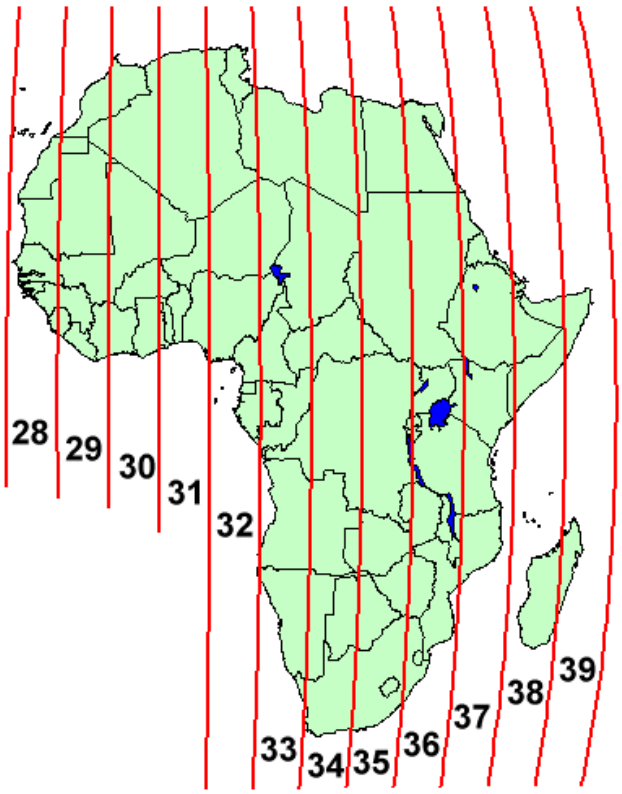
Supported Projections

XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

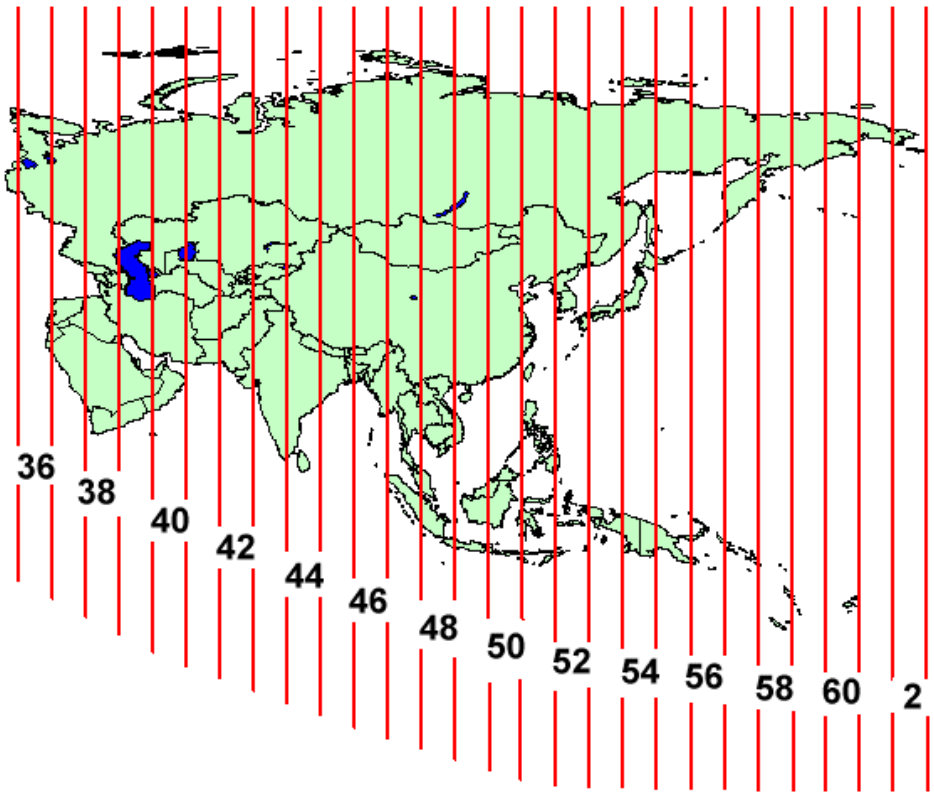
References

[1] http://en.wikipedia.org/wiki/Universal_Transverse_Mercator_coordinate_system
[2] http://www.globalmapper.com/

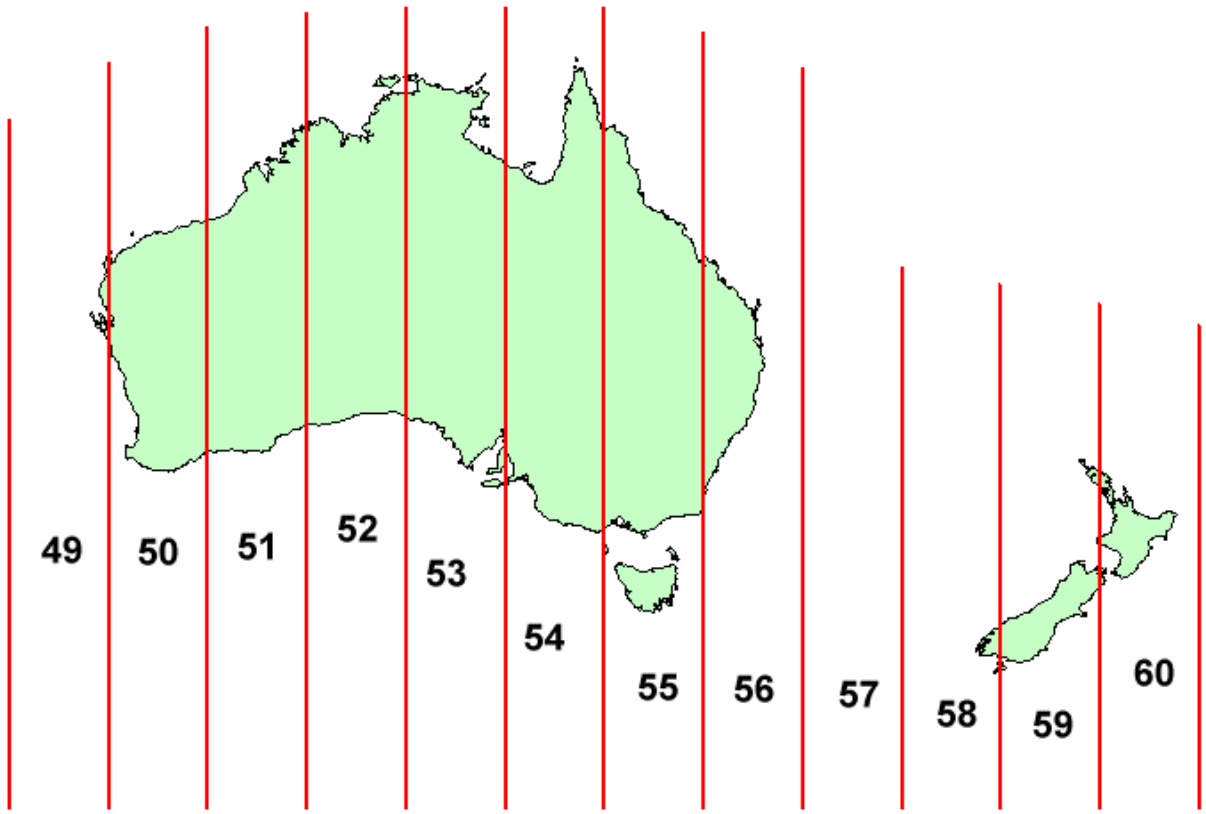
UTM Africa



UTM Asia



UTM Australia



UTM Coordinate System

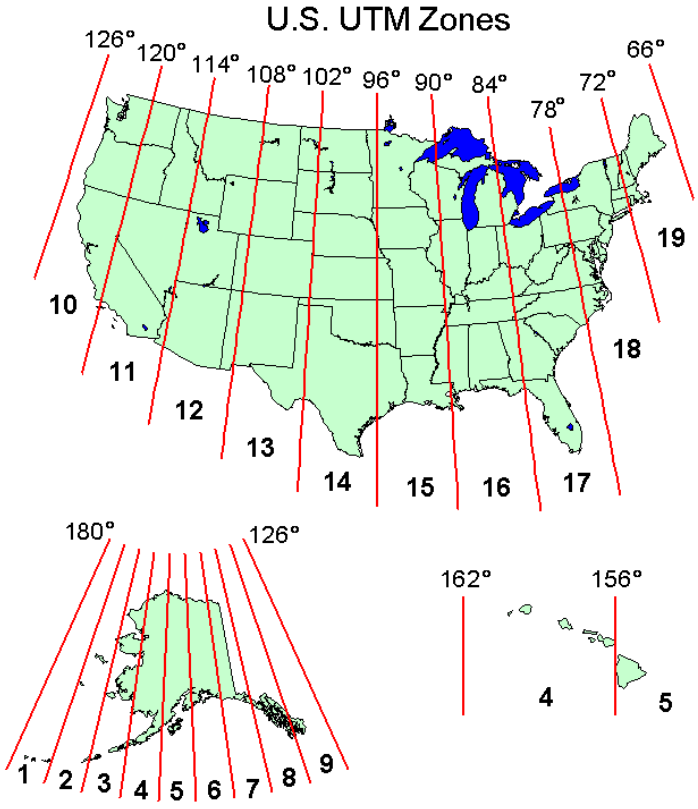
A UTM (Universal Transverse Mercator) system is a world-wide system defined in meters. The world is divided into 60 zones, 6 degrees of longitude, running from 84°N to 80°S latitude. Supported UTM systems include:

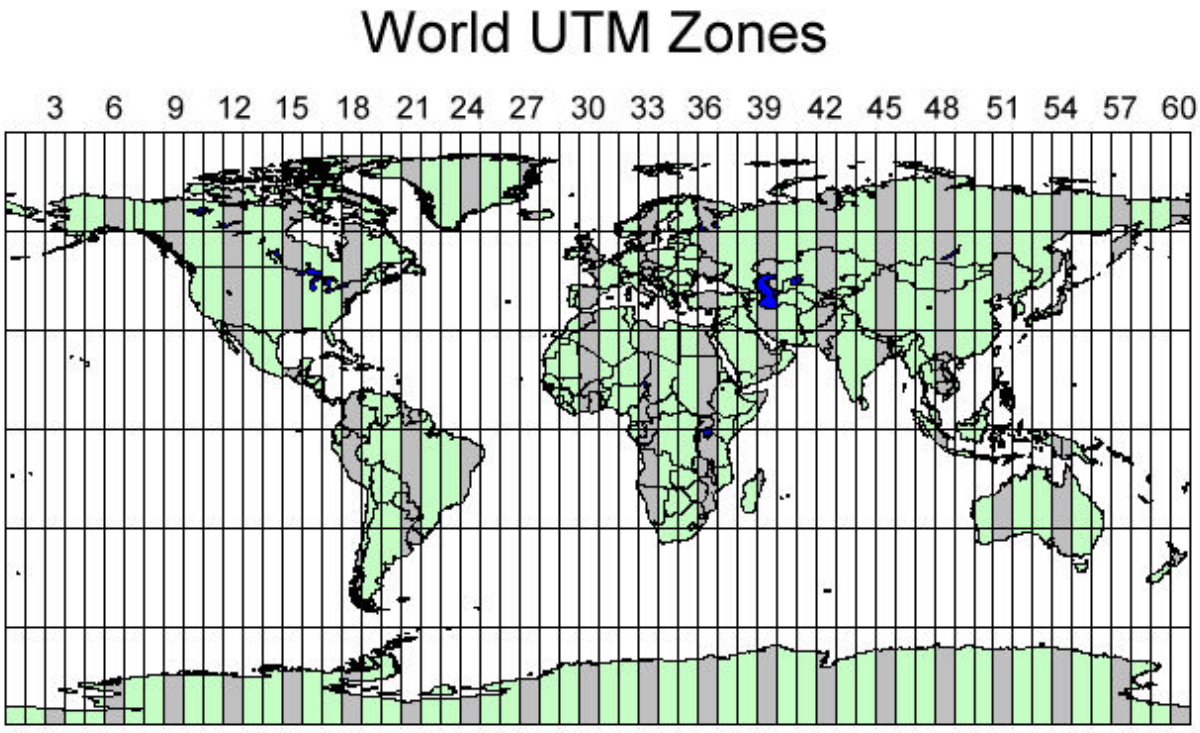
- NAD (North American Datum) 1927 and NAD 1983
- HPGN (High Precision Geodetic Network, now known as HARN - High Accuracy Precision Network)

The hemispheres are defined for non-NAD systems. The hemisphere cannot be changed for NAD systems (Northern, Western hemispheres). An additional HPGN zone must be defined for HPGN systems.

UTM zones for the southern hemisphere have a "false northing" of 10,000,000 meters at the equator with northings decreasing as you move south. This ensures all northings are positive in the southern hemisphere.

The US and World UTM Zones are shown below.





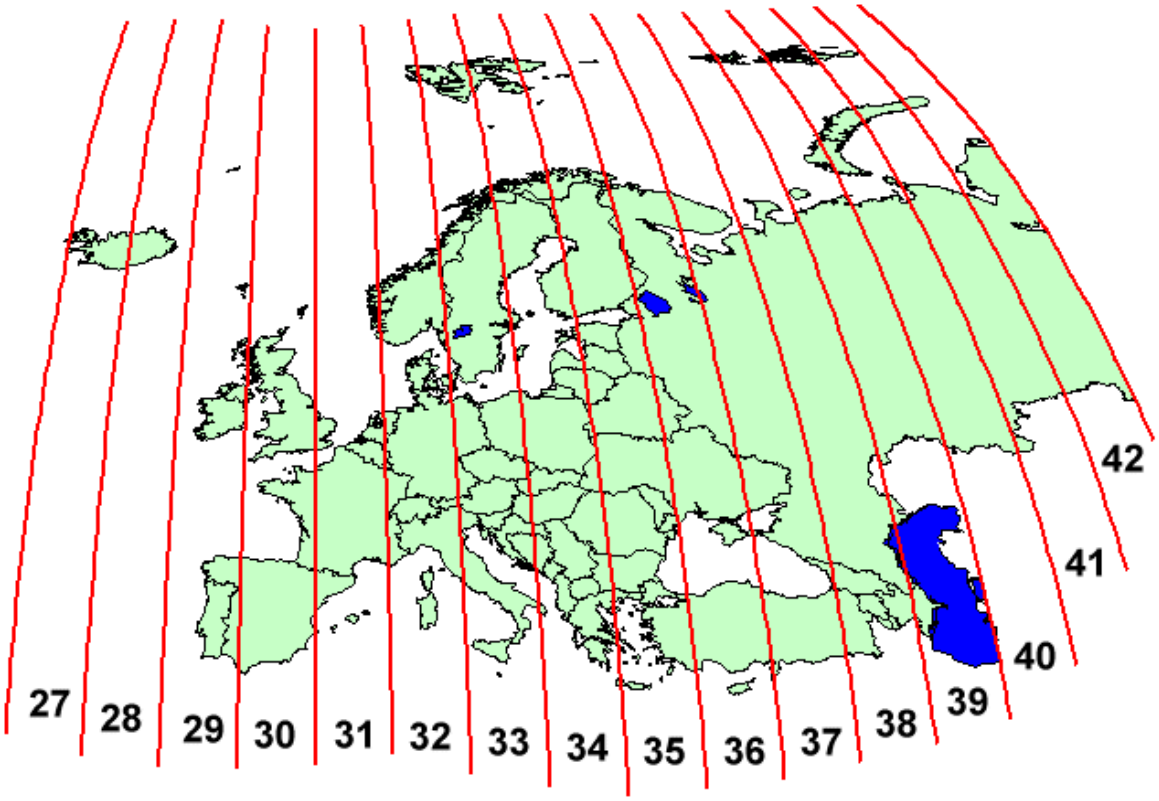
UTM Zones By Continent

- North America
- South America
- Africa
- Asia
- Europe
- Australia

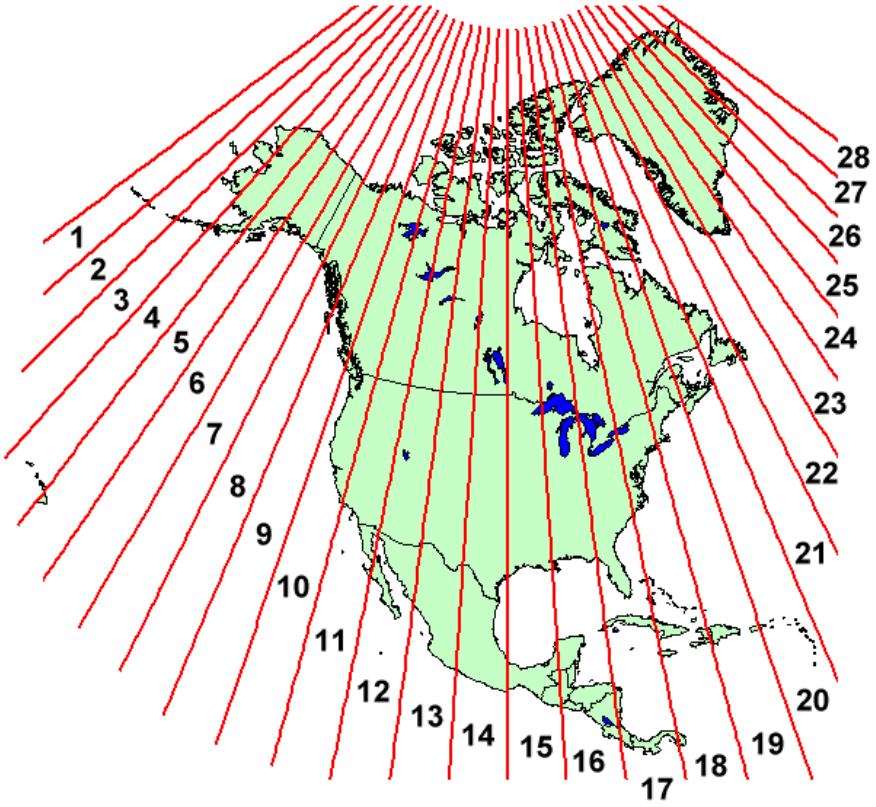
Related Topics

- Coordinate Systems
- UTM Coordinates [1]

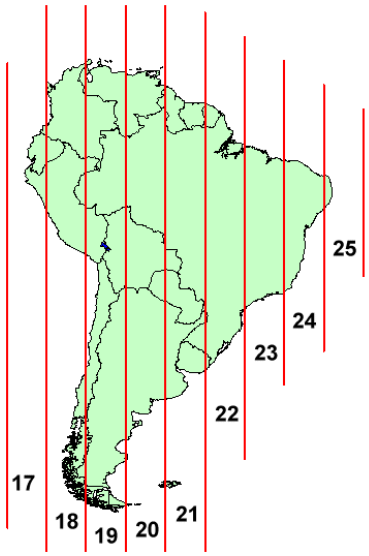
UTM Europe



UTM North America



UTM South American



2.3.b. State Plane Coordinate System

State Plane Coordinate System

A State Plane system is a coordinate system used in the US. Each US state is divided into one or more zones, known as State Plane zones. Supported State Plane systems include:

- NAD (North American Datum) 1927
- NAD 1983

Additionally, an HPGN (High Precision Geodetic Network, now known as HARN - High Accuracy Precision Network) zone can be specified for each state plane zone.

The boundary of most of the state plane zones remained the same from 1927 to 1983. The US State Plane Zones are shown in the map below. The boundaries are shown for each state plane zone by clicking on a region on the map. The boundaries that changed between 1927 and 1983 are highlighted for each state plane zone that changed.

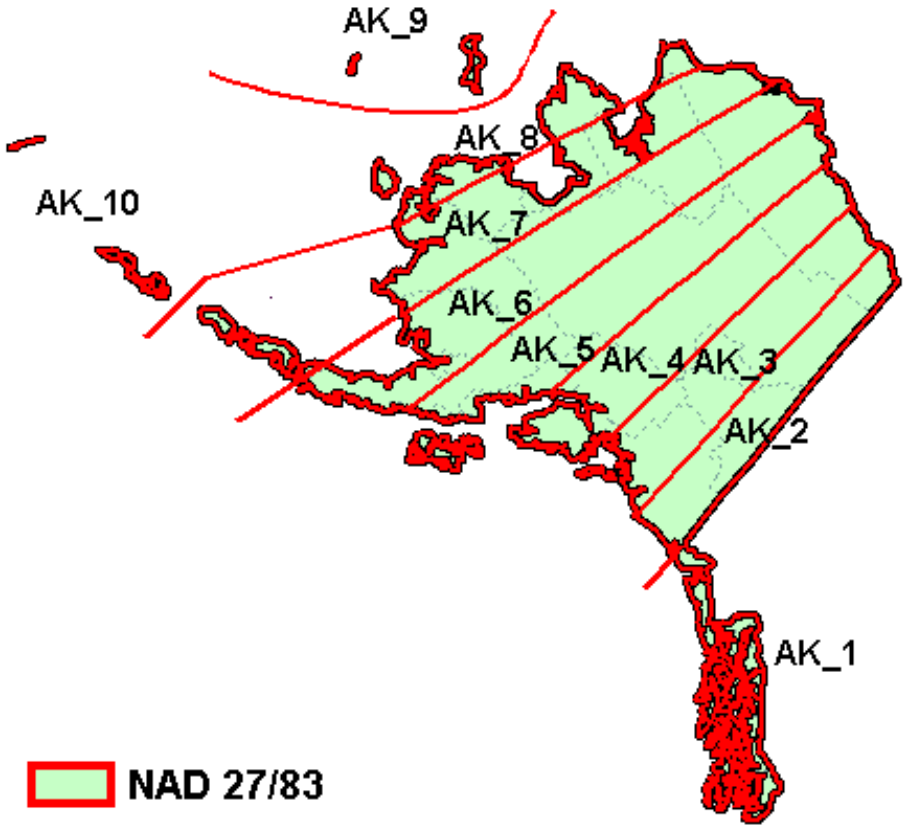
State Zone Maps

- Alaska
- Hawaii
- Mideast
- Midwest
- New England
- Northwest
- South Central
- South East
- Southwest
- Virgina Area

Related Topics

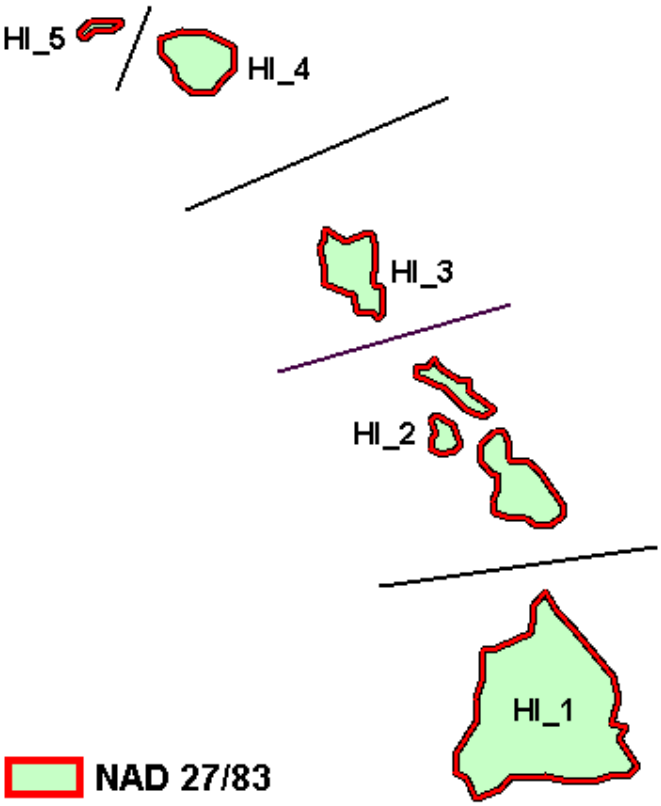
Coordinate Systems

Alaska State Plane



NAD 27 / 83		
Zone Name	Map Code	Zone ID
Alaska 1	AK_1	5001
Alaska 2	AK_2	5002
Alaska 3	AK_3	5003
Alaska 4	AK_4	5004
Alaska 5	AK_5	5005
Alaska 6	AK_6	5006
Alaska 7	AK_7	5007
Alaska 8	AK_8	5008
Alaska 9	AK_9	5009
Alaska 10	AK_10	5010

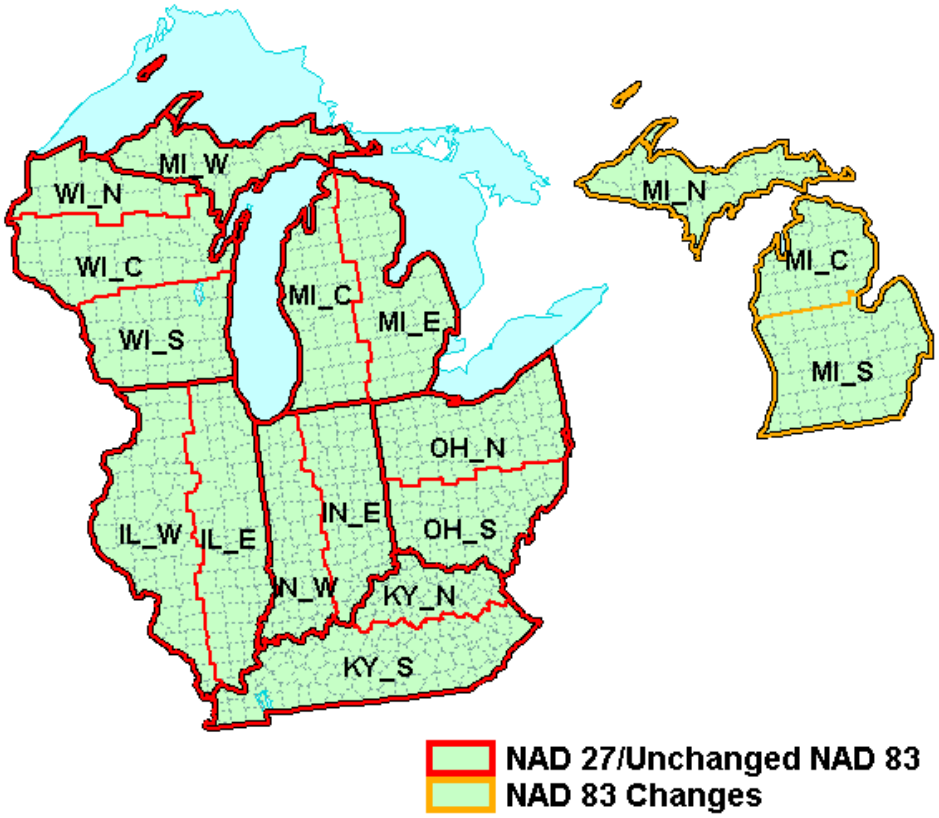
Hawaii State Plane



NAD 27 / 83

Zone Name	Map Code	Zone ID
Hawaii 1	HI_1	5101
Hawaii 2	HI_2	5102
Hawaii 3	HI_3	5103
Hawaii 4	HI_4	5104
Hawaii 5	HI_5	5105

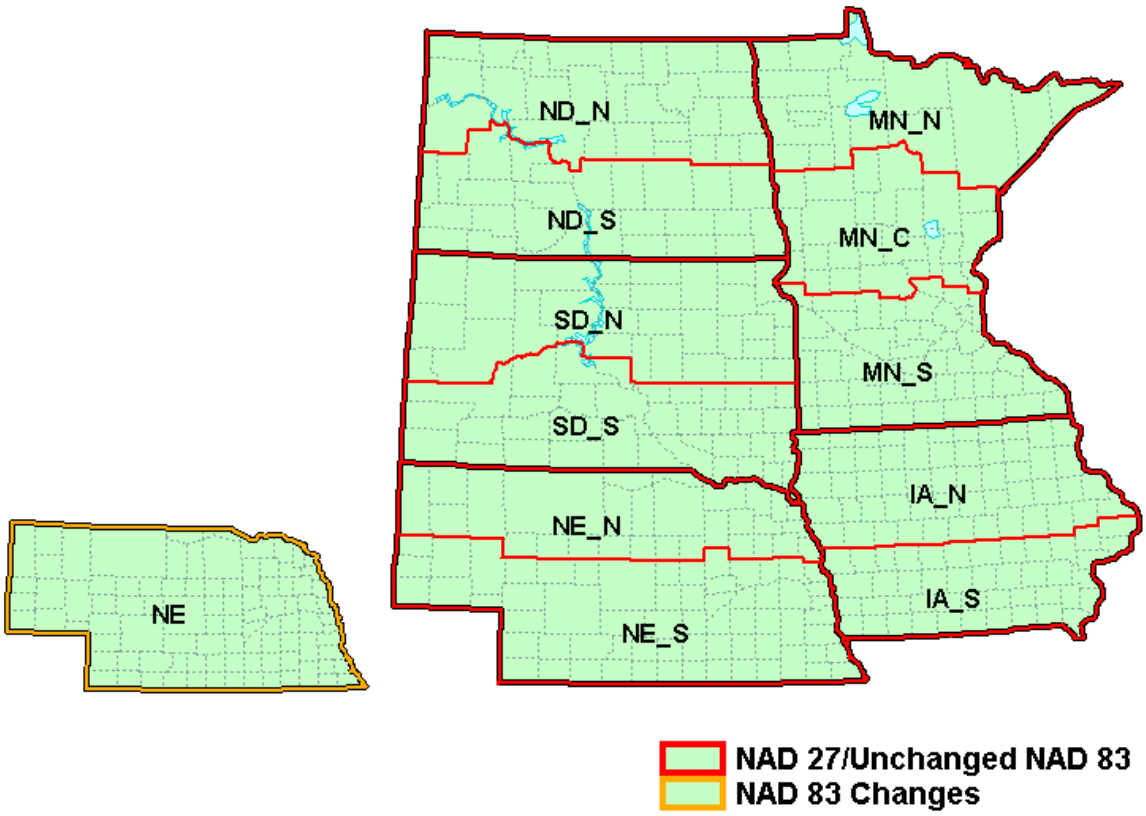
Mideast State Plane



NAD 27 / 83		
Zone Name	Map Code	Zone ID

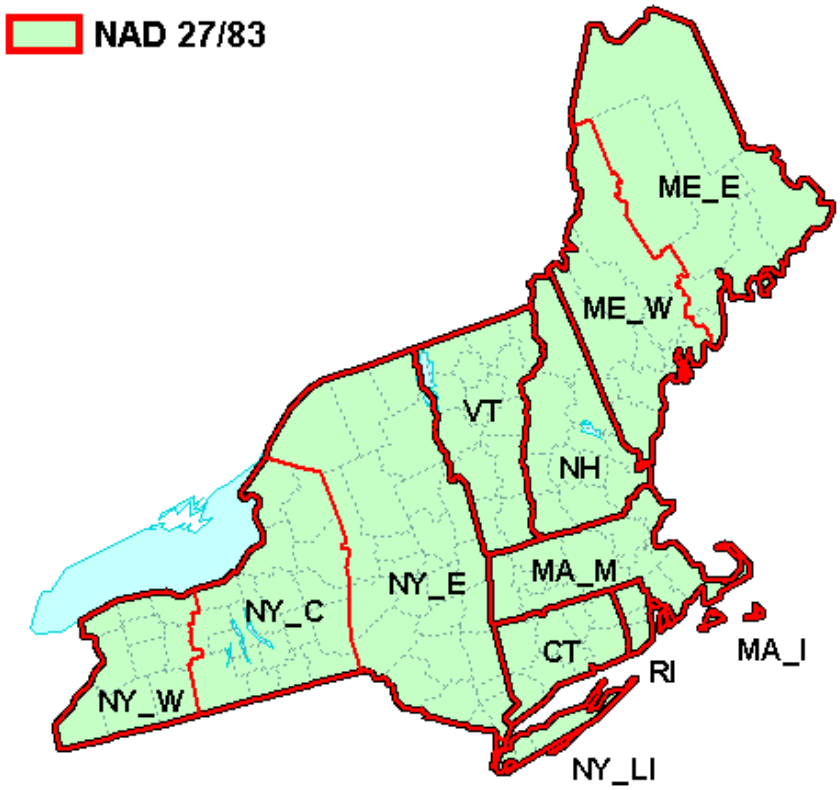
Illinois East	IL_E	1201
Illinois West	IL_W	1202
Indiana East	IN_E	1301
Indiana West	IN_W	1302
Kentucky North	KY_N	1601
Kentucky South	KY_S	1602
Michigan East (NAD 27)	MI_E	2101
Michigan Central (NAD 27)	MI_C	2102
Michigan West (NAD 27)	MI_W	2103
Ohio North	OH_N	3401
Ohio South	OH_S	3402
Wisconsin North	WI_N	4801
Wisconsin Central	WI_C	4802
Wisconsin South	WI_S	4803
NAD 83 Zone Changes		
Michigan North	MI_N	2111
Michigan Central	MI_C	2112
Michigan South	MI_S	2113

Midwest State Plane



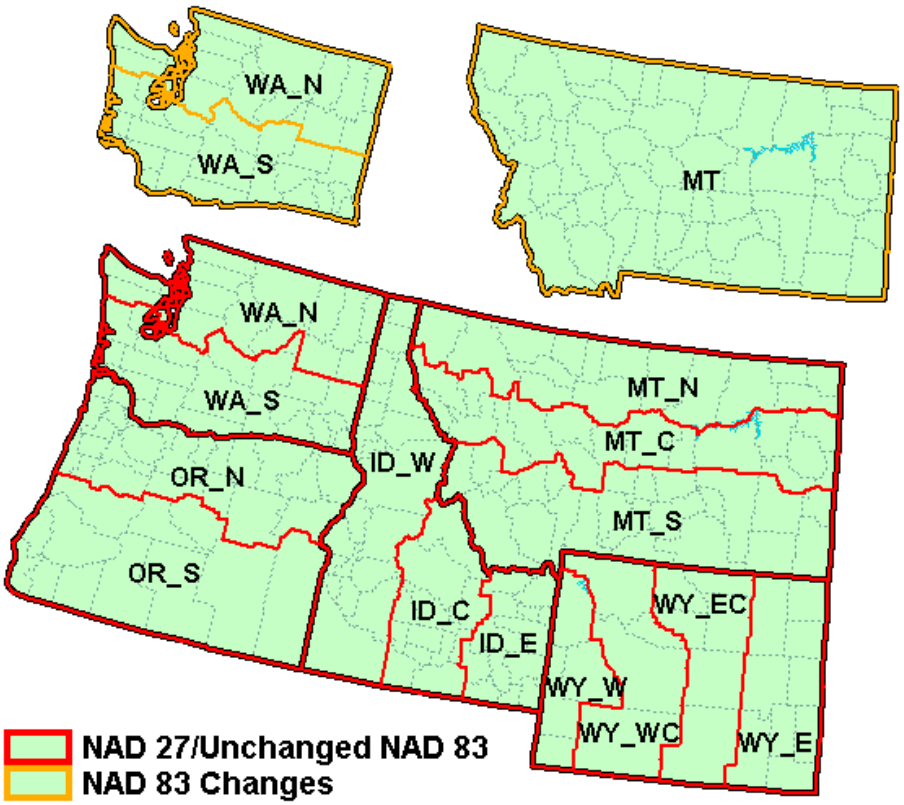
NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Iowa North	IA_N	1401
Iowa South	IA_S	1402
Minnesota North	MN_N	2201
Minnesota Central	MN_C	2202
Minnesota South	MN_S	2203
Nebraska North	NE_N	2601
Nebraska South	NE_S	2602
North Dakota North	ND_N	3301
North Dakota South	ND_S	3302
South Dakota North	SD_N	4001
South Dakota South	SD_S	4002
NAD 83 Zone Changes		
Nebraska	NE	2600

New England State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Connecticut	CT	0600
Maine East	ME_E	1801
Maine West	ME_W	1802
Massachusetts Mainlaine	MA_M	2001
Massachusetts Island	MA_I	2002
New Hampshire	NH	2800
New York East	NY_E	3101
New York Central	NY_C	3102
New York West	NY_W	3103
New York Long Island	NY_LI	3104
Rhode Island	RI	3800
Vermont	VT	4400

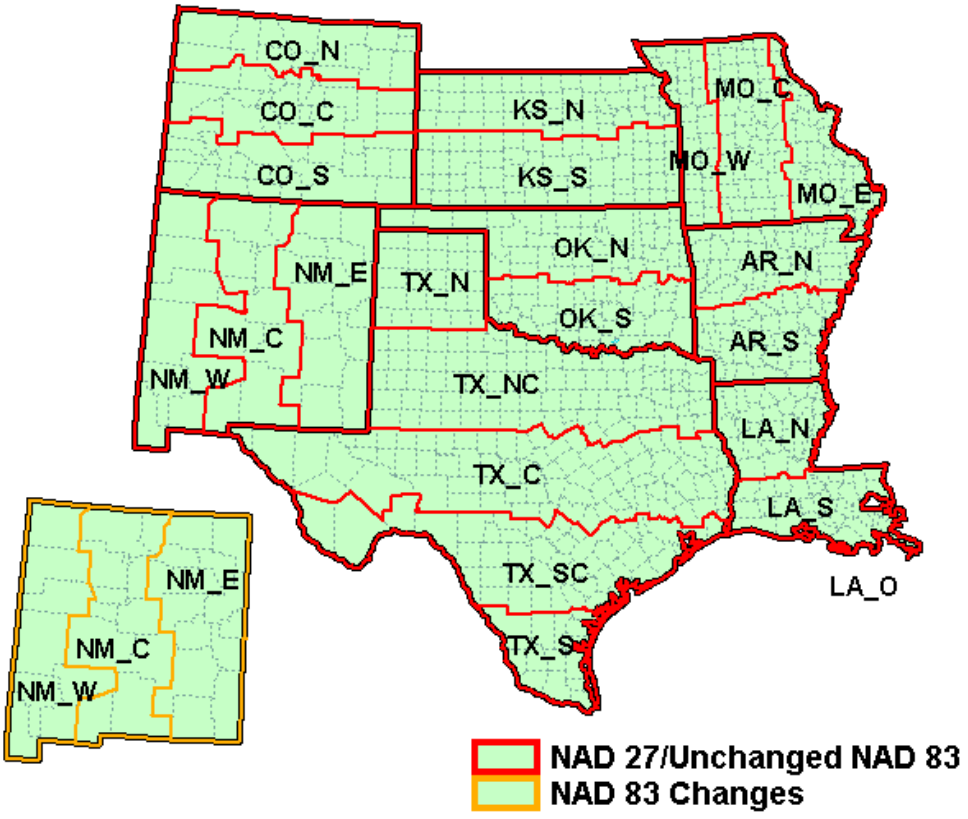
Northwest State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Idaho East	ID_E	1101
Idaho Central	ID_C	1102
Idaho West	ID_W	1103
Montana North	MT_N	2501
Montana Central	MT_C	2502
Montana South	MT_S	2503
Oregon North	OR_N	3601
Oregon South	OR_S	3602
Washington North	WA_N	4601
Washington South	WA_S	4602
Wyoming I	WY_E	4901
Wyoming II	WY_EC	4902
Wyoming III	WY_WC	4903
Wyoming IV	WY_W	4904
NAD 83 Zone Changes		
Montana	MT	2500

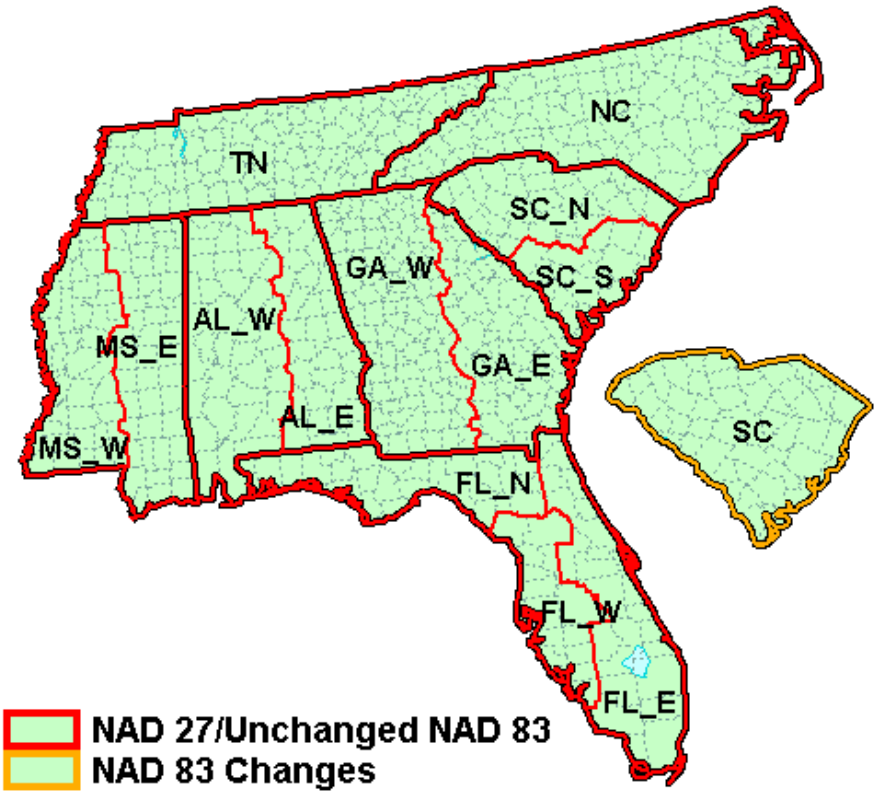
NAD 83 Name Changes		
Wyoming East	WY_E	4901
Wyoming East Central	WY_EC	4902
Wyoming West Central	WY_WC	4903
Wyoming West	WY_W	4904
NAD 83 Boundary Changes		
Washington North	WA_N	4601
Washington South	WA_S	4602

South Central State Plane



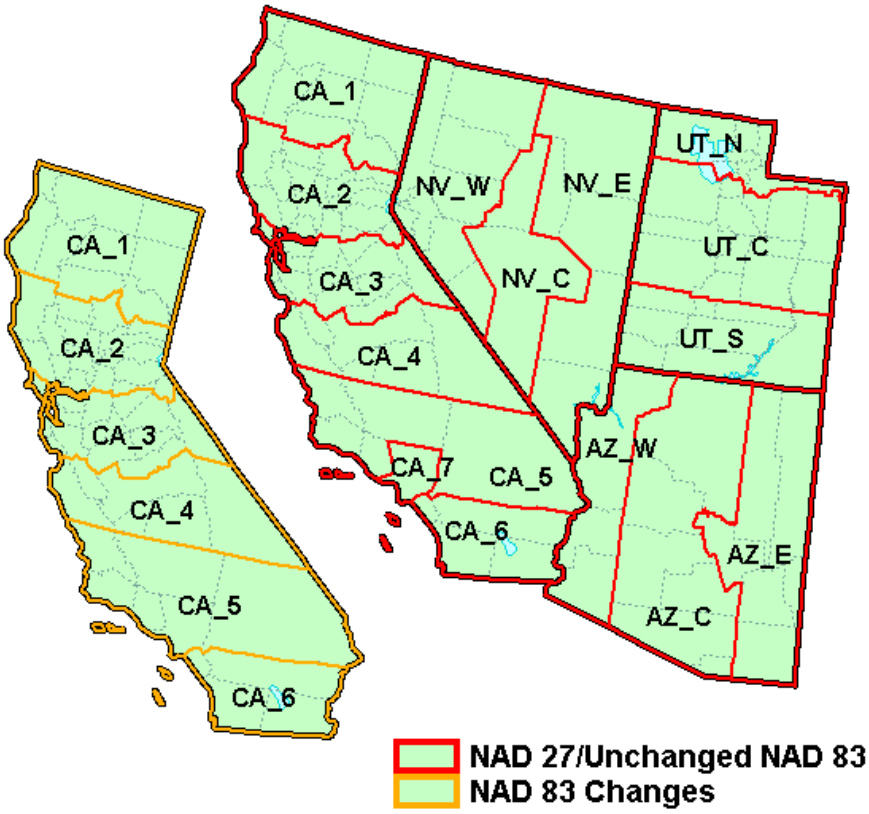
NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Arkansas North	AR_N	0301
Arkansas South	AR_S	0302
Colorado North	CO_N	0501
Colorado Central	CO_C	0502
Colorado South	CO_S	0503
Kansas North	KS_N	1501
Kansas South	KS_S	1502
Louisiana North	LA_N	1701
Louisiana South	LA_S	1702
Louisiana Offshore	LA_O	1703
Missouri East	MO_E	2401
Missouri Central	MO_C	2402
Missouri West	MO_W	2403
New Mexico East	NM_E	3001
New Mexico Central	NM_C	3002
New Mexico West	NM_W	3003
Oklahoma North	OK_N	3501
Oklahoma South	OK_S	3502
Texas North	TX_N	4201
Texas North Central	TX_NC	4202
Texas Central	TX_C	4203
Texas South Central	TX_SC	4204
Texas South	TX_S	4205
NAD 83 Boundary Changes		
New Mexico East	NM_E	3001
New Mexico Central	NM_C	3002
New Mexico West	NM_W	3003

South East State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Alabama East	AL_E	0101
Alabama West	AL_W	0102
Florida East	FL_E	0901
Florida West	FL_W	0902
Florida North	FL_N	0903
Georgia East	GA_E	1001
Georgia West	GA_W	1002
Mississippi East	MS_E	2301
Mississippi West	MS_W	2302
North Carolina	NC	3200
South Carolina North	SC_N	3901
South Carolina South	SC_S	3902
Tennessee	TN	4100
NAD 83 Zone Changes		
South Carolina	SC	3900

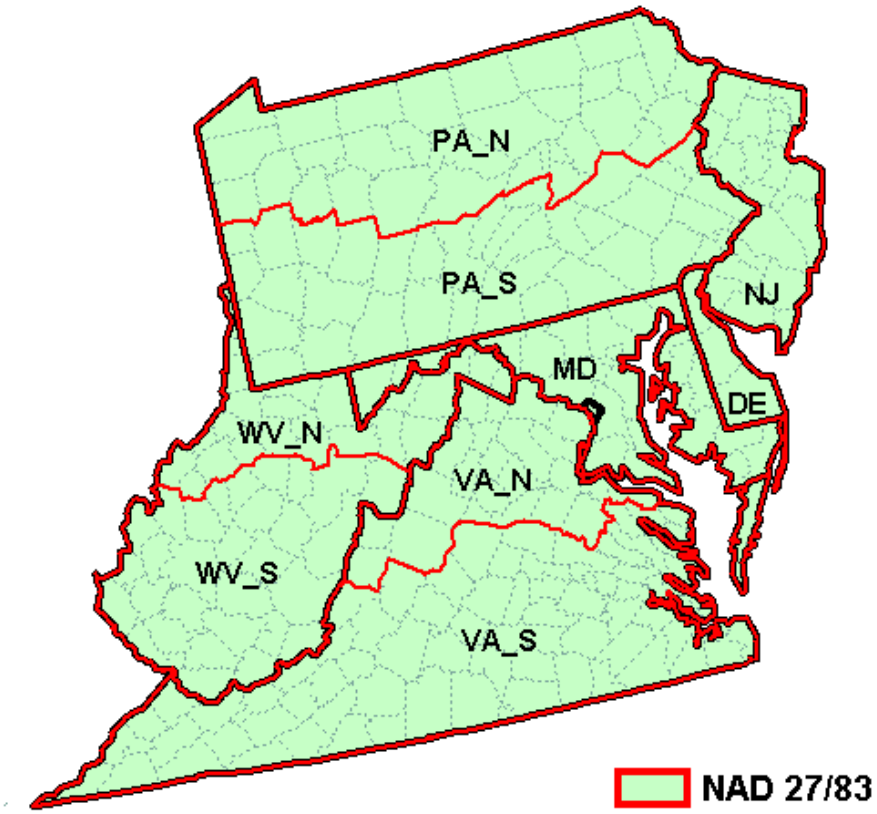
Southwest State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Arizona East	AZ_E	0201
Arizona Central	AZ_C	0202
Arizona West	AZ_W	0203
California 1	CA_1	0401
California 2	CA_2	0402
California 3	CA_3	0403
California 4	CA_4	0404
California 5	CA_5	0405
California 6	CA_6	0406
California 7	CA_7	0407
Nevada East	NV_E	2701
Nevada Central	NV_C	2702
Nevada West	NV_W	2703
Utah North	UT_N	4301
Utah Central	UT_C	4302
Utah South	UT_S	4303

NAD 83 Subtractions		
California 7	Removed	

Virginia Area State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Delaware	DE	0700
District of Columbia / Maryland	MD	1900
New Jersey	NJ	2900
Pennsylvania North	PA_N	3701
Pennsylvania South	PA_S	3702
Virgina North	VA_N	4501
Virgina South	VA_S	4502
West Virginia North	WV_N	4701
West Virginia South	WV_S	4702

2.4. Datasets

Datasets

A dataset is a set of values associated with each node, cell, vertex, or scatter point in an object. A dataset can be steady state (one value per item, one time step) or transient (one value per item, multiple time steps). The values in the dataset can be scalar values or vector values. Certain types of objects in SMS have an associated list of scalar datasets and a list of vector datasets. Each of the following objects in SMS can have both scalar a vector datasets:

- Scattered Datasets
- 2D Meshes
- 2D Cartesian Grids
- Particle Sets
- 1D Grids

Datasets are used for both pre- and post-processing of models. For example, a scalar dataset associated with a 2D mesh can represent starting values of elevations or initial water surface elevations for a surface water modeling problem. Another dataset associated with the same mesh may represent computed water surface values. Datasets can be used to generate contours, vector plots, functional surfaces and animation sequences. The commands for manipulating datasets are located in the Current Model's *Data* menu.

Generating Datasets

Data sets can be generated in a variety of ways such as:

- Output from a surface water model (water level, velocity, concentration, transport, etc.)
- Tabular values in a text file entered by the user or exported from another application such as a GIS
- Created by interpolating from a scatter point set to a grid, or mesh
- Generated by performing mathematical operations on existing datasets with the Data Calculator

One advantage of the dataset approach for managing information is that it facilitates transfer of information between different models with differing resolution. This is accomplished through scatter sets and interpolation. Grids and meshes can be converted to a 2D scatter set. When an object is converted to a scatter set, all scalar datasets associated with the object are copied to the new scatter set. The datasets can then be transferred from the scatter set to other objects of any type using interpolation.

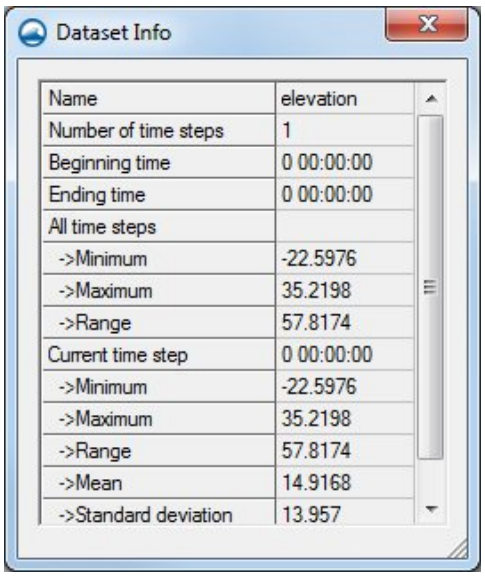
Right-Click Menus

Datasets are displayed and managed in the *Project Explorer*. Right-clicking on a data set invokes the right-click menu which consists of a list of commands that can be performed on the dataset.

Dataset Information

The *Dataset Info* dialog allows the user to examine statistical properties of the dataset.

A dataset consists of a set of values. It is often useful to be able to get general information about the dataset. SMS displays this information in the **Dataset Info** dialog which is invoked by right-clicking on the dataset and selecting the **Info...** command.



Dataset Info dialog

The dialog displays the following information:

- Name – display the name of the dataset
- Number of time steps – displays the number of discrete times represented by the dataset. It will be 1 for a steady state dataset.
- Beginning time – this shows the lowest time for any discrete time represented by the dataset. For steady state datasets, this will be 0.0. It will be displayed in the *information* dialog according to the current time settings.
- Ending time – this shows the largest time for any discrete time represented by the dataset. For steady state datasets, this will be 0.0. It will be displayed in the *information* dialog according to the current time settings.
- All time steps statistics – this displays the minimum, maximum and range of values for all values in the dataset across all times represented by the dataset.
- Current time step statistics – this displays the minimum, maximum, range, mean and standard deviation of values for all values in the current or actively selected time.
- Reference Time – If a dataset is referenced to a universal date/time, it often will be stored relative to a "zero time". SMS allows the user to specify a "zero time" for a project that will be applied to all datasets that do not have their own reference. This field displays the "zero time" for this dataset if it exists.

Active Dataset

Each module in SMS has a set of values designated as the "active dataset." The active data set is an important part of model visualization in SMS. Each time the display is refreshed, the contours and other display features are generated using the active dataset. Left-clicking on a solution or dataset in the Project Explorer makes that item "active". The icons used to identify the different datasets shown in the Project Explorer are as follows:

Dataset Type	Inactive Icon	Active Icon
Elevation		
Scalar		
Vector		

If the active dataset is transient then the time steps are displayed in the Time Step Window.

Solutions

Solutions are output from a numerical model that SMS supports. Solutions are shown in the *Project Explorer* as a folder. If a solution is transient then the time steps are displayed in the Time Step Window. The solution may contain text files such as the *.out and *.prt files produced by a model. These files can viewed by right-clicking on the item and selecting **View File** from the pop up menu, or you can double click on the item.

Folder

The datasets and solutions are organized by folders. The user can create new folders and move datasets, solutions, and folders to other folders anywhere on the *Project Explorer*. Folders can be created by right-clicking on the certain items in the *Project Explorer* and selecting **New Folder** in the menu. A dataset or folder can be deleted simply by selecting the folder and pressing the *DELETE* key or by right-clicking on the item and selecting the **Delete** option in the corresponding pop-up menu.

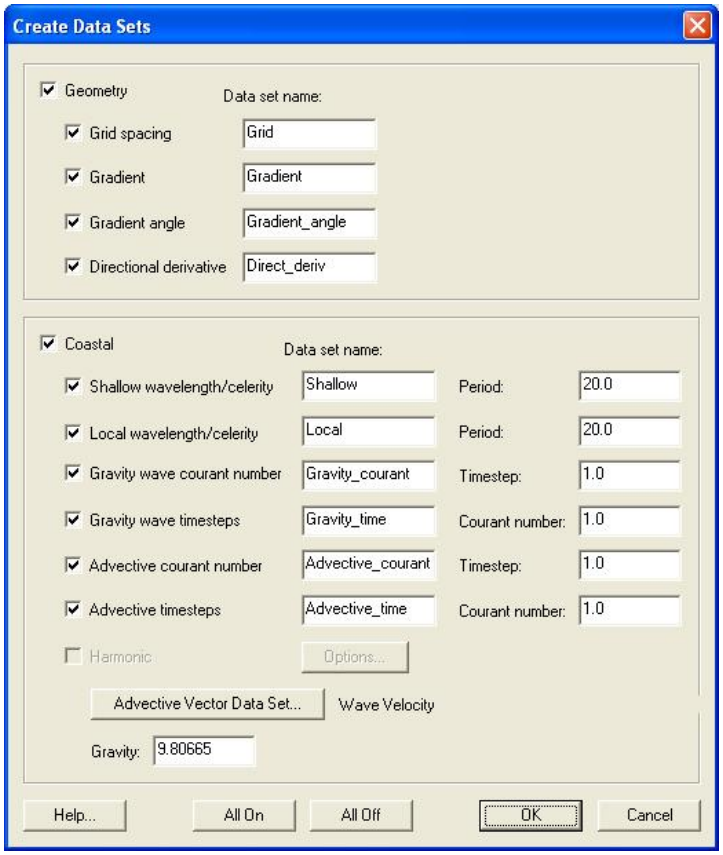
Related Topics

- Layout of the Graphical Interface

Create Datasets

At a glance

- Automatically generate commonly used datasets



Create Datasets

The *Create Datasets* dialog is used to create functions for the entire mesh or active scatter set. The option is in *Data*| **Create Datasets** in both the Scatter and Mesh modules.

Each function that is toggled on will be created. All of the available functions can be turned on by pushing *All On*. All of the functions can be turned off by pushing *All Off*. The Gravity can be set and is used in several of the function calculations. The functions that can be created include:

- **Grid Spacing –** Creates a function that gives the average distance between a node and its neighbors.
- **Gradient –** Creates a function that gives the gradient at each node. The gradient is calculated at each point by averaging the normals of the faces connected to that point. For vertices in a TIN, this includes all the triangles connected to the vertex. For nodes in a mesh, it includes all the elements connected to the node. Currently no adjustment is made to account for the varied area of these faces. The gradient is defined as the run divided by the rise.
- **Gradient Angle –** Creates a function that gives the direction in degrees of the maximum gradient at each point. (See above for the method used to compute the gradient.)
- **Directional Derivative –** Creates a vector function that gives the gradient (run/rise) in the x and y directions. (See above for the method used to compute the gradient.)
- **Shallow Wavelength/Celerity –** Creates two functions that calculate the celerity and wavelength at each node in shallow water. The celerity is calculated as: $Celerity = (Gravity * Nodal Elevation)^{0.5}$. The Wavelength is: $Wavelength = Period * Celerity$.
- **Local Wavelength/Celerity –** Creates two functions that calculate the celerity and wavelength at each node for any depths.
- **Gravity Wave Courant Number –** Creates a function that gives the courant number for each node given the Time Step. The equation is: $CourantNumber = TimeStep*(Gravity*Nodal Elevation)^{0.5}/Nodal Spacing$.
- **Gravity Wave Time Steps –** Creates a function that calculates the gravity wave time step given the Courant Number. The equation is the same as for the Gravity Wave Courant Number, solved for the Courant Number.

The final three functions can be created when the current numeric model is set to ADCIRC.

- **Advective Courant Number –** Creates a function that calculates the courant number given the Time Step and a velocity function. The velocity function can be selected by clicking on the **Advective Functions&ldots;** button. This brings up a *Select Dataset* dialog that lists the vector functions currently in memory. The courant cumber is calculated as: $CourantNumber = NodalVelocityMagnitude * TimeStep/NodalSpacing$. This option is disabled if no vector functions exist.
- **Advective Time Steps –** Creates a function that calculates the time step given the Courant Number and a velocity function. The velocity function can be selected as described above in the description of the Advective Courant Number. The equation is the same as for the Advective Courant Number, solved for the time step. This option is disabled if no vector functions exist.
- **Harmonic –** Creates a scalar harmonic function and/or a vector harmonic function. Pushing the **Options** button brings up the *Harmonic Options* dialog. The name of the function(s) to be created can be set in the *Name* fields. The frequencies to be used in creating the function can be chosen be double-clicking on a frequency name shown in the *Scalar Frequencies* window or be clicking on the name and by pushing **Select**. A frequency can be unselected by double-clicking on the name again or by selecting it and pushing **Unselect**. The time values that will be used in calculating the time steps can be set in the fields at the bottom of the dialog.

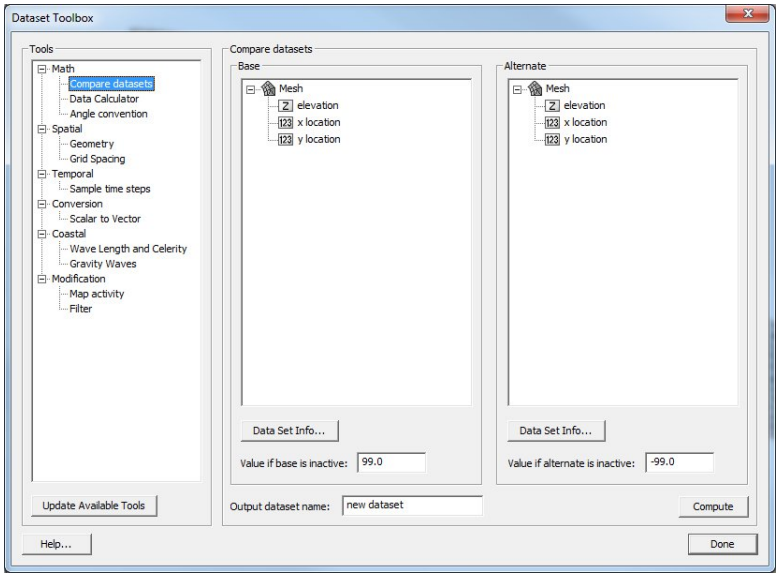
Related Topics

- Mesh Module Data Menu
- Scatter Module Data Menu
- Particle Module Data Menu
- Particle Module Create Datasets

Dataset Toolbox

The *Dataset Toolbox* contains numerous tools for working with datasets. Once the options for the current tool have been set and a name for the resulting dataset has been specified, selecting the **Compute**, **Sample**, etc. button will create the new dataset. The name of the new dataset will appear in the list of datasets.

The *Dataset Toolbox* tools are organized as follows:



Dataset Toolbox dialog

Temporal

Sample Times

Create a new dataset from sampled times of an existing dataset. If **Interpolate times** is selected, linear interpolation will be used to determine the sampled times. If **Interpolate times** is not selected, the value from the nearest existing dataset time step will be used.

Merge Datasets

Starting SMS 11.2, two or more datasets can be merged together. The selected datasets must not have any overlapping time steps.

Derivatives

Create a new dataset of the change from one time step to the next, or the derivative from one time step to the next of an existing dataset. When computing a derivative, the time units must be specified. The the new dataset will output data in between the existing dataset time steps, resulting in one fewer time step than the original dataset.

Math

Compare

Compare two datasets by subtracting the "Alternate" dataset from the "Base" dataset. User specified NULL values are assigned if the base or alternate dataset is inactive.

Data Calculator

For more information, see Data Calculator.

Angle Convention

Create a new dataset with a different angle convention from a scalar dataset containing directions in a given angle convention. With datasets for CMS-WAVE and STWAVE cartesian grids, the angle can be converted to and from a shore normal convention.

Spatial

Geometry

- **Gradient:** Creates a function that gives the gradient at each node. The gradient is calculated as the run divided by the rise.
- **Gradient Angle:** Creates a function that gives the direction in degrees of the maximum gradient at each point.
- **Directional Derivative:** Creates a vector function that gives the gradient (run/rise) in the x and y directions.

Smoothing Datasets

For more information, see Smooth Dataset.

Grid Spacing

Creates a function that gives the average distance between a node and its neighbors.

Conversion

Scalar to Vector

Converts two scalar datasets to a single vector dataset. The specified scalar datasets can be either magnitude and direction or x and y components.

Vector to Scalar

Converts a single vector dataset into two scalar datasets. The resulting scalar datasets can be either magnitude and direction or x and y components.

Coastal

Local Wave Length and Celerity

Creates two functions that calculate the celerity and wavelength at each node for any depths.

- $Celerity = (Gravity * NodalElevation)^{0.5}$.
- $Wavelength = Period * Celerity$.

Gravity Waves (Courant or Time Steps)

Creates a function that gives the courant number for each node given the Time Step, or the gravity wave time step given the Courant Number.

- $CourantNumber = TimeStep*(Gravity*NodalElevation)^{0.5}/NodalSpacing$.
- $TimeStep = CourantNumber*NodalSpacing/(Gravity*NodalElevation)^{0.5}$.

Advective (Courant or Time Steps)

Advective requires a vector function as input and is disabled if no vector functions exist. The courant option creates a function that calculates the courant number given the Time Step and a velocity function. The time step option creates a function that calculates the time step given the Courant Number and a velocity function.

- $CourantNumber = NodalVelocityMagnitude * TimeStep/NodalSpacing$.
- $TimeStep = CourantNumber * NodalSpacing/NodalVelocityMagnitude$.

Modification

Map Activity

This maps the activity array from one dataset to second dataset. This may be used to show only the values of interest on a particular dataset. This operation creates a new dataset.

Filter

This creates a new dataset based on specified criteria. The following options are available for filtering:

- < (less than)
- <= (less than or equal to)
- > (greater than)
- >= (greater than or equal to)
- equal
- not equal
- null
- not null

If the value passes the specified filter, the following can be assigned:

- original (no change)
- specify (a user specified value)
- null (the dataset null value)
- true (1.0)
- false (0.0)
- time – The first time the condition was met. Time can be specified in seconds, minutes, hours or days, and includes fractional values (such as 3.27 hours).

In addition, if the value passes none of the criteria, a default value can be assigned (see available options above).

The filtering is applied in the order specified. This means as soon as the new dataset passes a test, it will not be filtered by subsequent tests.

Related Links

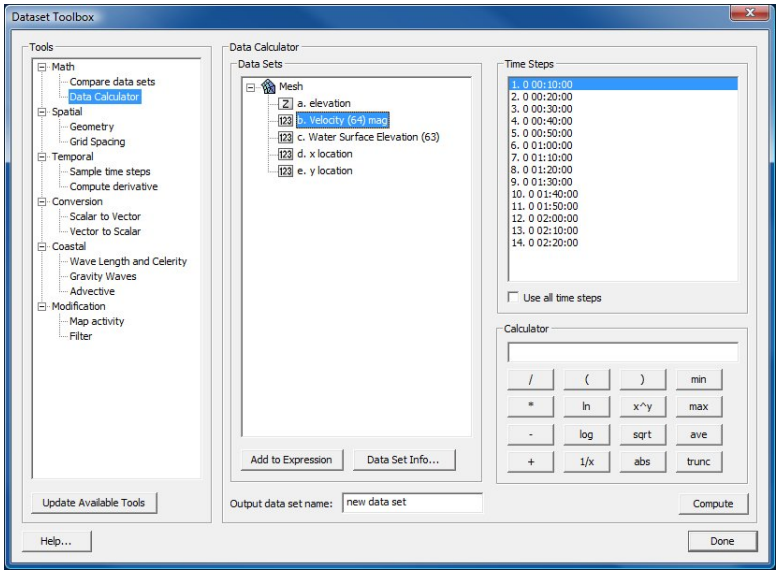
- Data Calculator
- Datasets

Data Calculator

At a glance

- Performs mathematical calculations on scalar datasets
- Calculations can include any number of scalar datasets and user supplied numbers
- Useful for computing derived values such as Froude numbers
- Useful for comparing scalar datasets

The Data Calculator can be used to perform mathematical operations with datasets to create new datasets. The Data Calculator is accessed by selecting the ***Data Calculator*** command from the *Data* or *Edit* menu. The components of the Data Calculator are as follows:



SMS Data Calculator

Expression Field

The most important part of the Data Calculator is the Expression field. This is where the mathematical expression is entered. The expression should be formulated using the same rules that are used in formulating equations in a spreadsheet. Parentheses should be used to clearly indicate the preferred order of evaluation. There is no limit on the length of the expression. The operators in the expression should be limited to the operators shown in the middle of the Data Calculator. The operands in the expression should consist of user-defined constants (e.g., 3.14159), or datasets.

List of Datasets

All of the datasets associated with the active object (TIN, Grid, Mesh, or Scatter Point Set) are listed at the top of the Data Calculator. If a transient dataset is highlighted, the time steps are listed on the right side of the Data Calculator. When a dataset is used in an expression, the name of the dataset should NOT be used. Rather, the letter associated with the dataset should be used. For example, if a dataset is listed as "b. head1", the dataset is referenced in the expression simply as "b"

When a transient dataset is used in an expression, either a single time step or the entire sequence of time steps may be used. For example, the expression "abs(d:100)" creates a single (steady state) dataset representing the absolute value of the dataset at time = 100.0. However, the expression "abs(d:all)" creates a transient dataset representing the absolute value of each of the time steps in the original dataset.

Result Name

When an expression is evaluated, a new dataset is created and the name of the new dataset is designated in the *Result* field.

Operators

The allowable operators are listed in the middle of the dialog. Selecting one of the operator buttons adds the selected operator to the end of the expression. However, the operators can also be typed directly in the expression field. The function of each of the operators is as follows:

Operator	Function
" + "	Add
" - "	Subtract
" * "	Multiply
" / "	Divide
" ("	Left Parenthesis
") "	Right /Parenthesis
" log(x) "	The base 10 logarithm of a dataset
" ln(x) "	The natural logarithm of a dataset
" x^a "	(x) raised to the (a) power. (x) and (a) can be any mixture of constants and datasets
" abs(x) "	The absolute value of a dataset
" sqrt(x) "	The square root of a dataset
" ave(x,y) "	The average of two datasets
" min(x,y) "	The minimum of two datasets
" max(x,y) "	The maximum of two datasets
" trunc(x,a,b) "	Truncates a dataset (x) so that all values are >= a and <= b
" 1/(x) "	The inverse of (x) - Only available in SMS

Operating With Transient Datasets

Each argument in the operators listed in the table above may be:

- A steady state (1 time step) dataset
- A specified time step of a transient dataset (i.e., x:#). In this case the # represents the index of the time step as specified in the time step window.
- A transient time step (i.e., x:all). These operations are only valid if all arguments have matching time step values. In this case, the result will be a new transient dataset with identical time values as the arguments.

The data calculator supports an alternate format for computing attributes of a transient dataset. This alternate format applies to three of the operators. These operators compute a single time step (steady state) dataset representing the spatially varied attribute operating on all the time steps.

Operator	Function
" ave(x:all) "	The average at each location of all time steps in the dataset
" min(x:all) "	The minimum at each location of all time steps in the dataset
" max(x:all) "	The maximum at each location of all time steps in the dataset

Compute Button

Once an expression is formulated and a name for the resulting dataset has been specified, the expression can be evaluated by selecting the **Compute** button. At this point, the dataset is created and the name of the new dataset should appear in the list of datasets.

Related Links

- Datasets (GMS)
- Datasets (SMS)
- Datasets (WMS)
- Dataset Toolbox (SMS)

Size Dataset

The **Size Dataset** command is used to determine how far apart to space the mesh nodes when:

- Generating a mesh using the Scalar Paving Density mesh types
- Relaxing a mesh

External Links

- Howlett, John David (2005). *Size Function Based Mesh Relaxation.* Thesis, Brigham Young University. [1]

Related Topics

- Smooth Dataset

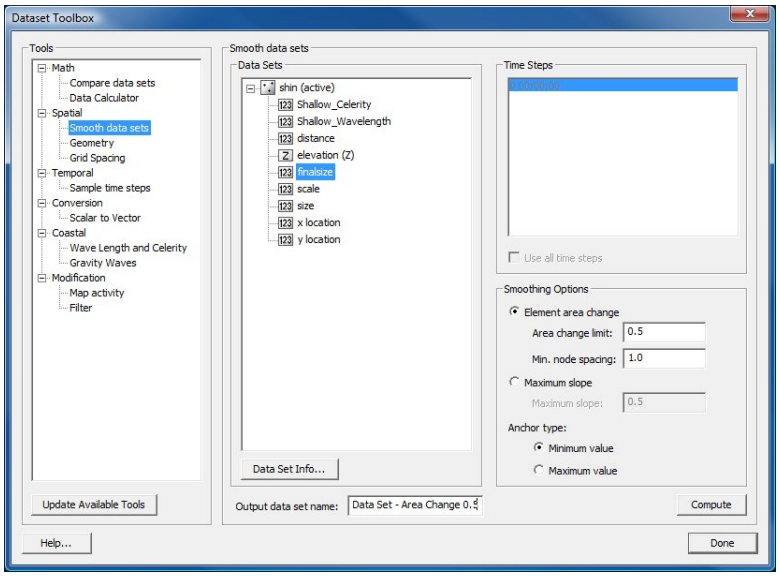
References

[1] http://contentdm.lib.byu.edu/utils/getfile/collection/ETD/id/287/filename/etd761.pdf

Smooth Dataset

The *Smooth Dataset* dialog is used to condition scattered data scalar values before those values are used in an interpolation process. This includes two general applications, smoothing a size dataset to prevent the dataset values from changing too quickly, and smoothing depth/elevation values to prevent extreme slopes.

The *Smooth Dataset* dialog is accessed via the Dataset Toolbox by selecting *Data* | **Dataset Toolbox** in the Scatter module.



Smooth Datasets dialog

Smoothing Size Datasets

One measure of mesh quality is element area change. If the dataset values change too quickly in a size dataset, the element area change of adjacent elements may be too great, resulting in poor mesh quality.

Smoothing options

- **Element area change limit –** The selected dataset values will be modified to honor the specified element area change limit. This value defines the maximum ratio between adjacent points based on the distance between points.
  - **Minimum value anchor type –** Dataset values are decreased. Results in a more refined (more nodes/elements) mesh when used as a size dataset.
  - **Maximum value anchor type –** Dataset values are increased. Results in a less refined (fewer nodes/elements) mesh when used as a size dataset.
- **Minimum node spacing –** The minimum value allowed in the smoothed dataset.

Tips

After smoothing a size dataset, use the data calculator to subtract the smoothed sized dataset from the original dataset and create a "change" dataset. Contour the "change" dataset to easily determine what and where changes were made by the smoothing algorithm.

Smoothing Elevation/Depth Datasets

This option allows the user to specify a maximum slope. The process creates a new dataset which honors the maximum specified slope.

- Minimum value anchor type – The smoothing operation anchors the minimum dataset value (such as the lowest elevation or smallest depth) and adjusts the adjacent values to ensure the slope is less than or equal to the specified slope.

- Maximum value anchor type – The smoothing operation anchors the maximum dataset value (such as the highest elevation or largest depth) and adjusts the adjacent values to ensure the slope is less than or equal to the specified slope.

The operation includes all scatter points if none are selected. Alternatively, the user may select a group of scatter points to be operated on. Points not selected will not have their scalar value modified. This means the only way to modify a point is if it has an adjacent point that is also selected and the slope between these two selected vertices is steeper than the maximum specified slope.

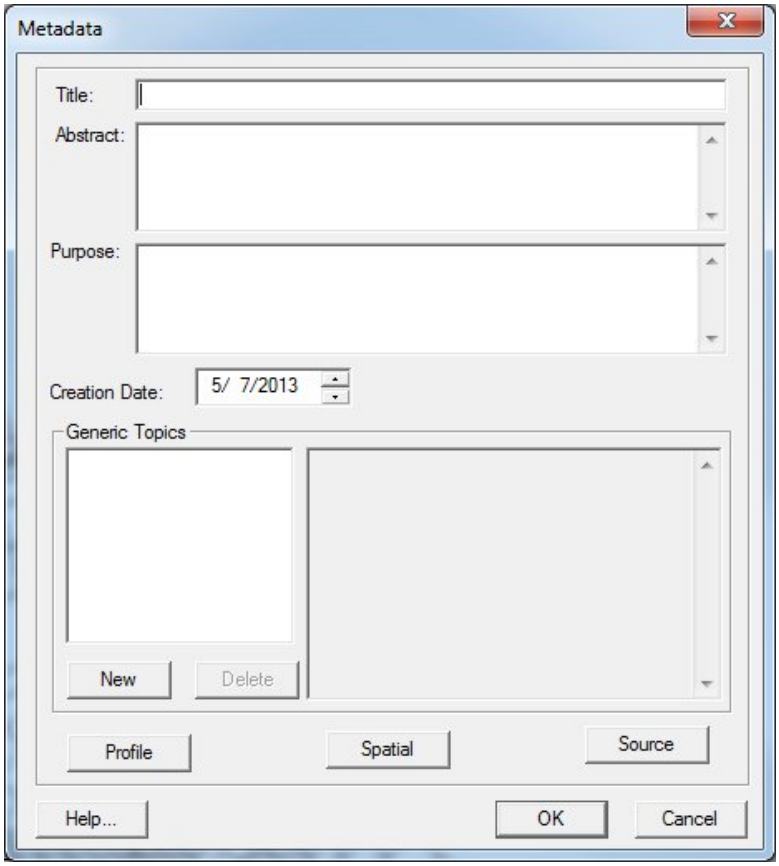
Related Topics

- Scatter Data Menu
- Size Dataset
- Relax Elements
- Dataset Toolbox

Metadata

Metadata, or data about data, can be crucial in the modeling process. In many situations, metadata is saved in a separate file (called a metadata file). A common problem is that metadata and the data it describes are often separated. Metadata is of little value without the data files it relates too. At the same time, metadata makes the data more usable and therefore, more valuable.

In the SMS, metadata can be cataloged inside the project file. Metadata can be associated with the project as a whole, a single geometric object such as a survey or finite element domain, or individual components down to the dataset level.



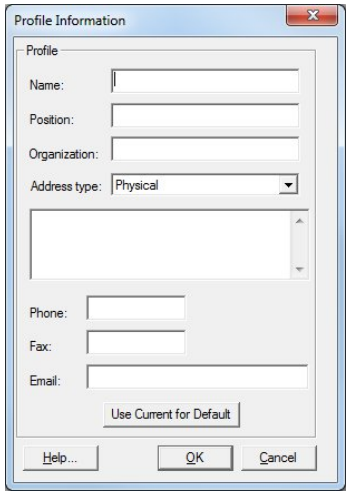
Metadata dialog

Project/Geometric Object Metadata

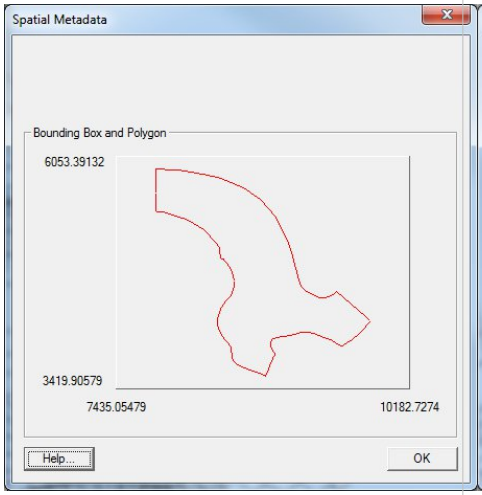
The project metadata can be accessed through the *Edit*|**Metadata...** command. Object metadata can be accessed by right-clicking on the object and choosing the **Metadata...** command. Either method invokes the *Metadata* dialog which includes edit fields for the following metadata:

- **Title of the project or object**
- **Abstract** – a brief description

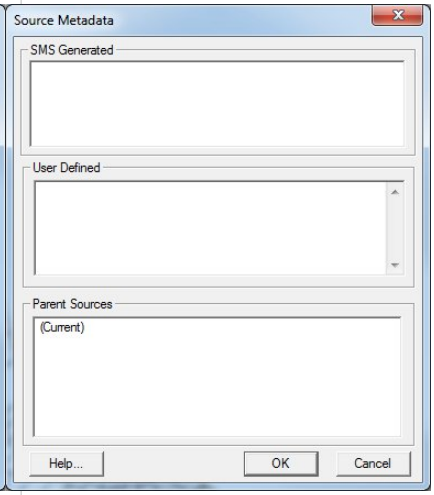
- **Purpose –** this can change over time, but if it is recorded with the project when generated, it provides a valuable backdrop when applying the project for other purposes at later times.
- **Creation Date –** This is filled in automatically to the current date when the project is created. It can be edited if an existing project is simply being organized.
- **General Topics –** This section includes a list of topics and description or blurb for each topic. As the project develops or is modified, notes of these developments, consisting of dates, individuals involved, purposes, etc, can be annotated to the project.
- **Profile –** this button invokes the *Profile Information* dialog which documents who created this project and provides information about this individual. The profile can be set up and associated with an installation of SMS and then this information is automatically added to all projects created with this installation. In order to associate a profile as the default, simply click the **Use Current for Default** button in the *Profile Information* dialog.
- **Spatial –** this button invokes the *Spatial Metadata* dialog displaying the projection used by the project and the spatial limits of the project. This information is automatically filled in.
- **Source –** this button is only available for objects (not the project as a whole). It brings up a the *Source Metadata* dialog. Whenever SMS generates a new object, such as a mesh from a conceptual module, or reads in a new object from a file, the source is recorded. In the former case, the coverage and scatter set used will be recorded. In the latter, the filename. This dialog allows the modeler to record additional notes about this object.



Profile Information dialog



Spatial Metadata dialog



Source Metadata dialog

Dataset Metadata

Dataset metadata is accessed by right-clicking on the dataset. This invokes the *Dataset Metadata* dialog which includes a text string describing the dataset.

Dataset Metadata dialog

2.5. Display Options

Display Options

Display Options in SMS refers to the control of what entities are displayed, and how (color and style) they are displayed. Each entity in each module has its own display options. The display options for the active module are shown when the *Display Options* dialog opens.

The *Display Options* dialog is opened by any of the following methods:

- Right-clicking on the Module type folder in the Project Explorer and selecting the **Display Options** command from the right-click menu
- Using the Display | **Display Options** menu command
- Clicking the *display options* Macro
- Using the keyboard shortcut *CTRL+D*

Display Option Pages

SMS supports a display option page for general display options and for each type of data (i.e. 2D mesh, scatter sets, map data, ...) managed in a simulation. The *display option* dialog includes the option to set all options on all pages if desired. The toggle at the lower left of the dialog hides the option pages for data types that are not included in the current project. This toggle is selected by default and reduces the amount of information a user sees. The following display option pages exist:

- 1D Grid – This page controls the display options related to the coastal morphology model GenCade.
- 2D Mesh – This page controls the display options related to unstructured 2D meshes and the models (such as ADH and the generic model interface) that use them.
- Cartesian Grid – This page controls the options related to Cartesian grids and the models (such as TUFLOW, CMS-Flow and BOUSS2D) that use them.
- Curvilinear Grid – This page controls the options related to the curvilinear or boundary fitted grids and the models (such as LTFATE) that use them.
- General – This page controls the general display options.
- GIS – This page controls the display options related to GIS data using the internal GIS support in SMS. If ArcGIS is enabled inside of SMS, the ArcObjects display options control the display of these entities.
- Map – This page controls the display options related to the map module (coverages) in SMS.
- Mesh – This page controls the display options related to the general unstructured mesh objects.
- Particle – This page controls the display options related to the PTM Lagrangian particle tracker model.
- Raster – This page controls the display options related to raster (or DEM) type objects.
- Rivhyd – This page controls the display options related to one dimensional river models.
- Scatter – This page controls the display options related to scattered datasets (also referred to as Triangulated Irregular Networks or TINs).

Tabs

For each page, additional tabs may appear to allow the user to specify the display settings for vectors or contours in that type of data. The General Display Options also includes separate tabs for Lighting Options and viewing control.

Functional Surfaces

For some types of data, it could be useful to display a surface of the datasets associated with the geometric data. For example, a computed water surface can be displayed as a surface to intuitively illustrate how a flooding scenario looks in three dimensions. Other datasets, which don't have such a direct connection to elevation can also be viewed as a functional surface to give insight to the situation. The options to set the functional surface for a specific geometric data type will appear in the page for that data type.

Entity Display Options

For each entity type, the dialog includes a toggle, and where appropriate, a button. If the toggle is selected, SMS will display the entity type. The buttons are of various types, as described below:

Points

If the display of an entity is focused around a single location, like a mesh node, the button displays a circle, drawn in the color that will be used to display that entity. To change the circle size, or color, the user can click on the button, and the point attributes dialog will appear. The color can also be changed by clicking on the combo box arrow next to the button. It is recommended that if there are many of points, and their position can be inferred from other displayed entities (such as the position of mesh nodes by the edges of the elements) that the symbols be turned off to increase efficiency.

Symbols

If the display of an entity is focused around a single location, like a scatter point, the button displays a symbol, drawn in the color that will be used to display that entity. To change the symbol, symbol size, or color, the user can click on the button, and the *Symbol Attributes* dialog will appear. The color can also be changed by clicking on the combo box arrow next to the button. It is recommended that if there are many of these entities, and their position can be inferred from other displayed entities (such as the position of scatter points by the edges of the scatter triangles) that the symbols be turned off to increase efficiency.

Line

If the entity to be displayed encloses a region, such as a triangle or element, the button displays a sample line drawn in the color and width that will be used to display the line around the edge of that entity. To change the color and/or width, the user can click on the button, and the *Line Attributes* dialog will appear. This allows the user to select a line style (dashed or solid), a width (in pixels) and a color. The color can also be changed by clicking on the combo box arrow next to the button.

Font

If the entity to be displayed is a text string, such as the node id, the button displays a sample string ("AaBb") drawn in the color and font that will be used to display the string. To change the font, the user can click on the button, and the *Font* dialog will appear. The color can be changed by clicking on the combo box arrow next to the button.

Color

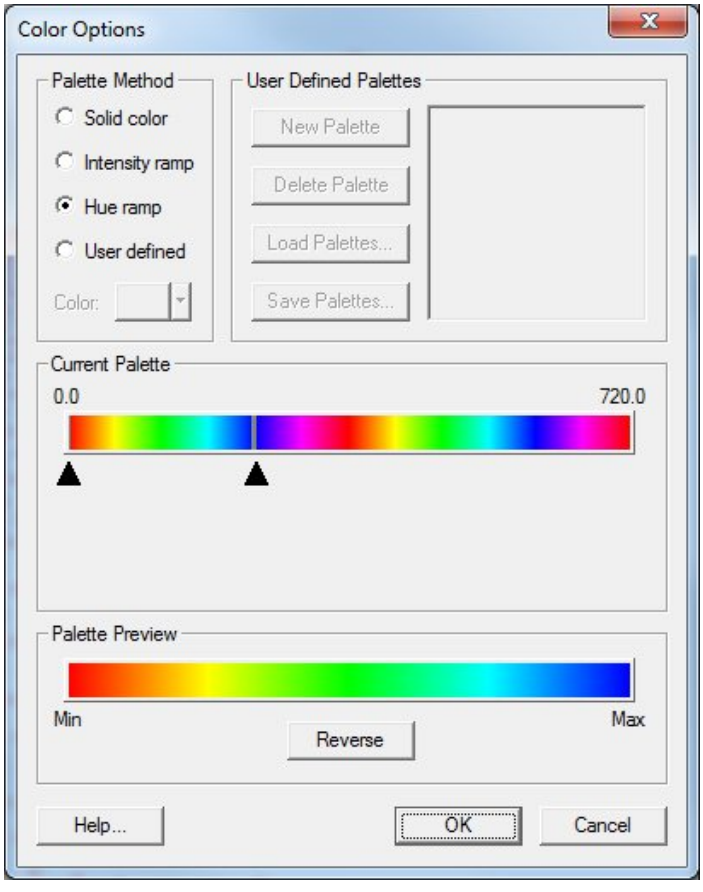
If the entity has only a color associated with it, the button displays a square, drawn in the color that will be used to display that entity. To change the color, the user can click on the button, and the *Color* dialog will appear. The color can also be changed by clicking on the combo box arrow next to the button.

Options

Other specific display attributes can be accessed through the **Options** buttons. The options available will vary based on the current model and includes attributes such as boundary conditions assigned to nodes or nodestrings.

Color Options

The *Color Options* dialog lets the user determine how the contours and vectors will be colored. The *Color Options* dialog is opened from the *Data* menu, or from the **Color Ramp...** button on the *Contours* and *Vectors* tabs of the *Display Options* dialog.



Color Options dialog

The default color method is the *Solid* color option. This method uses a single color for all contours. As an alternative, the user can define a ramp of colors. These colors are distributed across the range of contour values in a continuous fashion, giving each contour its own color. If specific contour values are specified, the user controls whether the colors are distributed by index or by value in the upper right portion of the dialog. The following types of color ramps are supported by SMS:

- Solid color – A single color is used for all values.
- Intensity ramp – The color ramp is defined as a continuous variation of the intensity of the default solid color. This is the same color used for the *Solid color* option.
- Hue ramp – The ramp is a continuous variation of hues using the hue-saturation-value color model.
- User Defined Palettes – User defined color ramp.

If using an intensity ramp or hue ramp, the ramp can be edited to include only a portion of the entire ramp, or converted to a *User Defined Palettes* for further editing. The user modifies the portion of the ramp to be used by

setting the minimum and maximum values for hue or intensity with the scroll bars in the *Color Options* dialog. These controls specify where the minimum value will be mapped into the ramp and where the maximum value will be mapped. The **Reverse** button changes the direction of the color gradation in the color ramp.

User Defined Palettes

The user may define and edit a color palette for use with contours. In the *Display Options* dialog, on the *Contours tab*, pressing the **Color Ramp** button, opens the *Color Options* dialog. Inside the *Color Options* dialog, selecting the *User defined* option enables the following options:

User Defined Palettes Frame

- **New Palette –** Create a new color palette. This opens the *New Palette* dialog, which is used to define a color ramp palette. The user selects a preset palette and the initial number of colors in the palette. The palette can be fine tuned once it is created in the *Color Options* dialog.
- **Delete Palette –** Delete the selected palette.
- **Load Palettes –** Load palettes from an SMS defined palette file.
- **Save Palettes –** Save all of the user created palettes to a file using the format shown above.

Current Palette Frame

The color pallet selected in the *User Defined Palettes* frame is displayed. The user can select, edit, and drag colors in the *Color Palette* using the following tools:

- **Create a breakpoint** tool
  - Mouse left-click – Creates new breakpoints
  - Select an individual *breakpoint* tool
    - Mouse left-click and drag – Changes the value associated with a breakpoint
    - Mouse left double-click – Opens the *Color* dialog to change the color associated with a breakpoint
    - Mouse left-click, then *DELETE* key – Delete the selected breakpoint

      Value edit field – Change the value of a selected color. Changing a value will move the color inside the color palette window.

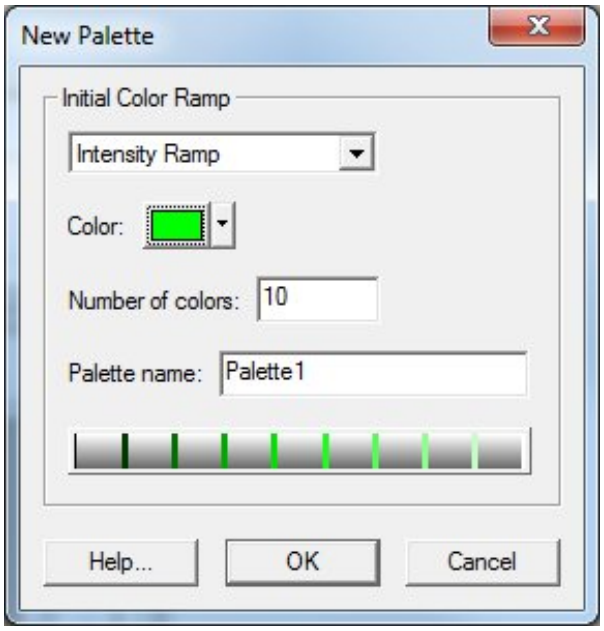
      Edit Table – This button opens the *Color Table* dialog. Values and colors associated with each breakline can be viewed and edited. This dialog is useful for creating a palette with a logarithmic scale. It may be difficult to select colors very close to one another at the lower end of a log scale using the mouse left-click button, but the values can easily be specified in this dialog.
    - Display Value As – Show the value of each color as:
      - **Percentages (0.0-1.0) –** A percentage across the palette, with 0.0 being to the left of the palette and 1.0 at the right edge of the palette.
      - **Numerical Values –** The actual value of each color. Each color will represent a value such as elevation.

New Palette

The *New Palette* dialog is used to create a new user defined palette. The following palette options can be set:

- Initial Color Ramp Type:
  - Solid Color
  - Intensity Ramp
  - Hue Ramp
  - Elevation
  - Ocean
  - Magnitude Difference
  - Color – Only available for *Solid Color* and *Intensity Ramp*
  - Number of Colors – Only available for *Solid Color*, *Intensity Ramp*, and *Hue Ramp*
  - Palette name

    Once the general options have been set in the *New Palette* dialog, the palette can be fine tuned using the tools in the *Color Options* dialog.



New Palette dialog

Related Topics

- Contour Options
- Contour Labels
- Display Options
- Vector Visualization

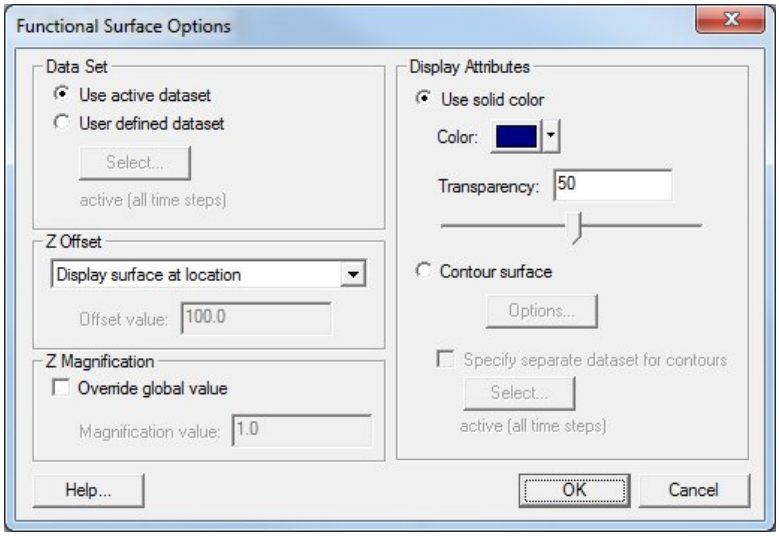
Functional Surfaces

At a glance

- Surface with elevation based upon scalar dataset values
- Very useful for wave models and models with large change in water surface elevation
- Elevations can be exaggerated to better visualize dataset variations
- Surfaces can have a solid color or use color filled contours
- Transparency can be used to allow see through surfaces

A functional surface is exactly that. It is a surface representing one of the functional datasets associated with a mesh, grid or TIN. The most intuitive example of a functional surface is the display of the water surface over a model's bathymetry. In this case, the surface represents an actual physical surface, but the functional surface could just as easily represent the velocity magnitude, or concentration, or any other scalar quantity.

To create/display functional surfaces, the user must enable them in the display options of the appropriate module, and specify their attributes which include:



Functional Surfaces Options dialog

- **Dataset –** the user selects which dataset is to be used to form the functional surface.
  - Use active dataset
  - User defined dataset
  - **Z Offset –** SMS displays functional surfaces at a simulated z-value. This may be the actual surface value (such as is the case with water surfaces elevations), but more often the value will not have a physical meaning, and may intersect the bathymetry or not even be in the same area. For this reason, SMS offers options for placing the functional surface at its real values, relative to the bathymetry, or at a user specified offset.
  - **Z Magnification –** Functional data may not vary significantly when compared to the horizontal extents of the model. For this reason, the interface allows magnification (scaling) of the functional surface. By default, the surface is scaled based on the global z-magnification specified in the general display options. This may be overridden.
    - Override global value
    - Magnification value
    - **Display Attributes –** the user controls the color of the functional surface. It may be a constant color or colored based on the contour colors specified. The colors may be associated with the value of the functional surface, or another dataset. The surface may also be partially transparent.
      - Use solid color
      - Transparency
      - Contour surface
      - Specify separate dataset for contours

Related Topics

- Cartesian Grid Display Options
- Mesh Display Options

Lighting Options

This dialog allows the user to control the shading of faces in the SMS display. By default, all objects are displayed in the color specified by their attributes. However, objects such as elements, cells and triangles which cover an area, can be more intuitively understood if they are shaded as a three dimensional entity. The shading options includes two toggles, one slide bar and a light position window.

The lighting options are accessed by clicking on the *Lighting Options* item or tab in the *Display Options* dialog. The default options vary between applications, and the options may be changed, saved, and restored within the project.

Toggles

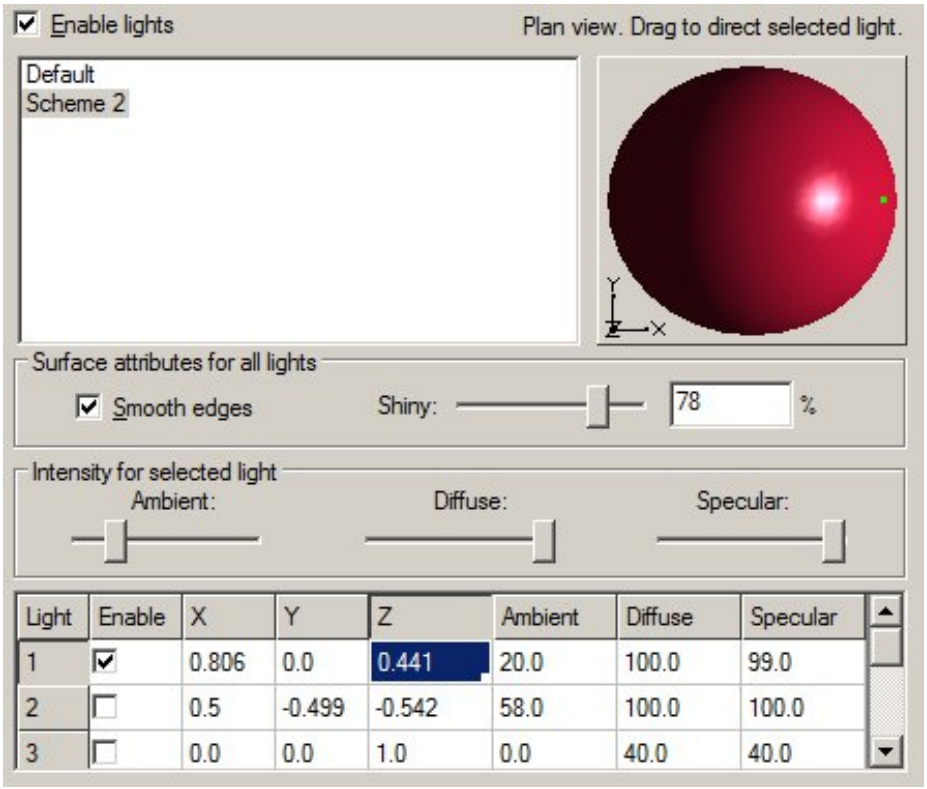
The first toggle allows the user to turn on the use of a light source. When this toggle is selected, the second toggle becomes available. The second toggle tells SMS to smooth corners between adjacent faces. This allows the faceted surface to appear as a smooth surface.

Slider

The slide bar allows the user to specify the amount of ambient light. Ambient light is the minimum intensity (brightness) to be displayed. A recommended value is between 0.2 and 0.4.

Light Position

The right side of the dialog allows the user to set the light direction and gives a preview of that direction displayed on a sphere.



The following table describes the lighting display options.

Display Option	Description
Enable lights	This check box controls whether light sources are used in the lighting process for generating lighted images. These light sources control the intensity of the colors on the lighted image and highlight the relief or geometrical variation in the surface of the objects being lighted.
Lighting list box	This list contains preset lighting schemes and highlights the scheme currently displayed.
Renaming a scheme	Double click on a scheme to begin editing its name.
Deleting a scheme	Right click on a scheme and select Delete. The final scheme cannot be deleted.
Creating a scheme	Right click on a scheme and select duplicate.
Plan view preview	This preview shows the current light scheme on a sphere in plan view, i.e., looking along the z-axis. Click or drag within the preview to direct both the diffuse and the specular components of the light currently selected in the light table. The selected light direction is shown by a dot on the sphere. A direction from in front of the sphere is shown by a green dot, and from in back by a red dot.
Smooth edges	Check this box to smooth all diffuse and specular lights of this scheme so that the surface does not appear faceted.
Shiny	Increase this value to sharpen all specular highlights of this scheme. At 100% this value turns off the specular highlight since it assumes that all specular lights are points whose reflection shrinks to a imperceptible point at maximum shinyness. At 0% this value assumes that the full intensity of the light is reflected in all directions (decrease the specular values proportionally to get a realistic effect of less and less light reflecting to the eye from each surface).
Ambient slider	Shows the Ambient value of the light currently selected in the table, and can change the value. The ambient value is light from all directions which lights each and all surfaces uniformly leaving no surface unlighted. It is most useful on surfaces facing away from directional light such as diffuse and specular light.
Diffuse slider	Shows the Diffuse value of the light currently selected in the table, and can change the value. The diffuse value is for a point light which brightens surfaces in all directions the more they face the that light, and which leaves surfaces in darkness that face away from the light.
Specular slider	Shows the Specular value of the light currently selected in the table, and can change the value. The specular value is a point light which brightens surfaces if they reflect like a mirror from the direction of the light to the direction of the viewer, and which leaves surfaces in darkness that do not have this angle of reflection.
Light table	Displays the enable, xyz position, Ambient, Diffuse, and Specular values for each of 8 lights in the current scheme, and highlights the currently selected light. Any of these values may be modified by clicking them and editing their value.
Enable column	Check these boxes to turn on each light.
X, Y, and Z columns	Edit these values or click/drag in the plan view preview sphere to change the direction of the light. These values are will be normalized to a unit direction vector.
Ambient, Diffuse, and Specular columns	Edit these values or drag their corresponding slider.

Raster Options

Importing Rasters

Import a raster file by selecting **Open** in the *File* menu. Select the proper raster file as shown in the following table. Select **Open**. At the popup *Load it as...*, select DEM.

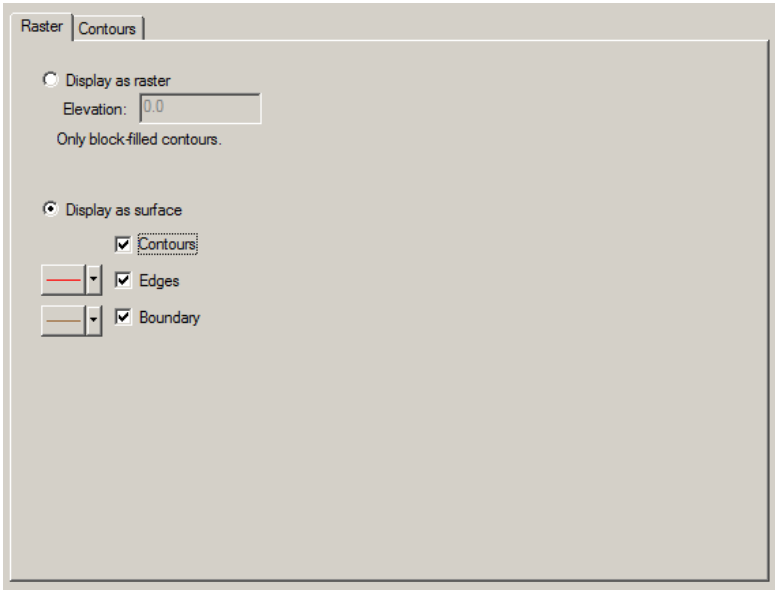
Format	File to Open	Source	Importance	Level of Support
ArcInfo Binary Grid	w001001.adf	ESRI	1	Supported in GMS and in SMS
ArcInfo Ascii Grid	*.asc	ESRI	2	Supported in GMS and in SMS
USGS DEM Grid Float	*.flt	[1] in GMS and in SMS		
USGS NED Grid Float	*.flt	[1] in GMS and in SMS		
Canadian DEM	*.dem	[2] in GMS and supported in SMS		
DTED	*.dt0	ERDC	3	Supported in GMS and supported in SMS
Aster DEM	*.tif	[3] in GMS and supported in SMS as images		
SDTS	*.ddf	[4] in GMS and supported in SMS		

How to export TINs in adf format from ArcGIS in a format that XMS will read

1. Load the TIN into ArcMap
2. Expand the “3D Analyst Tools | Conversion Tools | From TIN” toolset in ArcToolbox
3. Double click the **TIN to Raster** tool in ArcToolbox (specify your current TIN file in the *Input TIN* field). Make a note of the path in the *Output Raster* field.
4. Expand the “Conversion Tools | From Raster” toolset in ArcToolbox
5. Double click the *Raster to ASCII* tool in ArcToolbox (specify the raster file that you created in the previous step as the input raster, and make a note of the path for the output file)
6. Open Windows Explorer (My Computer) and browse to the location of the ASCII *.txt file output in step 5
7. Make a copy of the *.txt file created in step 5
8. Change the extension of the *.txt file to *.dem
9. Open the *.dem file in WMS

Displaying Rasters

The raster display options are accessed by clicking on the *Raster Options* item or tab in the *Display Options* dialog. The default options vary between applications, and the options may be changed, saved, and restored within the project.



The *Raster* tab of the *Display Options* dialog.

The following table describes the raster display options.

Display Option	Description
Image Display	Select the *Display as raster* radio button to display the raster as a flat image rather than as a surface with elevation changes. Contour options are applied to form the image with block color fill.
Image Elevation	The raster image is drawn at an elevation of 0.0 by default. Change the *Elevation:* value to draw it at a different elevation.
Surface Display	Select the *Display as surface* radio button to display the raster as a height varying surface rather than as a flat image. Enable either Contours, Edges, or Boundary to see that type of surface or nothing will be shown.
Surface Contour	Select the *Contours* check box to apply contour options to the surface with contour lines and/or smooth color fill.
Surface Edges	Select the *Edges* check box to display the polygonal edges between height samples in the surface. The control to the left sets line color and either enables line dashes or species line width for the edges. This is typically the slowest surface to render.
Surface Boundary	Select the *Boundary* check box to display only those polygonal edges between height samples on the perimeter of the surface. The control to the left sets line color and either enables line dashes or species line width for the boundary. This is typically the fastest surface to render.

Back to XMS

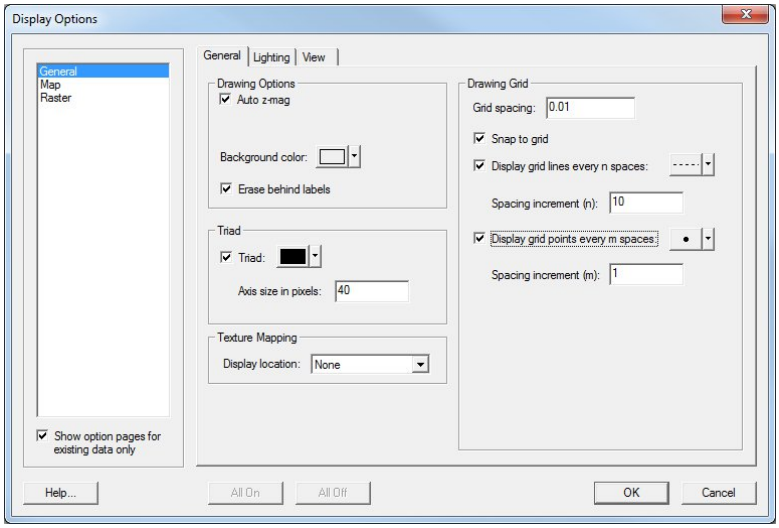
References

[1] http://seamless.usgs.gov||2||Supported
[2] http://www.geobase.ca||3||Supported
[3] http://asterweb.jpl.nasa.gov/gdem-wist.asp||4||Supported
[4] http://data.geocomm.com/dem/demdownload.html||5||Supported

General Display Options

The *General Display* options control display of general graphical control. It includes three tabs including:

General Tab



Display Options dialog showing the *General* tab

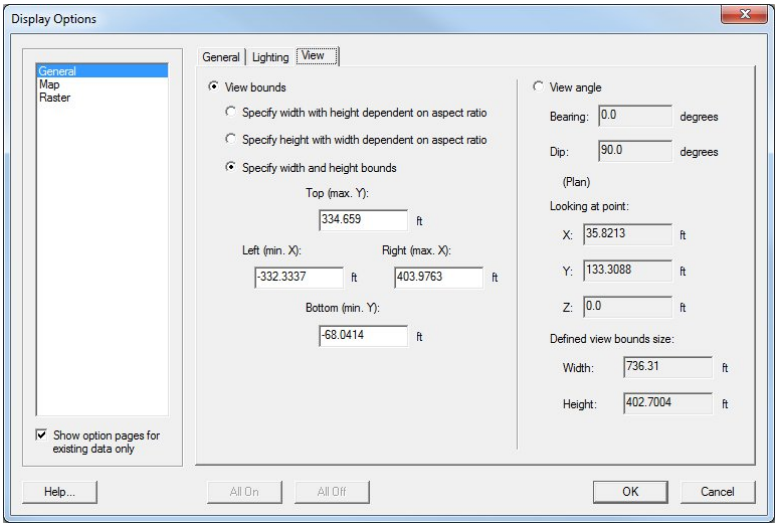
- **Drawing Options:**
  - Z magnification
  - background color
  - erase behind labels
  - **Triad:**
    - SMS can display a coordinate triad at the lower left of the screen to display the orientation of the data in the display window. The user can control the size and color of this triad.
- **Texture mapping:** Currently SMS supports images displayed in the background and texture maps draped over TINS, grids, and meshes.
- **Drawing Grid:** SMS can display a grid (in plan view) behind all data on the graphics window.
  - *Grid Spacing:* The user can enter the increment between grid points. Remember that the grid can be used for both snapping and display, and not all grid lines need to be displayed.
  - *Snap to Grid:* If this toggle is on, newly created points, nodes and vertices are moved to the nearest point on the grid.
  - *Display grid lines every n spaces:* This option allows the user to specify how many grid lines to between displayed grid lines. The line style is also selected.
  - *Display grid points every m spaces:* This option allows the user to enable the display of a point at selected intervals along with the symbol attributes for the points.

Lighting Tab

The *Lighting* tab allows the user to access to the Lighting Options in SMS.

View Tab

The *view* tab in the general display options allows the user to see/edit the current view parameters. This includes the specification of the type of view (plan or 3D) and the range of the data that is displayed on the screen.



Display Options dialog showing the *View* tab

Related Topics

- Display Options

Z Magnification

Z Magnification options can be found in the *General Tab* under *General Display Options*.

Justification for Z magnification

In most situations simulated in SMS, the data range in the horizontal direction is not similar to the data range in the vertical direction. For example, when simulating a river reach, the river may cover miles (or kilometers) along the length of the river, but in the z-direction, the change in elevation will only be in the tens to hundreds of feet (meters). In an opposite situation, when working with a coastal circulation model in geographic coordinates, the horizontal variation of the data may only be a few degrees, while the vertical change in depth can be thousands of meters. When displaying data in plan view, this inconsistency of data ranges does not cause a problem. However, when attempting to view data in an oblique view (from an angle in three dimensions), the first case of a long river ends up looking like a flat plane while the second case is just a mass of vertical bumps.

To allow for intuitive display of the data in three dimensions SMS allows the specification of a Z magnification term. This scale factor exaggerates or reduces the relief of the data in the simulation.

Auto Z magnification

SMS also includes the option to compute a Z magnification term automatically. This option is turned on by default. This means that every time SMS frames the data in a display, the Z magnification term is computed to ensure that the scaled span of the vertical data is just under 10% of the horizontal data. This prevents the data from becoming too flat (unless it is totally flat) and prevents the relief from becoming to drastic or dramatic.

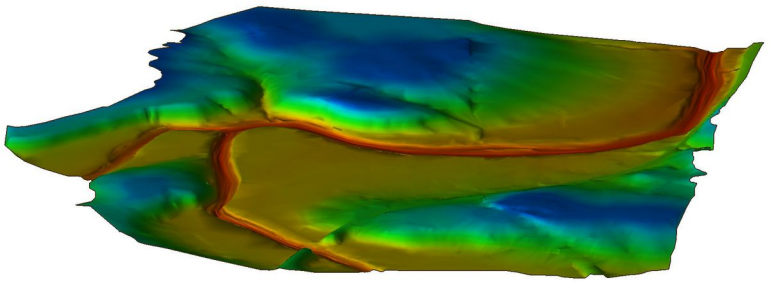
Since the Z magnification value is computed when SMS frames the data, modifications to the data that change the Z range are not incorporated into the magnification value until a frame command is encountered. This may result in difficulties rotating a scene in three dimensions. Operations like generating new elevation data for a mesh by interpolation from a raster or scatter set may cause this to occur.

The user can disable the auto Z magnification feature by unchecking the toggle next to the *Auto z-mag*. When the toggle is unchecked, an edit field appears allowing the user to specify a Z magnification value. The value is set to the previously computed auto Z magnification.

Contour Options

At a glance

- Visualize scalar datasets
- Linear, color filled or both at the same time
- Variable level of transparency
- Full control of ranges and colors
- Precision control for labels and legends



3D view of color filled contours

SMS can generate contours from a scalar dataset. Contour display is enabled using in the Display Options dialog. Unique contour display options can be set for each module that uses scalar datasets. Contour options can also be set for an individual dataset.

The module contour options can be edited by:

- The menu command *Data* | **Contour Options**. The data menu is available in most modules. See Module Specific Menus for more information.
- Opening the Display Options dialog
- The **Contour Options Macro**

The dataset specific contour options for an individual dataset can be edited by:

- Right-click on the dataset in the Project Explorer and choose **Dataset Contour Options**. This will open a dialog where you can define dataset specific contour options. If you want to go back to using the module contour options, right click on the dataset in the *Project Explorer* and choose **Clear Dataset Contour Options.**

Data Range

The name, minimum value, and maximum value for the active time step of the dataset are shown. These values are sometimes useful when choosing an appropriate contour interval.

A minimum and maximum contour value can be specified, restricting the contours interval which will be shown. If the range is not specified, SMS will automatically choose a range based on the minimum and maximum value for the active time step.

Contour Interval

The contour interval is user controlled. Options include:

- *Number of Contours* – Specify the number of contours to use. SMS will automatically determine the value for each interval based on the specified range or dataset range.
- *Specified Interval* – Specify the interval to use for contours. SMS will automatically determine the number of contours needed based on the specified range or dataset range.
- *Specified Values* – Specify the number of contours to use and interval.

The items in the upper right section of the *Contour Options* dialog control the display of a contour legend and the option to accentuate some of the contours. If the *Show Color Legend* option is selected, and the contours are not being displayed as a single color, a legend of colors and corresponding data set values is displayed in a corner of *Graphics Window*. For color filled contours, this legend is a vertical strip of colors with text labels for the contour levels. If the contours are being displayed as linear segments or cubic splines, the legend is displayed as a series of

contour level values and a line drawn in the color corresponding to that level. The size, location, label and font for the legend are set using the *Legend Options* dialog. If the user enters the title "DS" for the legend title, the name of the current dataset is used. If the user enters "DS:TS" the current dataset and time step are used as the title.

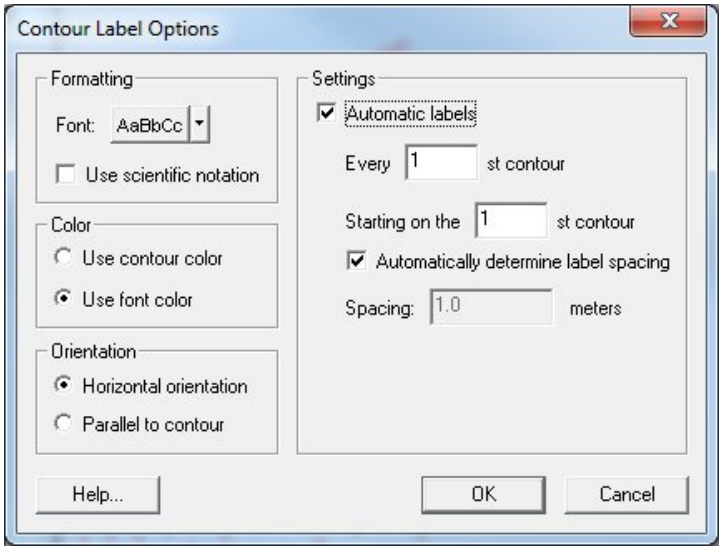
The options in the middle of the right side of the dialog control how the contours are computed and displayed. Three contouring methods are available:

- The default method is *Normal Linear Contours* and causes the contours to be displayed as piece-wise linear strings.
- If using the *Color* fill between contours method, the same linear contour strings are computed, but the regions between adjacent contour lines is filled with a solid color.
- If using the Cubic Spline Contours method, the contours are computed in strings and drawn as cubic splines. Drawing the contours as splines can cause the contours to appear smoother. Occasionally, loops appear in the splines or the splines cross neighboring contour splines. These problems can sometimes be fixed by adding tension to the splines. A tension factor greater than zero causes the cubic spline to be blended with or converge to a linear spline based on the same set of points. A tension factor of unity causes the cubic spline to coincide with the linear spline.

In the lower right corner of the *Contour Options* dialog, two buttons allow the user to specify the contour colors and the contour labeling options.

Contour Labels

The **Contour Label Opts** command in the *Data* menu is used to access the *Contour Labels Options* dialog which can be used to set the label color, font, spacing, size, etc. The dialog may also be invoked through the *Contour Options* dialog.



Contour Label Options dialog

Labels can be added to contours one of two ways:

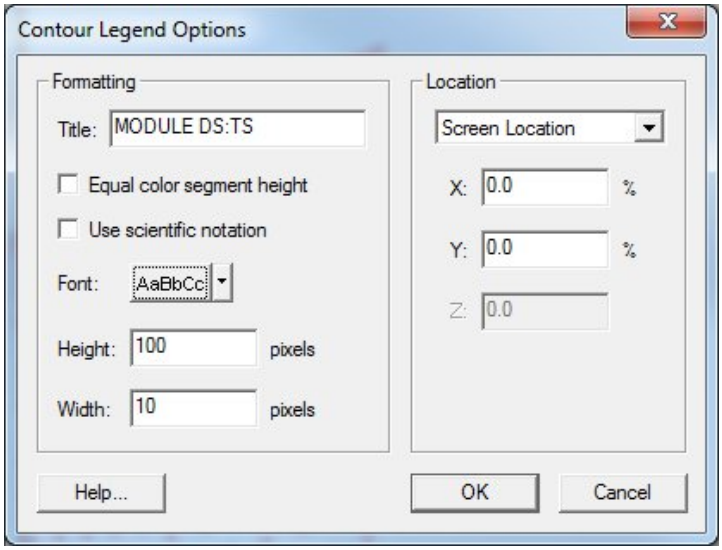
1. The upper left portion of the *Contour Label Options* dialog controls the generation of automatically spaced contour labels. The user can toggle the generation of automatic contour labels on or off. If the toggle is on, the user also specifies which contours should be labeled and the distance along the contour between labels.
2. In some modules, contour or function labels can be added manually to an image by selecting the **Contour Labels** tool in the *Tool Palette* and clicking on the mesh or grid where a label is desired. If the Place on contours option in the upper right portion of the *Contour Label Options* dialog is selected, the label is moved to the closest contour and the contour is labeled there. If the Place under cursor option is selected, the label shows the value of the point at the click location and is placed there. This option is useful to post data set value labels in regions where there are no contours. Contour labels can be deleted by holding down the SHIFT key while clicking on a label.

The bottom portion of the *Contour Label Options* dialog control how the labels appear. On the left side, the user can control how many digits of accuracy are desired. The default will match the contour legend. On the right side, the

user can select a color and font for the label. For labels on contours, the user may also specify that the contour be oriented to lie along the contour.

Contour Legend Options

The *Contour Legend Options* controls the formatting and location of a displayed legend. If a contour dataset exists and is displayed, the legend will be shown if the Legend check box on the *Contour Display Options* dialog is checked. This window is accessed only from the **Legend Options** button on the *Contour Display Options* page of the Display Options dialog.



Contour Legend Options dialog

The *Formatting* section includes fields for the Title and Units displayed with the legend, a **Font selection** button for text style, and Height and Width fields for legend size.

Since contour datasets can be displayed for multiple modules at the same time and, therefore, multiple contour legends can be displayed, the Title field can include keywords for convenient labeling. The following title keywords are case sensitive:

- "MODULE" – will be replaced with the title of the contour dataset's module
- "DS" – will be replaced with the name of the currently selected contour dataset
- "DS:TS" – will be replaced with the name of the currently selected contour dataset followed by the current time step

A title of "MODULE DS:TS" is best since it will automatically update as contour dataset selection changes.

The *Units* field includes the single case sensitive keyword of "DEFAULT", which will be replaced with the velocity units of meters per second (m/s) or feet per sceond (ft/s) based on the current coordinate system's horizontal units.

The Location section includes a combo box for specifying the location of the displayed legend. The locations include:

- Top left corner
- Bottom left corner
- Top right corner
- Bottom right corner
- Screen location – specify the location based on screen precentages
- World location – specify the coordinate location

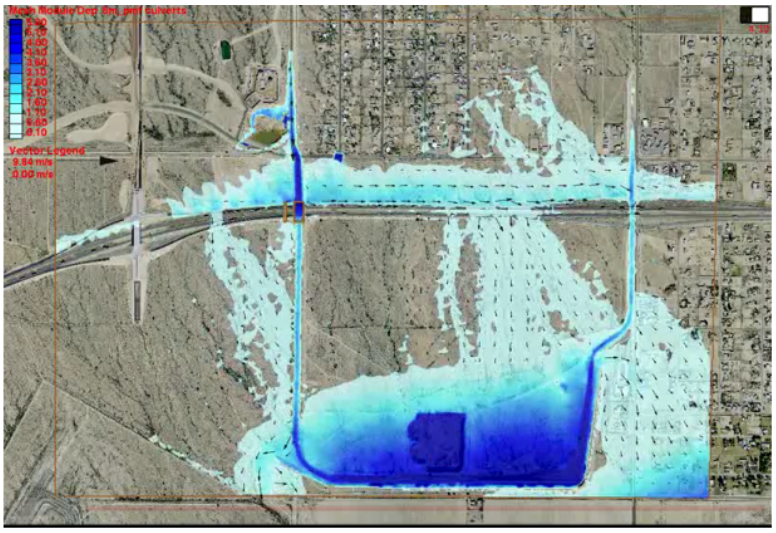
Related Topics

- Color Ramps
- Display Options

Vector Display Options

At a glance

- Visualize vector datasets as arrows
- Constant size or vary by magnitude
- Show just a range of magnitudes
- Color by magnitude



Vectors display

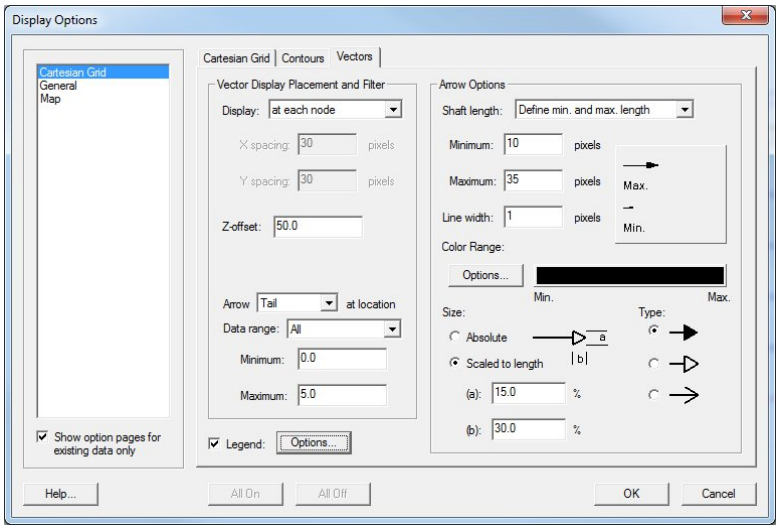
SMS can generate contours from a vector dataset. Vector display is enabled using in the *Display Options* dialog. Unique vector display options can be set for each module that uses vector datasets.

The options used to generate vectors can be edited by:

- The menu command *Data* | **Vector Options**. The *Data* menu is available in most modules. See Module Specific Menus for more information.
- Opening the *Display Options* dialog.
- The *Vector Options* Macro.

Vector Display Placement and Filter

Display



Display Options dialog showing the *Vectors* tab

The *Vector Display Placement and Filter* section allows the user to control the density of arrow to be displayed. In a very dense mesh, a large number of data points may be displayed very close together on the screen. Therefore, if a vector is displayed at every point, the picture can become a jumble of vectors on top of each other. One way to avoid this is to zoom in on a specific portion of the mesh, so the nodes are not displayed so close together. However, if the desired region of the mesh is still too dense, or zooming in is not acceptable, the user can filter the displayed vectors. SMS provides the following display locations for filtering vectors:

- **At each node –** Display vectors at each node (data location)
- **At corner nodes only –** Display vectors at corner nodes only (useful for quadratic elements)
- **On a grid –** Display vectors on a grid (uniform grid overlaying the mesh or grid geometry). The x and y spacing of the grid are specified in pixels, so regardless of the zoom level the grid remains constant.
- **On a coverage –** Display vectors on the vertices of feature arcs in the specified coverage (active or specified).

Origin

This combo box allows the user to display arrows at a constant elevation. It has the following options:

- Relative to bed
- Relative to max elevation
- Absolute elevation

Offset

To ensure that the vectors are visible, the Z-offset can be specified to display the vectors a distance above the geometry. The vectors can be filtered further by displaying only a range of magnitudes instead of all vectors.

Arrow Location

Vector arrows can be displayed with the following placement options:

- **Tip –** Display vectors with the vector arrow tip at the data location.
- **Tail –** Display vectors with the vector arrow tail at the data location.
- **Center –** Display vectors with the vector arrow shaft centered over the data location.

Arrow Options

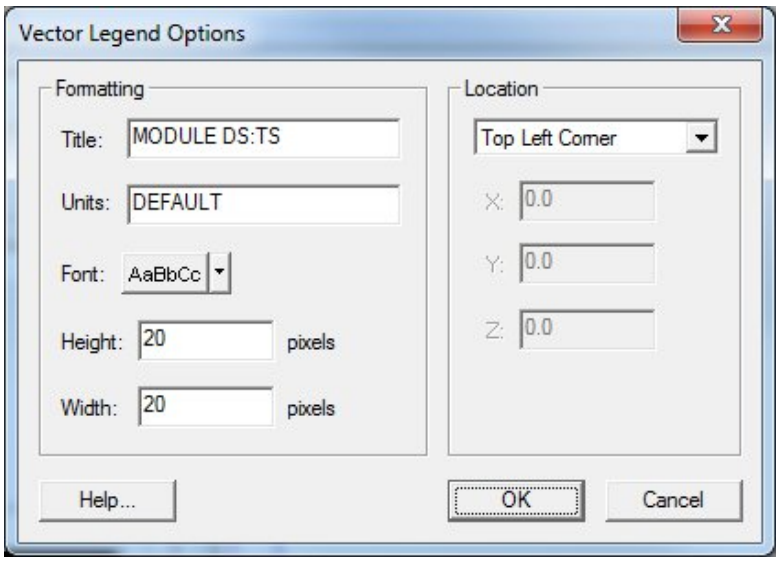
In the *Arrow Options* section, the user specifies how the arrows will appear in the graphics window. Arrow shaft length can be a constant length, a scaled length, or a range of lengths. The line width of the arrow can also be adjusted. Arrows may be a constant color, or shaded according to magnitude. If a ramp of colors is desired, the color of the vector is extracted from a ramp. By default, the arrow with the smallest magnitude is displayed in the color at the bottom end of the ramp, and the arrow with the largest magnitude is displayed in the color at the top of the ramp. Intermediate magnitudes are interpolated to select an appropriate intermediate color. Alternately, the user can define the magnitudes that map to the top and bottom of the ramp. If this option is used, any arrow with a magnitude lower than the minimum is displayed in the color at the bottom of the ramp, and any arrow with a magnitude greater than the maximum is displayed with the color at the top of the ramp. Users can specify the shape of the arrow head with absolute head length and width values or values proportional arrow length. The style of arrow head is based on the selection of the solid, hollow, and line head types. A preview of the arrows (fixed, or maximum and minimum) based on the selected options are displayed in this section of the dialog.

Legend

The *Vector Options* also includes a toggle for the display of the vector legend. The vector legend displays the significance of the size of the vectors displayed on the grid. Selecting the legend **Options** button opens the *Vector Legend Options* window.

Vector Legend Options

The *Vector Legend Options* controls the formatting and location of a displayed legend. If a vector dataset exists and is displayed, the legend will be shown if the *Legend* check box on the *Vector Display Options* dialog is checked. This window is accessed only from the Legend **Options** button on the *Vector Display Options* page of the *Display Options* dialog.



Vector Legend Options dialog

The *Formatting* section includes fields for the *Title* and *Units* displayed with the legend, a **Font** selection button for text style, and *Height* and *Width* fields for legend size.

Since vector datasets can be displayed for multiple modules at the same time and, therefore, multiple vector legends can be displayed, the Title field can include keywords for convenient labeling. The following title keywords are case sensitive:

- "MODULE" – will be replaced with the title of the vector dataset's module
- "DS" – will be replaced with the name of the currently selected vector dataset
- "DS:TS" – will be replaced with the name of the currently selected vector dataset followed by the current time step

A title of "MODULE DS:TS" is best since it will automatically update as vector dataset selection changes.

The Units field includes the single case sensitive keyword of "DEFAULT", which will be replaced with the velocity units of meters per second (m/s) or feet per sceond (ft/s) based on the current coordinate system's horizontal units.

The Location section includes a combo box for specifying the location of the displayed legend. The locations include:

- Top left corner
- Bottom left corner
- Top right corner
- Bottom right corner
- Screen location – specify the location based on screen precentages
- World location – specify the coordinate location

Related Topics

- Contour Options
- Color Ramps
- Display Options
- Visualization

Visualization for 3D Solutions

3D solutions can be viewed on a 3D mesh (VTK mesh module). A 3D solution includes data at multiple z layers and becomes a volume. This is not to be confused with displaying 2D results that represent a surface with 3D coordinates. A 2D solution represents depth averaged values and cannot represent a changing solution in the z direction.

3D Fence Diagram

A 3D fence diagram displays solution data on user specified vertical planes. Fences can be useful to illustrate how a 3D solution varies with depth. Multiple fences can be displayed at the same time to help visualization the solution.

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:

- A 3D mesh with solution datasets.
- 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.

Iso-surfaces

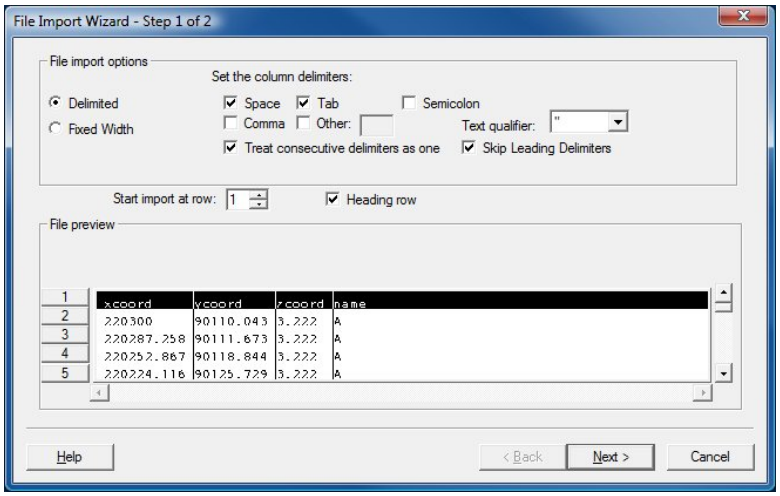
Iso-surfaces can be used to display 3D solutions. The display options for iso-surfaces are set using the contour options in the *Display Options* dialog.

2.6. File Import Wizards

File Import Wizard

SMS can import many files generated by other software in their native format. Refer to Importing Non-native SMS Files for a list. For files that are not included in the list, SMS provides the **Text Import Wizard**.

The *Text Import Wizard* enables users to import many different types of data into SMS. The *Text Import Wizard* is initialized by selecting a *.txt file in the **Open** command from the *File* menu. The wizard has two steps:

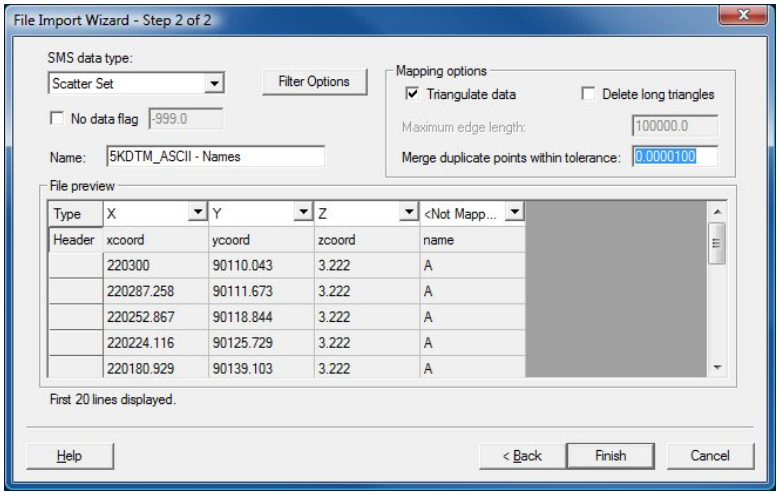


Step 1 of the Text Import Wizard – File import options

Step 1 – Delimiting Columns

The first step in the wizard allows users to delimit the data into columns. The following options exist to delimit the data:

- **Delimited –** For the *Delimited* option, typical delimiters are included as well as an option for users to specify a delimiter.
- **Fixed Width –** Columns can be specified with a fixed width by clicking on the ruler bar or the window with the data. Break lines can be dragged, and they can be deleted by double-clicking on the break line or dragging them off the screen.



Step 2 of the Text Import Wizard – File format

The user can specify the starting row the data will be imported at. If your data has a row of headings, you can indicate such and SMS will use the headings in the next step to determine what kind of data each column represents.

Step 2 – Assigning Column Types

The first 20 lines of the file are displayed in a spreadsheet according to the file outline specified in step 1. This step lets you pick what kind of data you are importing (see Supported File Formats). A "no data flag" can be specified for the file. This is a number that, when encountered in the file, tells SMS to mark the value as "NULL" or "no data". For example, a water surface elevation dataset would assign a no data flag to dry nodes.

The data in the columns are identified by selecting the type in the combo box at the top of each column in the spreadsheet. If a row of headings exists, SMS will automatically select the proper type if it recognizes the heading. Otherwise they are Not Mapped by default. The available column types changes depending on the SMS data type selected. Certain column types must be mapped for each file format before the user can progress to the next step in the wizard. The name of each column is changed by editing the Header cell.

Mapping Options

When reading in a scatter set or mesh data, the following mapping options are available:

- **Triangulate data –** Triangulates the scatter vertices / mesh nodes
- **Merge duplicate –** Merges duplicate scatter vertices / mesh nodes based on the specified tolerance
- **Delete long triangles –** Deletes scatter triangles with an edge length longer than the specified edge length
- **Append mesh –** Appends the mesh nodes to the existing mesh

Filter Options

When importing a Scatter Set, pressing the **Filter Options** button will open the *File Import Filter Options* dialog. The filter options are useful when reading scatter sets that are too large for SMS to successfully read in. Once the scatter set has been read into SMS, the more sophisticated normals filtering algorithm can be used.

Additional Options

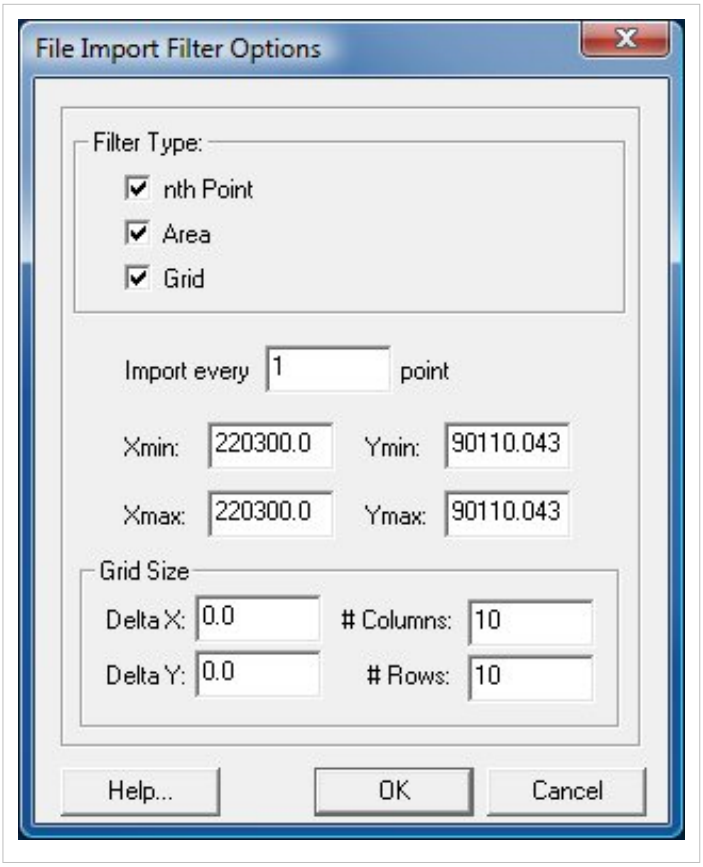
After the data have been imported, the coordinate transformation tools can be used to transform and translate the data.

Related Topics

- File Formats

File Import Filter Options

When importing a Scatter Set using the *File Import Wizard*, pressing the **Filter Options** button will open the *File Import Filter Options* dialog. The filter options are useful when reading scatter sets that are too large for SMS to successfully read in. Once the scatter set has been read into SMS, the more sophisticated normals filtering algorithm can be used.



Filter Options

The following filter options are available:

- **nth Point –** Simple method to reduce the scatter set size by reading a reduced number of vertices from the file. Reading every 2nd point will result in a 50% reduction in vertices, every 4th point will result in a 75% reduction, etc.
- **Area –** Only reads points falling within the specified x, y boundary. Useful for filtering data outside of the area of interest.
- **Grid –** Scatter vertices are created on a user defined grid. Each vertex has a "bucket" around it. The z-value is assigned to the vertex based on the average value of the vertices in the "bucket."

Related Topics

- File Import Wizard
- Normals Filtering

File Import Wizard Supported File Formats

The following types of data can be imported into SMS via the File Import Wizard.

- 2D Scatter Set Vertices
- 2D Mesh Nodes
- Feature Points
- Observation Data
- Wind, Wave, Water level

A description of the fields (columns) that SMS recognizes when importing text files is provided in the tables below.

2D Scatter Vertices

Field	Type	Required	Comments
X	Number	yes	X-location
Y	Number	yes	Y-location
Pt Name	Text	no	
Vector X	Number	no	Used in conjunction with Vector Y Field
Vector Y	Number	no	Used in conjunction with Vector X Field
Vector Magnitude	Number	no	Used in conjunction with Vector Direction Field
Vector Direction	Number	no	Used in conjunction with Vector Magnitude Field
Scalar Data	Number	no	
Breakline	Text or Number	no	See Scatter Breakline Options for a discussion of Breakline Delimeters

"id"	"x"	"y"	"xylene"	"toluene 0.0"	"toluene 2.0"
"OW-21"	32.4	5234.3	300	999	999
"0W-22"	93.4	5832.3	84	398	401
"0W-23"	83.3	8438.2	89	47	52

2D Mesh Nodes

Field	Type	Required	Comments
X	Number	yes	X-location
Y	Number	yes	Y-location
Z	Number	yes	Z-location

"x"	"y"	"z"
32.4	5234.3	12.34
93.4	5832.3	13.47
83.3	8438.2	21.54

Feature Points

Field	Type	Required	Comments
Name	Text	no	
X	Number	yes	X-location
Y	Number	yes	Y-location
Z	Number	yes	Z-location

"name"	"x"	"y"	"z"
"Pt. 1"	32.4	5234.3	12.34
"Pt. 2"	93.4	5832.3	13.47
"Pt. 3"	83.3	8438.2	21.54

Observation Points

Field	Type	Required	Comments
Point Name	Text	no	
X	Number	yes	
Y	Number	yes	
Z	Number	no	
Measurement	Text	no	Measurement name. Multiple measurements allowed.
Interval	Number	no	

"name"	"x"	"y"	"z"	"hd"	"int"
"OBS_Q5"	23.3	44.2	32.2	567.5	1.2
"OBS_Q6"	83.3	84.3	32.2	555.3	1.4
"OBS_Q7"	85.3	39.3	33.2	999	0

PTM Trap Output Data

Field	Type	Required	Comments
Step	Number	no	Time step index of an entry event for a trap
Date	####/##/##	yes	Date of an entry event - must have a year, month and day
Time	##:##:##.####	yes	This column must have hour, minute and second of the entry event
Particle	Number	no	This column could be used to reference other PTM output files. Not used in import wizard
Trap	Number	yes	Defines which trap this parcel entered
Value column	Number	no	This is an optional column. There may be more than one.
Filter column	Number	no	This is an optional column. There may be more than one.

See PTM Trap Output for more information on reading PTM trap output files.

Wind, Wave, Water level

Field	Type	Required	Comments
Date/Time	Number/Number	yes	
Date	Number	yes	
Time	Number	yes	
Primary Height	Number	yes	Used in conjunction with Primary Period & Direction Field
Primary Period	Number	yes	Used in conjunction with Primary Height & Direction Field
Primary Direction	Number	yes	Used in conjunction with Primary Height & Period Field
Secondary Height	Number	no	Used in conjunction with Secondary Period & Direction Field
Secondary Period	Number	no	Used in conjunction with Secondary Height & Direction Field
Secondary Direction	Number	no	Used in conjunction with Secondary Height & Period Field

Related Topics

- File Import Wizard

2.7. Export Options

Export Tabular File

SMS can export much of the data managed and displayed int the system to a tabular data format. This type of file is sometimes referred to as a CVS or comma separated values file. In actuality, the delimiter may be commas, spaces, tabs, or other typical white space characters.

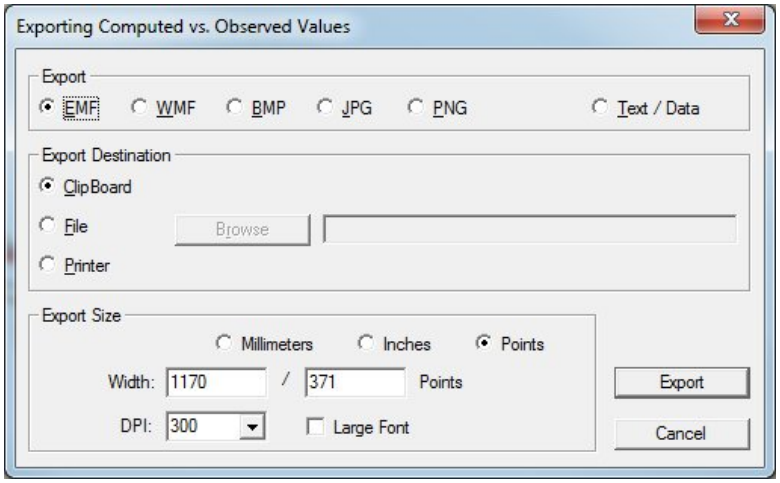
The data to be exported depends on the active module when the command is issued. For example, when the Mesh module is active, mesh nodes will be saved and when the Scatter module is active, scatter vertices will be saved. The user supplies the file name to contain the data.

If data points are selected when the command is issued, the option is given to output all data points of the defined type, or only the selected points. A default header is provided which defines the number of data points represented.

The user then selects the nubmer of columns and number of digits of precision to save each value. For each column the user then selects the data to be stored in that column. This can include the *x location*, the *y location* or any data set currently loaded into SMS for that data type. Transient datasets may be saved in a range of columns.

Exporting Profile Dialog

The *Exporting Profile* dialog allows you to export the plot data.



Exporting Profile dialog

Export

- Image Export
  - EMF
  - WMF
  - BMP
  - PNG
- Text / Data

Export Destination

- ClipBoard
- File – Creates an ASCII text file of the data
- Printer – Exports the data to your printer

Export Size

If exporting to an image format, allows the image size and resolution to be specified. This sections contains the following sections:

- Millimeters
- Inches
- Points
- Width
- DPI
- Large Font

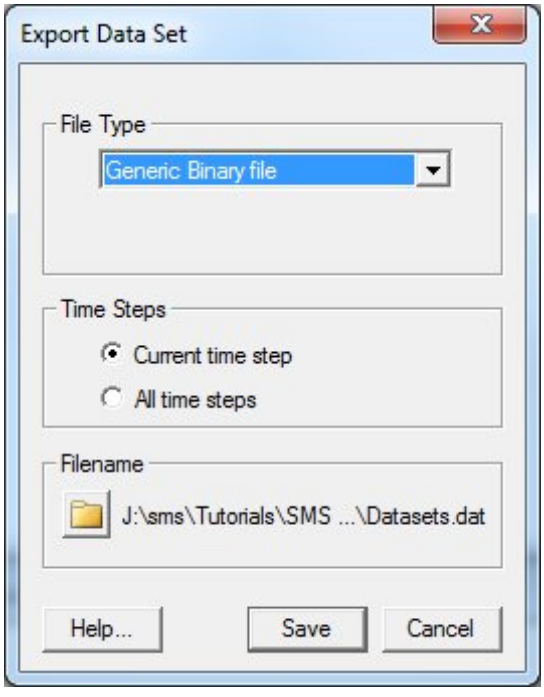
Related Topics

- Plot Window

Export Dataset Dialog

The *Export Dataset* dialog is used to export scalar or vector datasets. To open the *Export Dataset* dialog, use the dataset right-click menu.

- File Type
  - Binary Dataset Files (*.dat) – Benefits include fast read/write times, small file size.
  - ASCII Dataset Files (*.dat) – Can be imported into Microsoft Excel and viewed with standard text editors.
  - HDF5 Dataset Files (*.h5) – Benefits include fast read/write times, small file size, native compression.
- Time Steps
  - Current time step – Exported dataset will only contain the current time step.
  - All time steps – Exported dataset will contain all time steps.
  - Filename – Path and filename used for exported dataset.



Export Dataset dialog

Related Topics

- Binary Dataset Files (*.dat)
- ASCII Dataset Files (*.dat)
- XMDF Files

Export Map Data in Shape Format

Export

To export maps in shape format:

1. Load or create an existing map
2. Select *File*| **Save As**
3. Select *Save as type:* and choose *Catalog file (*.xml)*
4. In the menu item, select *Shape* as the format and verify the correct coverage is selected (has a check mark)
5. Click **Ok**

An XML file is created which outlines the directories of the shape files.

If the coverage created is a generic 2dm, feature point and feature arc boundary condition data will also be saved.

Import

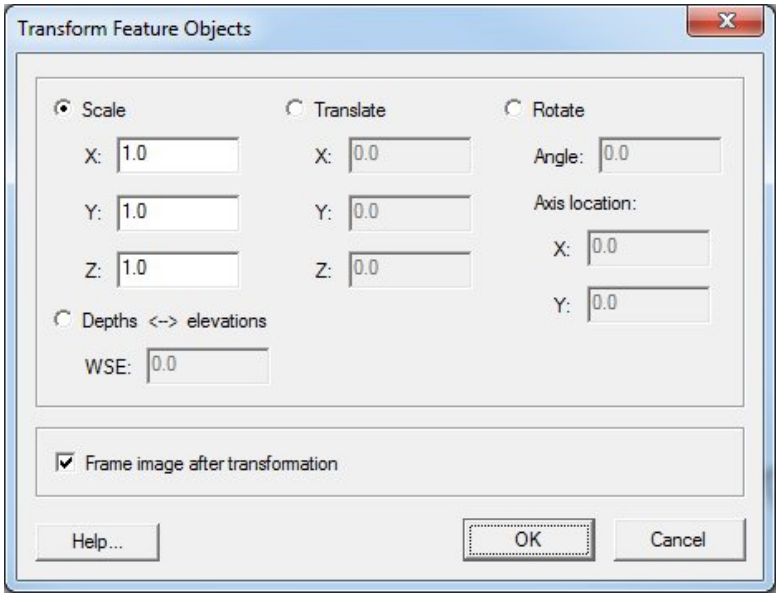
Open the saved *.xml file to import the data from the shape file.

2.8. Geometric Tools

Data Transform

At a glance

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



Transform Data

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. The following transformation types are available:

- **Scaling**: Scaling factors for the X, Y, and/or Z directions are entered. To prevent scaling a specific direction, the default value of 1.0 should be used.
- **Translation**: Translation values for the X, Y, and/or Z directions are entered. To prevent tranlation in a specific direction, the default value of 0.0 should be used.
- **Rotations**: When rotation is selected, the set of options on the right side of the dialog become available to define the center of rotation. If the *Specified Point* option is used, then the center of rotation is explicitly defined. Otherwise, after clicking the OK button from the *Nodes Transform* dialog, the user must click in the graphics window at the Point or on the Node about which the rotation should occur. The rotation will occur counter-clockwise by the specified angle around the specified center of rotation.
- **Datum Conversions**: Convert between elevation and depth data.

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.

The *Transform Feature Objects* dialog can be reached either through the *Feature Objects* menu or through the right-click menu of the selected feature object.

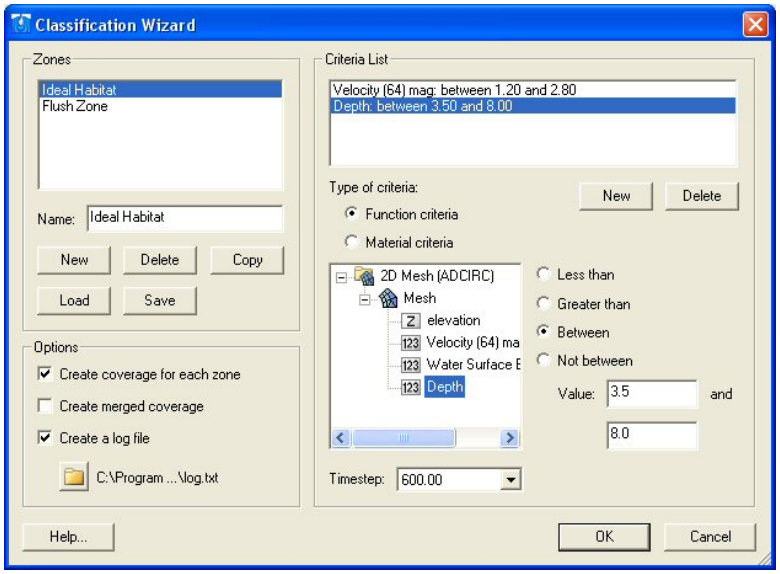
Related Topics

- Scatter Data Menu
- Map Feature Objects Menu

Zonal Classification

At a glance

- Generate a map coverage identifying areas that meet specific requirements
- Requirements can be based upon dataset values such as less than a specific value or based upon materials in an area property coverage



Zonal Classification

Zonal classification is a tool that will 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.

A zone may contain one or more criteria. A zone may identify areas that have a range of depths and also a range of velocities. Multiple zones can be evaluated at the same time. If you are using multiple zones, you can have SMS create a separate coverage for each zone, a coverage that includes all the zones where each polygon's material identifies the zone or zones the polygon is valid for, or both a coverage for each zone and a merged coverage.

Zones and criteria associated with them can be saved and loaded from within SMS. This makes it easier to evaluate multiple scenarios using the same set of criteria.

If desired, SMS can create a log file that contains information such as the areas found in each zone.

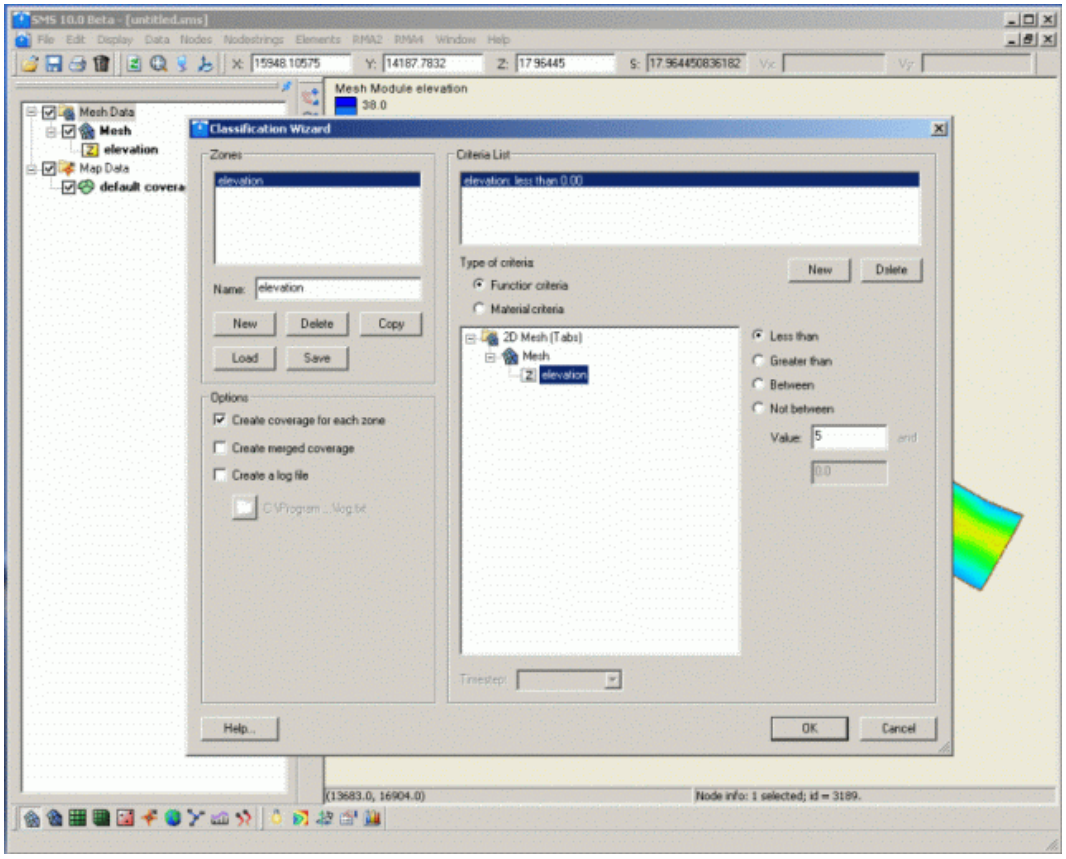
Example

One application of zonal classification is to help quantify the amount and quality of fish habitat. Certain types of fish prefer or require different depths, velocities, and substrate. These preferences depend upon the life-cycle stage for fish.

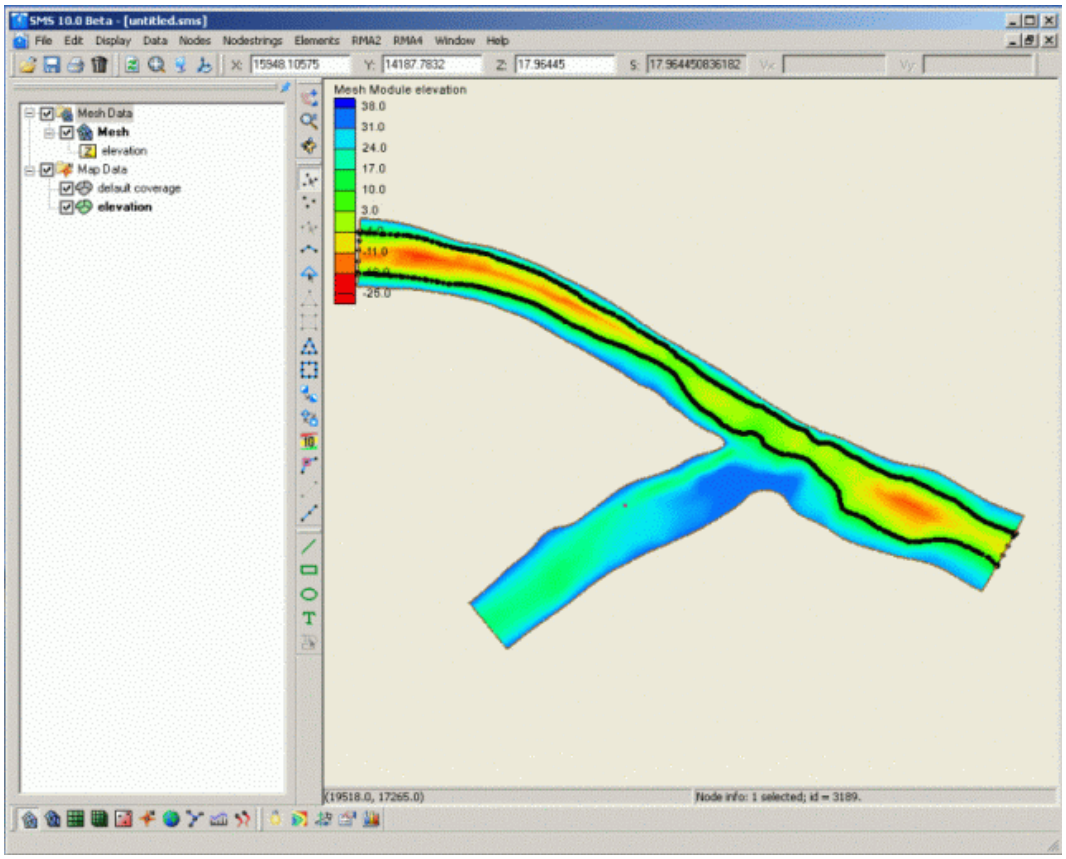
The following demonstrates how you could use zonal classification for a very simplified example to identify areas meeting a certain set of criteria. The example is fictitious and uses made up criteria.

For the first sample criteria we will identify areas that have an elevation of 5 ft or less. This is done by:

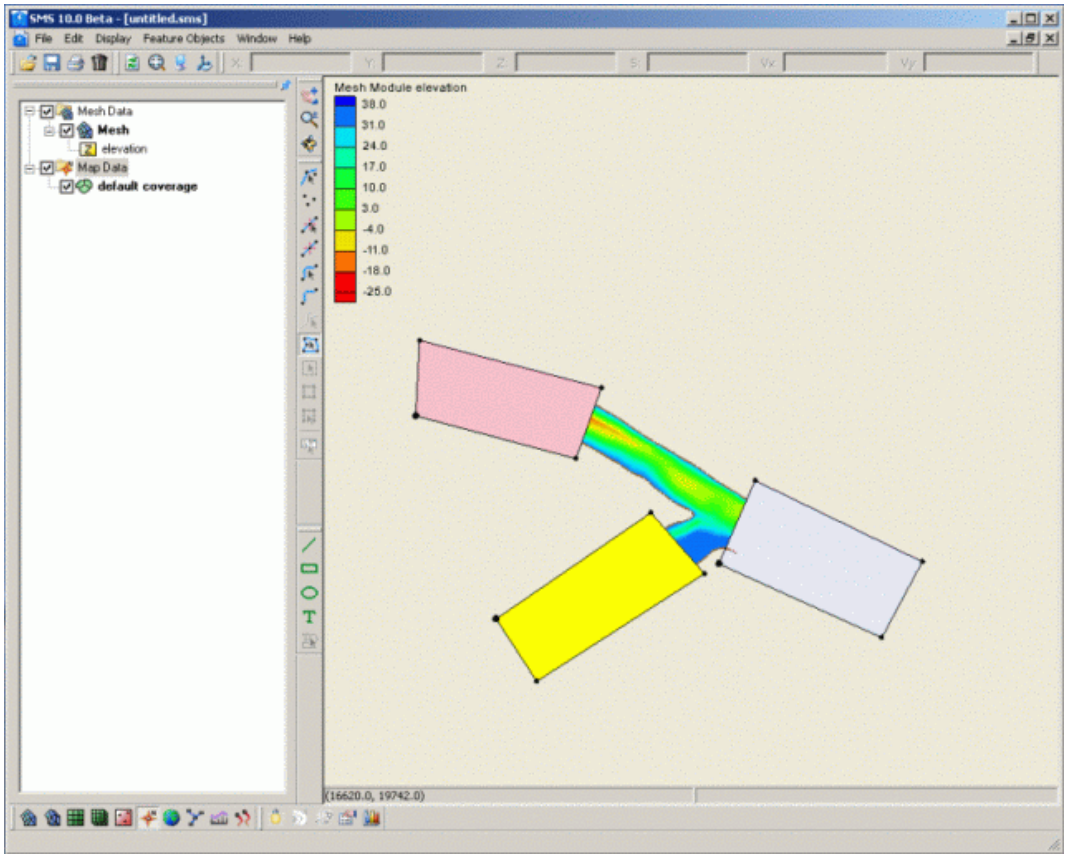
1. Creating a new zone based upon this criteria.
2. Creating a new criteria based upon a functional criteria.
3. Specify the elevation dataset and the criteria to be less than 5 ft.



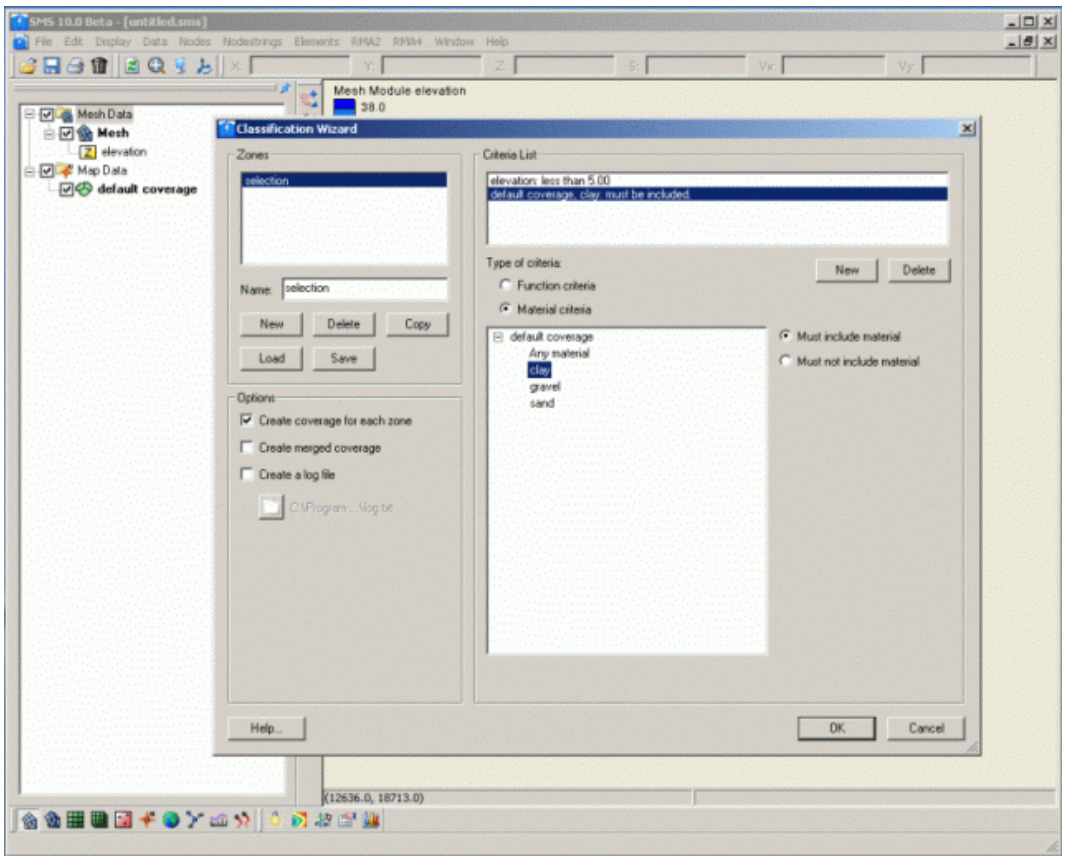
After executing the zonal classification a new coverage was created and polygons identify the areas that meet the requirements of the zone (in our case the elevation less than 5 ft). Assuming that our criteria identifies a target habitat, it is easy to see the areas that meet the criteria.



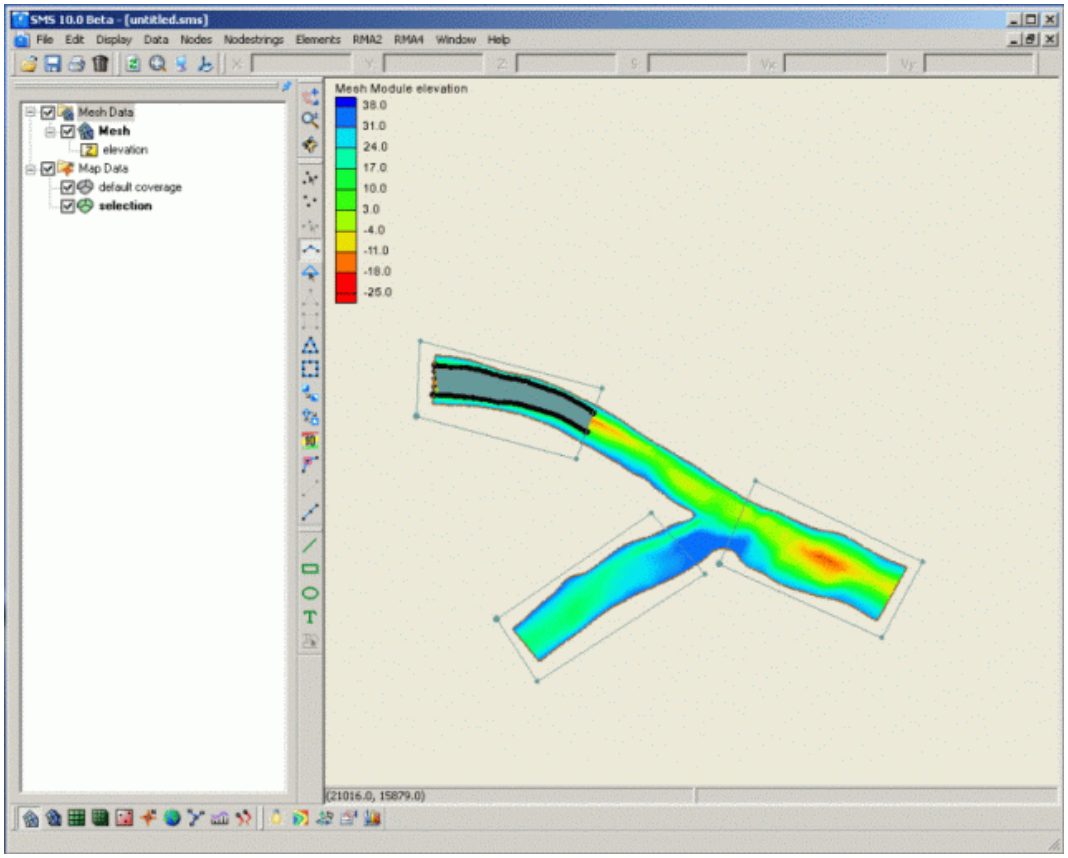
In addition to the elevation, let's assume that our target habitat also requires a specific type of substrate (bottom sediment type). For this example, we have created polygons in a area property coverage and identified areas with different substrates (again this information is fictional).



We added another criteria to our original zone so that the zone only includes areas that have clay substrate.



Now polygons only exist where the elevation is less than 5 ft and the substrate is clay.



External Links

- Vector based classification of zones from distributed datasets or GIS polygon data – Russell D. Jones, 2003 – Thesis (M.S.) Brigham Young University Dept. of Civil and Environmental Engineering. TA 4.02.J65482 2003

2.9. Images

Images

At a glance

- Multiple images can be read/viewed at the same time
- Many image formats are supported including JPG, TIFF, PNG, MrSID, and ECW
- Images can be geo-referenced to view images along with other data
- Images can be draped over mesh or scatter data
- Image pyramids can be created which improves viewing at various zoom levels

A color aerial photo under a semi-transparent topographic map

A background image is a digital picture detailing topographic and land use attributes of an area of interest. In SMS, these digital pictures are typically maps or aerial photos that are useful in locating and defining the boundaries of the study area and the extents and features in the project domain. Images can be imported to SMS and displayed in the background to aid in the placement of objects as they are being constructed or simply to enhance a plot. Images can also be draped or "texture mapped" or draped onto a scatter dataset (TIN) or finite element mesh.

Supported Image File Formats

- Enhanced Compression Wavelet [1] (*.ecw)
- Graphics Interchange Format [2] (*.gif)
- Joint Photographic Experts Group [3] – (*.jpg/jpeg)
- Multiresolution Seamless Image Database [4] – (*.MrSID)
- Tagged Image File Format [5] – (*.tiff)

Importing an Image

Images can be opened in SMS using the *File* | **Open** menu command. They can also be added to a simulation by dragging and dropping the file into SMS. The images are then added to the image folder in the Project Explorer and displayed in the background to aid in the placement of objects as they are being constructed or simply to enhance visualization of the project domain. All TIFF images are converted to JPEG when they are read in. Multiple images can be imported into SMS.

Exporting Image Files

Images (or files related to images) are saved in the following ways:

Save As

The image displayed in the *Graphics Window* can be saved as a Bitmap Image File (*.bmp) or JPEG Image File (*.jpg, *.jpeg) using the *File* | **Save As** menu command and specifying an image file as the save as type. The resolution of the saved image is based on the screen resolution and scale factor specified in the *Preferences* dialog.

Project File

When a project file is saved any images that are part of the project are saved. The registration information is saved in the project file to provide the coordinate system information for the image.

Copy to Clipboard

When the *Edit* | **Copy to Clipboard** menu command is selected, the image currently displayed in the *Graphics Window* is copied to the clipboard. This image can then be pasted into reports or other programs by pressing *CTRL + V*. The resolution of the saved image is based on the screen resolution and scale factor specified in the *Preferences* dialog.

Export World File

A World File can be exported for the selected image by right-clicking on the Project Explorer and selecting the *Export World File* command. A world file is a special file that contains registration data that can be used to register images.

Geo-Referencing

A geo-referenced image includes information specifying the real world size and location of the image. The coordinate system can be embedded in the file or given in a separate file called a world file (for example: a TIFF world file, *.tfw). When geo-referenced image files are opened, SMS automatically registers the image to the real world coordinate location specified. In the case where a separate world file is used, SMS will search for it and register the image if the world file has the same filename prefix as the image file and is in the same folder.

If the image file is not geo-referenced then you will have to register the image manually. (See Registering an Image)

When the SMS project is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Display Options

Image display options are changed in the Project Explorer. Display options include:

- **Visibility –** The visibility of an image is turned off by toggling the check box next to the image in the Project Explorer.
- **Transparency –** The transparency of each image can be changed by right-clicking on the image in the Project Explorer and selecting **Transparency** from the right-click menu.

Image Deletion

A single image is deleted by right-clicking on the image in the Project Explorer and selecting the **Delete** command. To delete all images, you right-click on the Images folder in the Project Explorer and select **Clear Images**.

Dynamic Imagery from ArcGIS

Starting in SMS 11.0 (32-bit only), dynamic background images can be accessed from the web through ArcGIS. If you have ArcGIS on your computer, you can use the GIS module within SMS to get background imagery that updates on the fly from the internet. To access these images, follow the steps below.

1. Switch to the GIS module (select the globe in the bottom left of the SMS screen)
2. Select *Data* | **Enable ArcObjects**
3. Select *Data* | **Add Data…**
4. Browse to the "C:\Program Files (x86)\SMS 11.0\Supporting Files\GIS Layer Files" directory
5. Select the desired layer
6. Select **Add**

Note: This feature is only available in the SMS 11.0 beta (32-bit). Since this version is still in beta, it should not be used in place of the release version of SMS (SMS 10.1).

Related Topics

- Image Pyramids
- Import from Web
- Registering an Image

References

[1] http://en.wikipedia.org/wiki/Enhanced_Compressed_Wavelet_File
[2] http://en.wikipedia.org/wiki/Gif
[3] http://en.wikipedia.org/wiki/Jpeg
[4] http://en.wikipedia.org/wiki/Mrsid
[5] http://en.wikipedia.org/wiki/Tiff

Image Pyramids

When an image is imported, the user has the option to “Build Pyramids.” The build pyramids option enhances the on-screen display of images by resampling the imported image at different resolutions. Based on the current extents of the graphics window, one of the resampled images is displayed to deliver the best on-screen display.

There are a few points to keep in mind when building pyramids. The initial generation of pyramid files can take several minutes, depending on the size of the original image and your computer hardware. Building pyramids uses more memory RAM, but the software performance is not otherwise affected. Building pyramids may not improve the on screen display of all images.

Image Preferences (SMS Only)

When pyramids are built for an image, up to four JPEG images are saved to disk. These image files can be saved to a temporary folder or can be saved in the same directory as the original image so that they are not regenerated each time the image is loaded. The user’s preferences for building pyramids can be set in the *Images* tab of the *Preferences* dialog (*Edit* | **Preferences**).

Related Topics

- Import from Web
- Registering an Image

Import from Web

Overview

The *Import from Web* feature allows you to connect to the internet to download free data – images, elevation data etc. If you have an internet connection, this is an easy and convenient way to acquire this type of data.

The data is made available for free by various entities who provide web services [1]. Each of the XMS programs has a number of available data types they can retrieve.

It should be noted that the *Import from Web* feature links to external internet sites which can change without warning. For example, historically the XMS programs retrieved data from the *Terraserver* site which was terminated. The termination or modification of an online source may result in invalid links in the XMS program until the links can be corrected.

The *Import from Web* command is accessed in a number of ways. These include:

- From the "File" menu (WMS) or "Web" menu (SMS).
- From the "Get Data From Map" macro (WMS only).
- From the "Get Data" tool (SMS and WMS).

In the first two options the XMS program brings up a map locator tool (Virtual Earth) that allows the user to select (via pan and zoom) an area of interest and download data for this area. (As shown below)

The "Get Data" tool is available from the data toolbar when the XMS application is using a global projection. When this tool active in SMS or WMS, the user can graphically select a rectangle in the graphics window and download data inside this rectangle.

The XMS programs also have a **Get Online Maps** tool which can be used to get dynamic raster data, such as image or raster elevation data. The dynamic map is updated automatically when zooming in or out in the graphics window.

Any instance of a dynamic map on your screen can be downloaded by right-clicking on the map and selecting the **Export** command. This command will download the map to your computer.

Note: The *Import from Web* feature is no longer used for GMS as of GMS 9.0. The feature is still used in GMS 8.3 and earlier. However, since these tools used the now defunct "TerraServer" services, they are no longer referenced here.

SMS	WMS
• World Imagery [4] More Info [5] • World Street Maps [6] More Info [7] • World Topo Maps [8] More Info [9] • MapQuest OpenStreetMap Worldwide Street Maps [10] • OpenSTreetMap.org (Global Street Maps) • Other data sources (use the advanced button)	• NED data – USGS [1] • ASTER and SRTM data – USGS & NASA [2] • NLCD and CORINE (European) Land Cover data [3] • World Imagery [4] More Info [5] • World Street Maps [6] More Info [7] • World Topo Maps [8] More Info [9] • USA Topo Maps [11] More Info [12] • MapQuest OpenStreetMap Worldwide Street Maps [10] • USA Flood Hazard Zones [2] • Land Use Shapefiles [3] • STATSGO and SSURGO Soil Type Shapefiles [4] • Harmonized World Soil Database v 1.1 [5] • Global Land Cover [6] • Other data sources (use the advanced button)

Data Availability

Elevation (NED, ASTER, and SRTM) Data

- NED data contains the best available raster elevation data of the conterminous United States, Alaska, Hawaii, and territorial islands. NED data are not available for other areas.
- ASTER and SRTM data are available for most of the earth's surface. The ASTER dataset is reliable and high-quality.

Imagery

Most of the imagery (World Imagery, Street Maps, Topo Maps, and OpenStreetMap.org data) are available for anywhere on the earth. Some imagery, such as US Topo Maps, are only available for areas of the United States. Besides downloading these images using the Import from Web command, these images can be read as online maps that change resolution dynamically depending on your location.

Land Cover Data

- The 100 m Resolution CORINE dataset (raster) is available for anywhere in Europe.
- The 30 m NLCD dataset (raster) is only available for the conterminous United States.
- The Land Use Shapefile dataset is available for the entire United States.
- The Global Land Cover dataset is available in 2 degree by 2 degree blocks for the entire world. The following steps were used to convert the Land Use data to a format that can be used for WMS hydrologic modeling:

1. Go to the European Space Agency site to download land use data [6].
2. Download the .zip file Globcover2009_V2.3_Global_.zip and unzip this file on your computer.
3. Open GLOBCOVER_L4_200901_200912_V2.3.tif in a GIS (such as ArcMap) and convert it to an ESRI raster file. Trim the raster as needed, then convert the raster to a shapefile.
4. Convert the file Globcover2009_Legend.xls to a *.dbf file and join this file with the shapefile values to get the land use names and IDs.

Soil Data

- SSURGO soil datasets are available for all available SSURGO survey areas in the United States (as of August 2013).
- A STATSGO soil dataset is available every state in the United States.
- Data from the Harmonized World Soil Database are available in 2 degree by 2 degree blocks for the entire world. The WMS developers used the following steps to convert the soil data to a format that can be used for WMS hydrologic modeling:

1. Download and install the Harmonized World Soil Database program to your computer from the Harmonized World Soil Database web site [7].
2. Launch the HWSD Viewer on your computer. The soil data will be copied to the folder c:\program files (x86)\HWSD_v<xxx>\Data where <xxx> is the version of the viewer you have downloaded. The program may also be installed in c:\program files\<...> if you are running a 32-bit version of Windows. The following files are contained in this folder:
    1. The HWSD Raster *.zip file.
    2. The HWSD DBF file.
    3. The HWSD_META DBF file.
3. Copy the files in the data folder to a writable location on your computer and unzip the HWSD Raster *.zip file.
4. Open the .bil file in ArcMap and convert the *.bil file to a shapefile using the IDs.
5. Join the HWSD DBF file with the IDs in the shapefile.
6. Join the attribute IDs with the HWSD_META DBF file. This gives you a shapefile with the soil IDs and various soil attributes that can be used for hydrologic modeling in WMS.

Additional Information

Note that more vector-based soil and land use datasets are available; Contact Aquaveo [8] if you are interested in adding data from a specific area to the list of available land use or soil data that can be downloaded. A comprehensive list of soil and land use data available for download is located here.

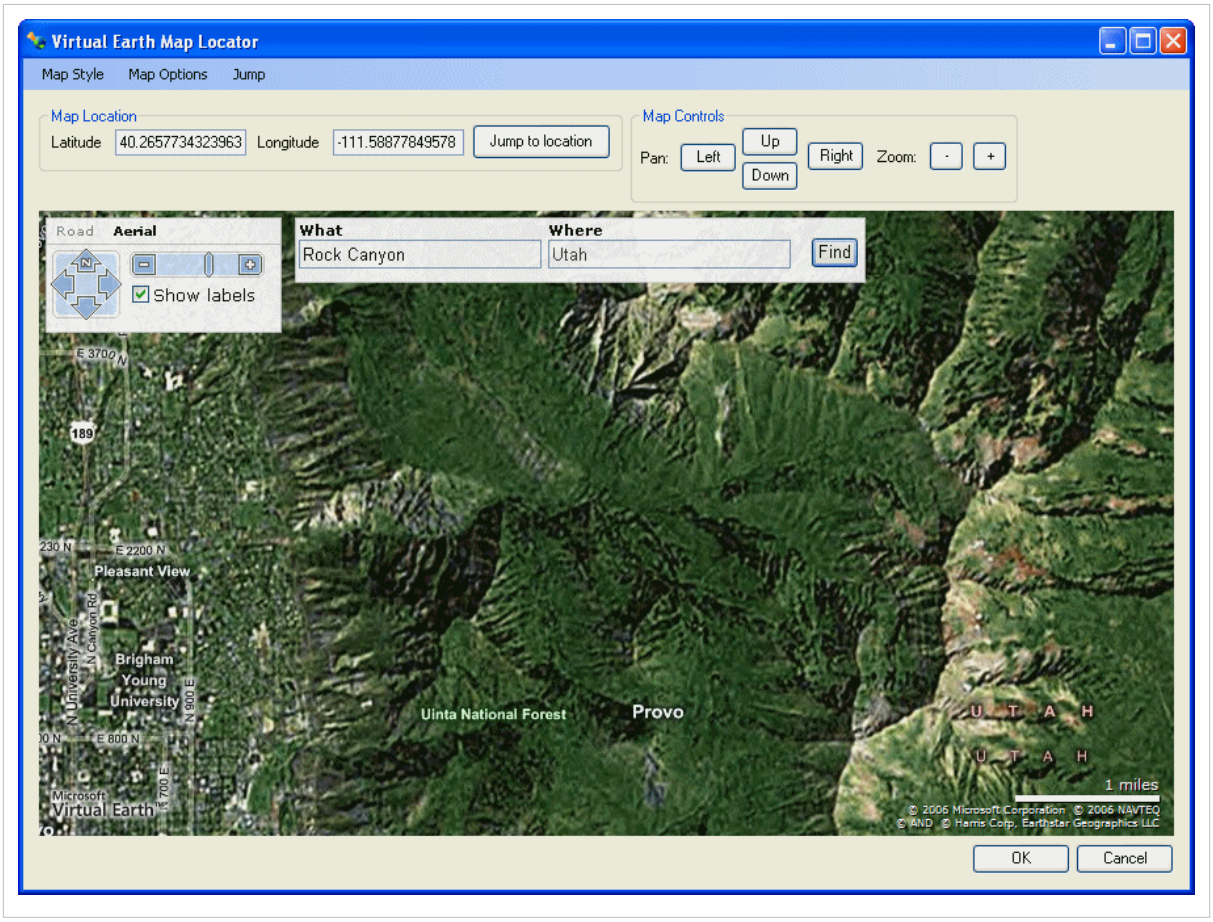
Terraserver images are no longer available because this web service has gone offline.

Using the Import from Web Command

When the *Import from web* command is invoked from a menu or macro, the virtual earth map locator is launched:

- **Virtual Earth Map Locator**:Use the map in this dialog to go to your location of interest.
    - You can **zoom** in or out using the controls or the mouse wheel.

- You can **pan** using the controls or by clicking and dragging. You can also enter the latitude and longitude to jump to a specific location.
- Use the *Map Options* menu to turn on the floating controls in the map (search, pan and zoom).
- Use the *Map Style* menu, or the floating controls, to change the map between **Road**, **Aerial**, and **Hybrid**.



Once the region for data download is defined, a series of dialogs appear which allow the user to define the data to be downloaded. These dialogs include:

1. **Data Service Options**

   Here you select which type of data you are interested in.

1. **Save**

   Now you are asked where you want to save the data. You only need to specify one file name, even if you've selected more than one type of data in the previous dialog. The files will all be given the same prefix but different suffixes.

1. **Confirm File Creation**

   You may be asked to confirm that you want to create the files.

1. **Initialize Connection**

   The following dialog is shown while the connection is being made.

1. **Select Scale**

   Smaller numbers (larger scales) will result in better resolution, but longer download times.

1. **Downloading**

   This dialog reports the download progress. If you click **Abort**, your image will exist but will be only that portion that you've downloaded so far.

Steps will repeat for each data type selected.

After everything is finished, the data (images etc.) will appear in the Project Explorer.

References

[1] http://en.wikipedia.org/wiki/Web_service
[2] https://hazards.fema.gov/femaportal/wps/portal/NFHLWMS
[3] http://www.webgis.com/lulc_shplatlong.html
[4] http://soildatamart.nrcs.usda.gov/
[5] http://www.iiasa.ac.at/Research/LUC/External-World-soil-database/HTML/
[6] http://ionia1.esrin.esa.int/
[7] http://webarchive.iiasa.ac.at/Research/LUC/External-World-soil-database/HTML/
[8] http://www.aquaveo.com/contact-us

Registering an Image

If an image file is not geo-referenced then the user must define the coordinate system of the image. The register dialog allows the user to specify the coordinate system for the image. When an image is opened, if the image is not self-referenced, XMS attempts to find world file with the same name as the image (*.wld or *.jpgw extension). If neither of these is found, the register dialog opens.

What is Image Registration?

Before an image can be displayed, the image must be "registered" or geo-referenced. Registering an image involves identifying points on the image corresponding to locations with known real world (XY) coordinates. Once these points are identified, they are used to scale and translate the image to the proper location when it is drawn with the other objects in the Graphics Window. If an image is not registered properly, any objects which are created using the background image as a guide will have the wrong coordinates.

Register Image Dialog

An image is registered using the *Register Image* dialog. The main feature of the *Register Image* dialog is a large window in which the image is displayed. Two or three points (shown by "+" symbols) are also displayed in the window. These points are used to identify locations with known real world coordinates. The real world coordinates (X,Y) and image coordinates (U,V) of the registration points are listed in edit fields below the image. The points are moved to the desired locations on the image by dragging the points using the tools described below. Once the points are located, the real world coordinates can be entered in the corresponding edit fields. The dialog contains the following options:

- **2 point or 3 point registration –** Two point registration rotates and uniformly scales an image. Three point registration allows for non-uniform scaling to account for some parallax.
- **Import World File –** Used to import a TIFF world file (*.tfw). A TIFF world file has the information needed to set the (X,Y) and (U,V) coordinates in order to place the image in the correct world coordinates.
- **Image name –** Used to associate a name with the file. This name will appear in the project explorer.

Register Image Dialog Tools

The following tools can be used to help position the registration points:

Tool	Tool Name	Description
	Select Point Tool	The Select Point tool is used to select and drag register points to a location on the map for which real coordinates are known so that they can be entered in the corresponding XY edit fields.
	Zoom Tool	In some cases, it is useful to magnify a portion of the image so that a registration point can be placed with more accuracy. The Zoom Tool is used to zoom in a portion of the image.
	Pan Tool	After zooming in on a portion of the image, the Pan Tool is used to pan the image vertically or horizontally.
	Frame Macro	The Frame Macro is used to automatically center the entire image within the drawing window of the dialog after panning and zooming in on a specific location.

Import World File

The **Import World File** button can be used to automatically define the registration data. A world file is a special file associated with a previously registered image that is exported from ArcView® [1] or Arc/Info® [2]. The file contains registration data that can be used to register the image.

Saving/Reading Image Registration Data

When a project file is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Convert Point Coordinate System

The x, y coordinates of each register point must be specified. If the user has the (x,y) coordinates in a different coordinate system than their project, the coordinates will need to be converted.

GMS Point Conversion

The **Convert Point** button in the image registration dialog will allow the user to convert the coordinates.

SMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

An alternative approach is to convert the coordinate system after importing by right clicking on the image in the Project Explorer and choosing ***Coordinate Conversion*** from the right click menu.

WMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

References

[1] http://www.esri.com/software/arcview/
[2] http://www.esri.com/software/arcgis/arcinfo/

Save as Image

It is now possible to save information displayed in the graphic windows in image format.

Saving in image format

To save information in the graphics window as an image, use the **Save As...** command in the *File* menu. Images can be saved in the following two formats:

- Bitmap Image Files (*.bmp)
- JPEG Image Files (*.jpg or *.jpeg)

It can be useful to save the contents displayed in the graphics window so the images can be used for presentation purposes, documents, etc.

Related Topics

- File Menu
- Save As...

Web Service for Background Imagery

SMS supports the ability to obtain image data from web servers. The imagery will update as you pan and zoom and use an appropriate resolution for your current zoom level. Since you are obtaining the information over the internet, the performance of these images will be dependent upon the speed of your internet connection.

System Requirements

SMS 11.0 and ArcGIS 9.3 or above are needed for this feature.

Projection

To use web layers, you must be in a non-local projection.

GIS Web Layers

SMS ships with several layers that you can use. Some of the layers are specific to the US and others are worldwide. You may want to experiment with different layers as some will give better performance or quality. To find the files shipped with SMS, go to the windows menu, go to the SMS 11.0 folder, and click on the item labeled "supporting files." This will open an explorer window to the folder that contains supporting files used with SMS. You will see a folder named "GIS Layer Files." This folder contains the GIS web layers that ship with SMS.

In order to load the GIS layers, you have to be using the ArcObjects interface inside of SMS. This is activated by switching to the GIS module and selecting the menu item, Data | Enable ArcObjects.

Zooming in to view more details

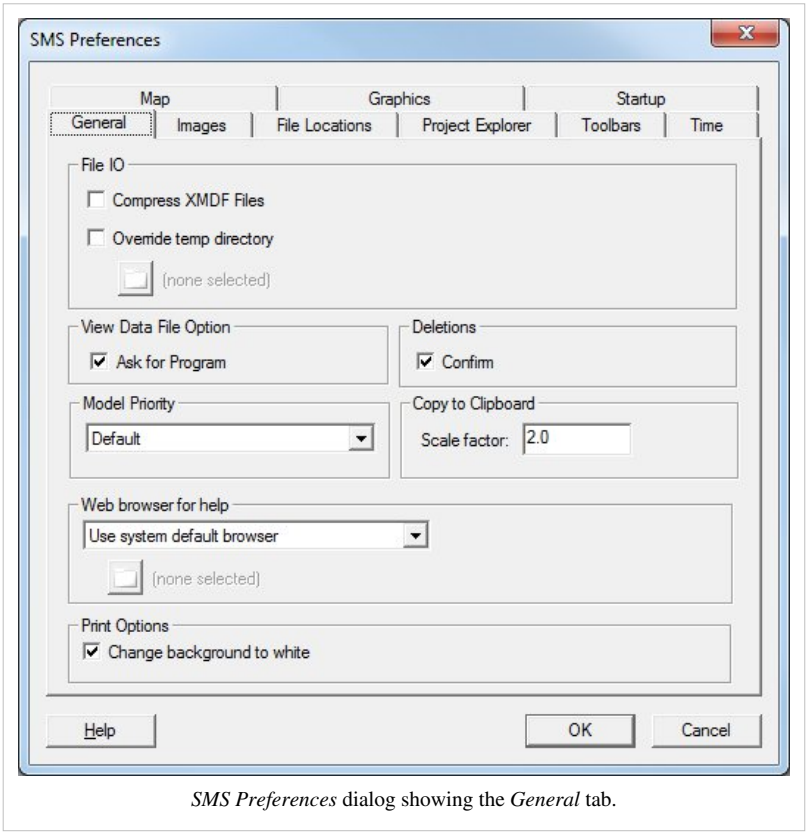
As you zoom in, more of the GIS layer features such as roads, peaks, etc will be more visible (as well as their labels in some cases depending on the GIS layer opened). The more you zoom in, the more details will appear. When you zoom out, the details will become less visible.

2.10. Preferences

Preferences

The *'Edit* | **Preferences** command brings up the *Preferences* dialog. The *Preferences* dialog contains the following tabs:

General



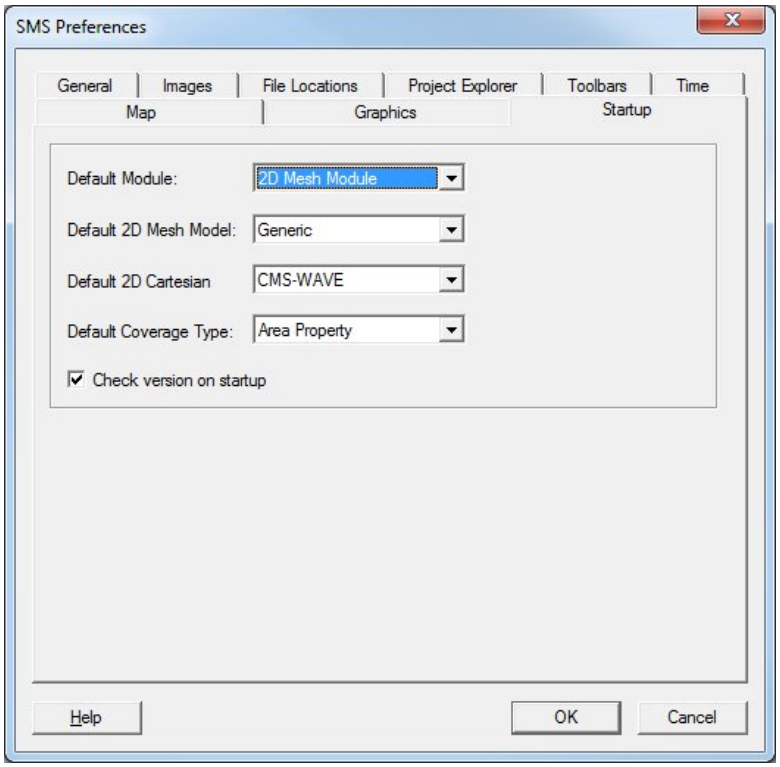
SMS Preferences dialog showing the *General* tab.

- **File IO**
  - XMDF File Compression – Use compression when saving XMDF files.
  - Override temp directory – Specify the location where SMS temporary files are written.
- **View Data File Option**
  - Ask for Program – SMS will prompt the user to specify the program used to open a data file when the *File* | *View Data File* command is called.
  - **Deletions**
    - Confirm Deletions – Whenever a set of selected objects is about to be deleted, the user can choose to be prompted to confirm the deletion. This is meant to prevent accidental deletion of objects.
  - **Model Priority** – Models can be launched using a particular process priority. This priority specifies how the operating system should treat the process. We recommend using the "Above Normal", "Normal", or "Below Normal" options in most circumstances. The options are as follows:
    - Realtime – Highest priority. May cause machine to become unresponsive. Use with extreme care.
    - High – Only allows realtime process to go before it. Can use nearly all CPU cycles. Use with extreme care.
    - Above Normal – Takes priority over normal processes. Will take CPU cycles before normal applications do.
    - Normal – No special scheduling takes place. This is the normal default.
    - Below Normal – Allows processes with normal priority to run first, but runs before low priority processes.
    - Low – The process will only run when the system is idle.

  - Default – The process will be launched with the same priority as it's parent (SMS in this case).
- **Copy to Clipboard**
  - Scale factor – When copying the contents of the main graphics window to clipboard, the resolution can be increase by specifying a scale factor greater than 1.0.

Startup

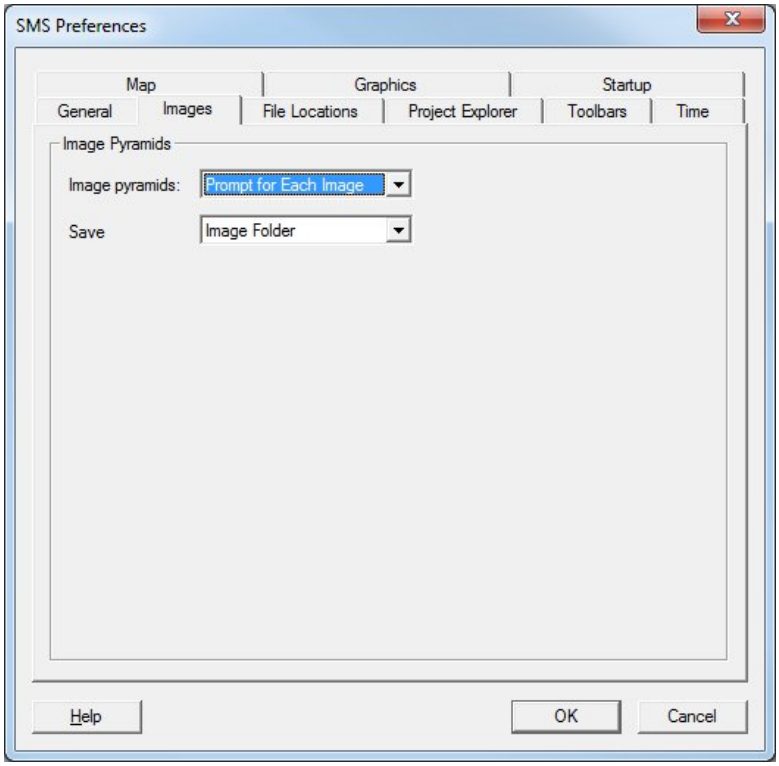
- Default Module – Specify the active module at startup.
- Default 2D Mesh Model – Specify the active 2D Mesh Module Model at startup.
- Default 2D Cartesian Grid Model – Specify the active 2D Cartesian Grid Module] Model at startup.
- Default Coverage Type – Specify the active Map Module Coverage Type at startup.
- Check version on startup
- Default TUFLOW Executable – Specify the executable to use by default when creating a new TUFLOW simulation. The options are double and single precision for both 32 and 64bit.



SMS Preferences dialog showing the *Startup* tab.

Images

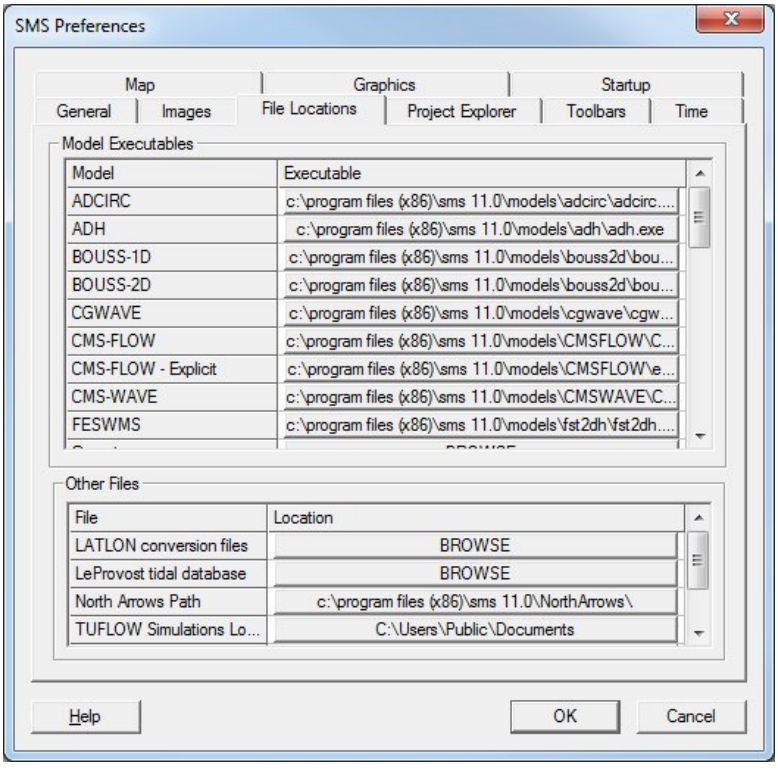
- Image Pyramids – Specify whether SMS will always, never, or prompt to build image pyramids.
- TIFF → JPEG Conversion – When a TIFF file is opened in SMS, the file is copied and saved as a JPEG file. You can specify where the JPEG copy of the image file is saved.



SMS Preferences dialog showing the *Images* tab.

File Locations

- Model Executables – Specify the location of numerical model executables.
- Other Files – Specify the location of files used by SMS (LeProvost tidal database, LATLON conversion files, etc.).

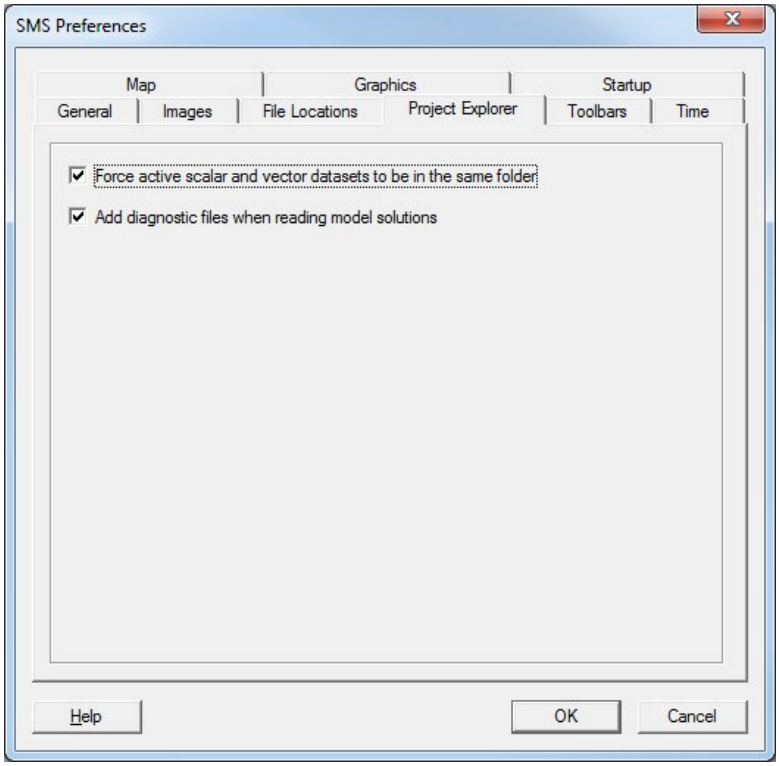


SMS Preferences dialog showing the *File Locations* tab.

Project Explorer

Options to:

- Force active scalar and vector datasets to be in the same folder
- Add diagnostic files when reading model solutions

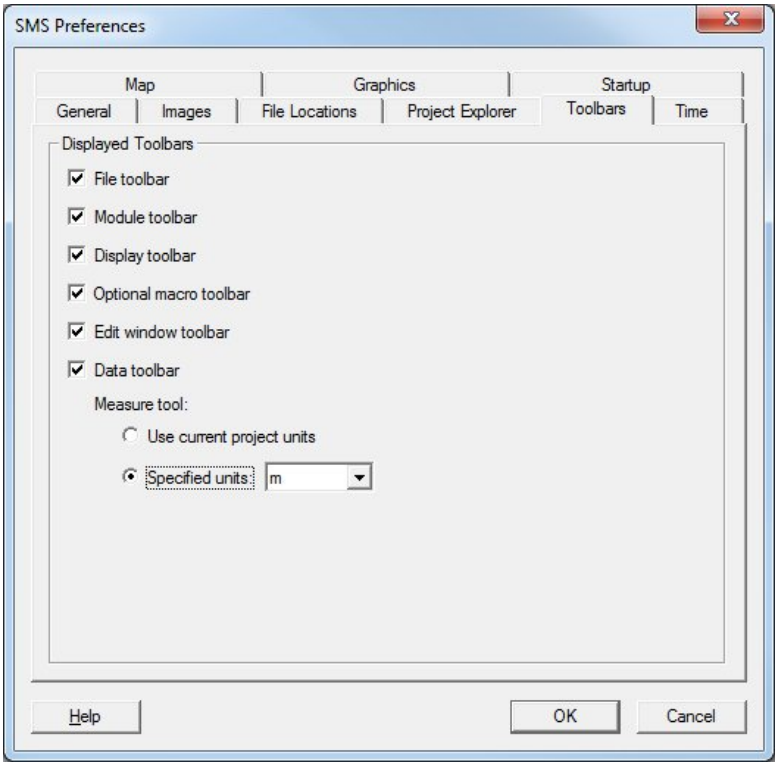


SMS Preferences dialog showing the *Project Explorer* tab.

Toolbars

Options to turn the following toolbars on or off:

- File Toolbar
- Module Toolbar
- Display Toolbar
- Optional Macro



SMS Preferences dialog showing the *Toolbars* tab.

Time

See the Time Step Window article for an explanation of absolute and relative time. The default format of the time steps in the Time Step Window can be set.

Available times options

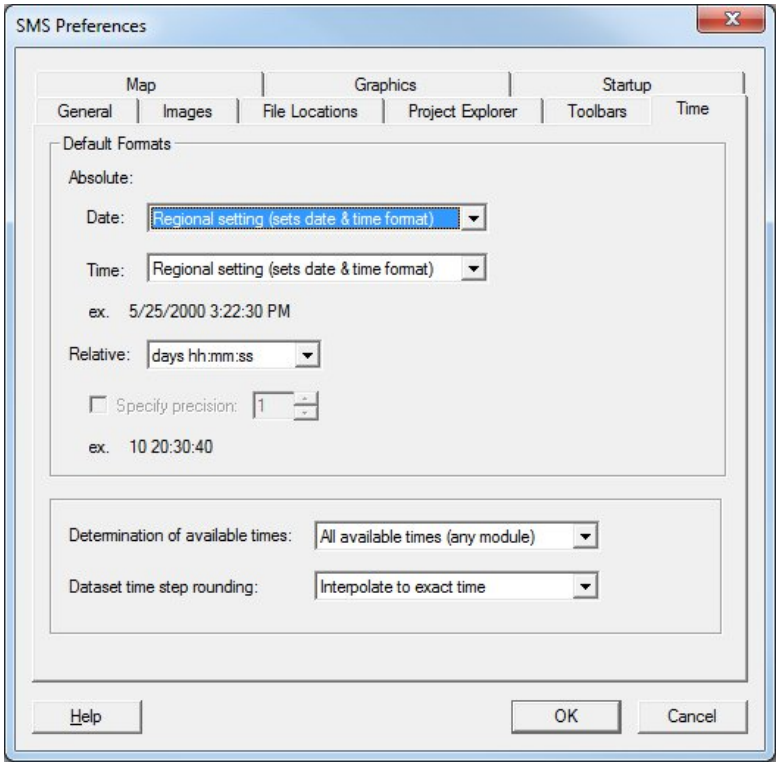
This option controls which times are displayed in the time step window. The available options are:

- Active datasets (current module only) – The times displayed in the timestep window are based only upon the active scalar and vector datasets in the current module. If neither of these datasets is transient, the time step window will not be displayed.
- All available times (all modules) – The times displayed in the timestep window are based upon times used by any transient object in SMS (includes datasets, some kinds of coverages, and PTM particle sets). All of the times from each of the objects will be used regardless of whether or not the object is active or visible.

Dataset time step rounding

The dataset being used for contours, vectors, or other display option may not have a timestep that corresponds exactly with the time currently chosen in the time step window. When this happens, SMS has two options for determining the values used by the dataset. These options are:

- Interpolate to exact time – Interpolate the dataset values for the selected time step from the nearest time steps before or after the display time. If the display time is before/after all of the time steps the nearest time step is used.
- Use nearest time – The dataset time step nearest the display time will be used (no interpolation).

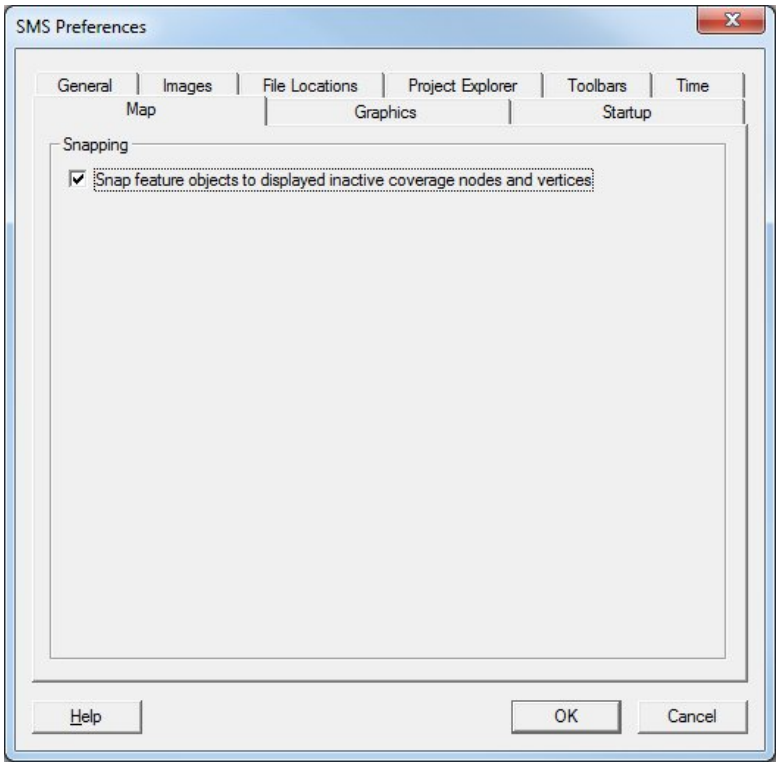


SMS Preferences dialog showing the *Time* tab.

Map

Options to:

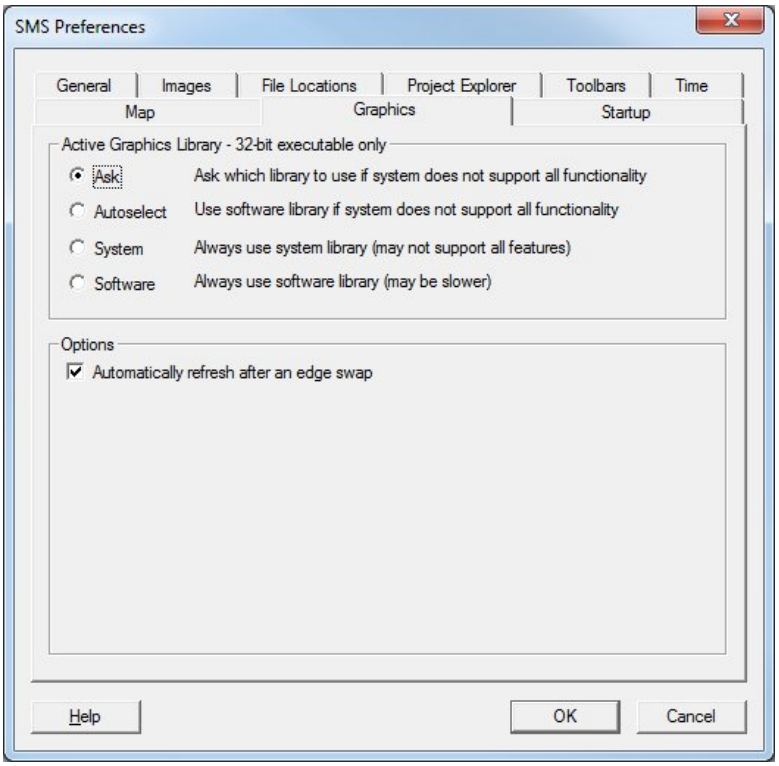
- Snap feature objects to displayed inactive coverage nodes and vertices when creating new feature objects.



SMS Preferences dialog showing the *Map* tab.

Graphics

- **Active Graphics Library**
- **Options**
- Automatically refresh after an edge swap
- Use vertex buffer objects (VBOs) – This option can be specified to change how SMS works with the graphics card. Vertex buffer objects are generally faster and often uses less of the computers main memory. However, there are circumstances where using vertex buffer objects can be significantly slower, such as when the machine's graphics card only has a small amount of onboard memory. By default, this option is on.



SMS Preferences dialog showing the *Graphics* tab.

Related Topics

- Edit Menu
- Time Settings

Time Settings

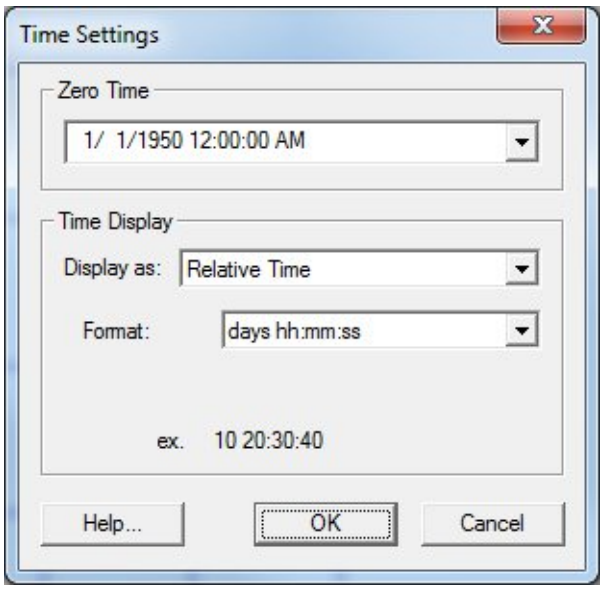
Transient dataset time values are displayed in the *Time Step Window* using either a relative time format (e.g., 100.0) or an absolute date/time format (e.g., 1/12/2006 3:23:48).

Zero Time

The zero time represents the date/time corresponding to time t=0. If a dataset does not have an assigned reference time, it will use the global zero time as its reference time.

Absolute Date/Time

When the display format is set to *Absolute Date/Time*, a date/time is shown in the *Time Step Window*. The date and time format can also be specified.



Time Settings dialog

Relative Time

When the display format is set to *Relative Time*, the days, hours, minutes, and seconds from the dataset reference time is shown in the *Time Step Window*. The display format for days, hours, minutes, and seconds can be specified. If a decimal format is chosen, the precision can also be specified.

Changing Time Settings

To change the time settings, select the *Menu* command Edit | Time Settings or right-click on the *time step window* in the *Project Explorer* and select *Time Settings* to open the *Time Setting* dialog.

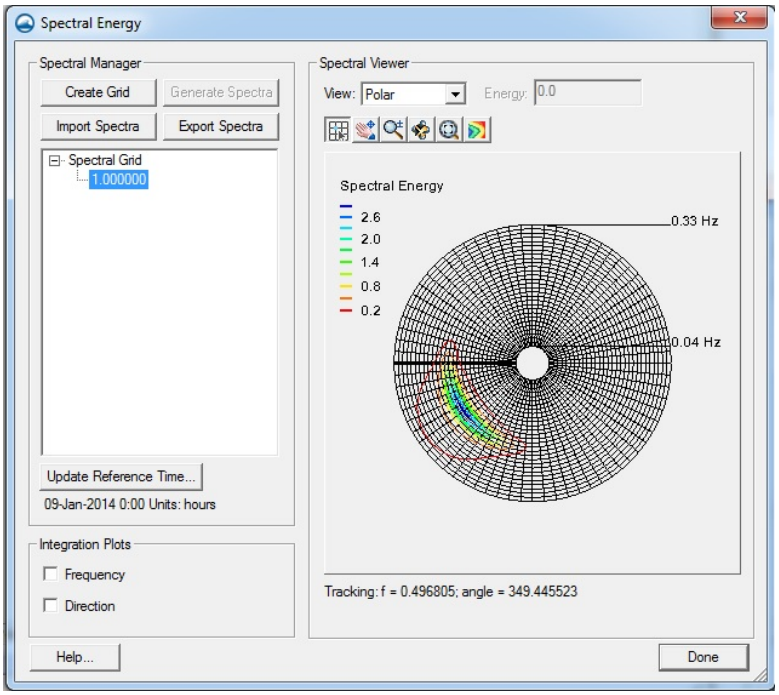
Related Topics

- Layout of the Graphical Interface

2.11. Spectral Energy

Spectral Energy

The *Spectral Energy* dialog can be accessed when the spectral coverage is active. The dialog is opened by either double-clicking on a selected node or by right-clicking on a selected node and choosing the **Node Attributes** command. The spectrum represents energy densities at discrete values over a range of angles and a range of frequencies for a given wave condition.



Spectral Energy dialog

Tools

- **Select Cell Corner –** Select a cell corner to view or edit the energy value.
- **Pan –** Pan the spectral grid.
- **Zoom –** Zoom in the window.
- **Rotate –** Rotate in the window.
- **Frame –** Frame, or zoom to the extents of the grid.
- **Contour Options –** Bring up the contour options dialog for setting the spectral grid contour display options.

Grid Options

- **Create Grid –** Brings up the Create Spectral Energy Grid dialog.
- **Generate Spectra –** Brings up the Generate Spectra dialog.
- **Import Spectra –** Brings up the Import Spectra dialog.
- **Export Spectra –** Brings up the *Export Spectra* dialog where you can choose a location to save the spectra.

Spectral Tree Options

- **Spectral Manager Tree –** Select a spectrum in the tree. The selected spectra will be displayed in the *Spectral Viewer*. The tree's right-click options are described below.

Grid/Spectra Right-click Options

- **Generate Spectra –** Opens the Generate Spectra dialog. This option is available when the grid in the *Spectral Manager Tree* is right clicked.
- **Edit Spectra –** Opens the Edit Spectra dialog where the *Parameter Settings*, *Angle Settings* and *Spectral Parameters* can be edited.
- **Export Spectra –** Brings up the *Export Spectra* dialog where you can choose a location to save the spectra.
- **Delete Grid** or **Delete Dataset –** Deletes a grid or spectrum.
- **Properties –** Brings up the Spectral Grid Properties dialog. This option is available when the grid in the *Spectral Manager Tree* is right-clicked.
- **Edit Time –** Edit the time offset for the dataset.

View Options

- **Cartesian/Polar View –** View and edit the spectral grid using a Cartesian or polar view.

Graphic Options

- **Selection –** View the Frequency, Angle, and Energy of the selected cell corner. The Energy can be edited for the selected points.
- **Cursor –** View the Frequency, Angle, and Energy as the cursor moves over the grid.

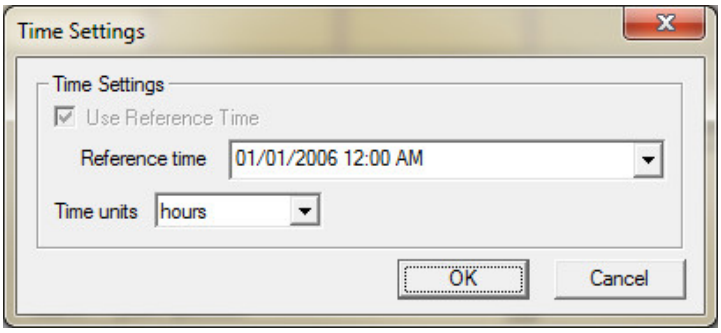
2D Plot Options

- **Frequency Integration Plot –** Turn on to show an energy vs. frequency plot for the selected spectrum.
- **Direction Integration Plot –** Turn on to show a direction vs. energy plot for the selected spectrum.

Reference Time

Clicking on the **Update Reference Time...** button will bring up the *Time Settings* dialog with the following options.

- **Reference Time –** Specify the reference time for all datasets assigned to the node
- **Units –** Specify the units for the time offsets assigned to the datasets



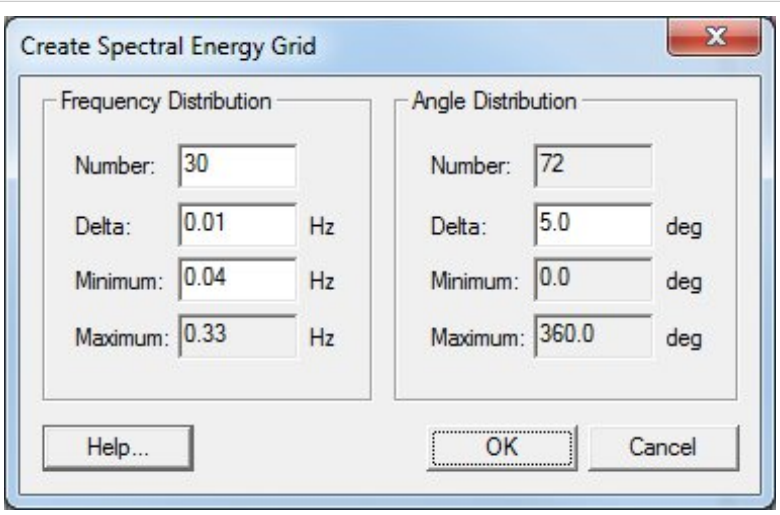
The spectral energy *Time Settings* dialog.

Related Topics

- Create Spectral Energy Grid
- Generate Spectra
- Import Spectra
- Spectral Grid Properties
- STWAVE Menu
- CGWAVE Menu
- Cartesian Grid Module

Create Spectral Energy Grid

The *Create Spectral Energy Grid* dialog is accessed through the *Spectral Energy* dialog (by pushing the **Create Grid** button). A spectral energy grid is created after setting the options. All units for the options are hertz and degrees. The new spectral energy grid will be displayed in the *Spectral Manager* and the *Spectral Viewer* in the *Spectral Energy* dialog.



Create Spectral Energy Grid dialog

Plane Type and Angle

- **Plane type (full global, full local, half local) –** set the plane type for the spectral data
- **Angle –** set the grid orientation

Frequency Distribution

- **Number –** Set the number of frequency bands (Number = 30)
- **Delta –** Set the step size (Delta = 0.01) in Hz.
- **Minimum –** Set the minimum frequency (Minimum = 0.04) in Hz.
- **Maximum –** View the maximum frequency (Maximum = 0.33) in Hz.

Angle Distribution

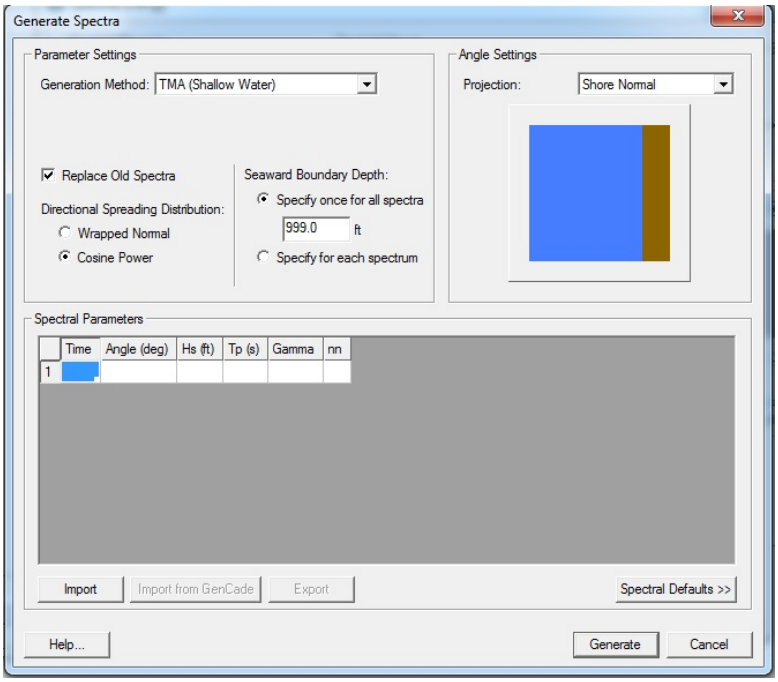
- **Number –** View the number of angle bands (Number = 35).
- **Delta –** Set the step size (Delta = 5) in degrees.
- **Minimum –** View the minimum angle (Minimum = 0.0) in degrees.
- **Maximum –** View the maximum angle (Maximum = 360.0) in degrees.

Related Topics

- Spectral Energy

Generate/Edit Spectra

This page describes both the *Generate Spectra* and *Edit Spectra* dialogs since they are almost the same dialog. The *Generate Spectra* dialog is accessed through the *Spectral Energy* dialog (by pushing the **Generate Spectra** button). The *Edit Spectra* dialog is accessed through the *Spectral Energy* dialog by right-clicking on a grid or spectra in the *Spectral Manager* and clicking **Edit Spectra**. SMS creates/edits the spectra when the **Generate/Edit** button is clicked. All units for the options are feet or meters, depending on the coordinate system. STWAVE runs in metric units, so if the current units are in English units inside SMS, all data is converted to metric when it is saved. The settings are shown below (default values are shown in parenthesis):



Generate Spectra dialog

Parameter Settings

- **Generation Method –** Choose the method you want to use to generate the spectra. Each method has slightly different options.
- **Replace Old Spectra –** Delete all existing spectra in *Spectral Energy* dialog after the new spectra are generated. Not available when editing.
- **Directional Spreading Distribution –** Choose to use either the wrapped normal distribution or the cosine power distribution. With the wrapped normal option, a standard deviation and maximum angle cutoff must be specified. With the cosine power option, the spreading index and the maximum cutoff angle must be specified. The recommended cutoff angle is three times the standard deviation of the directional distribution.
- **Gauge Depth –** Water depth in meters (d = 5.0). Choose whether to specify once for all spectra or to specify for each spectrum.

Angle Settings

- **Coordinate System –** The wave direction can either be specified in a shore normal/local coordinate system or a meteorologic/global coordinate system.

Spectral Parameters

The following are used by SpecGen to generate the spectra.

- **Time (hrs) –** Time offset using the specified units for the spectral node.
- **Angle (deg) –** Approach angle relative to the shore normal in the clockwise direction measured in degrees (wvang = 25.0).
- **Hs (m) –** The incident zero moment wave height (hm0 = 1.0).
- **Tp (sec) –** Wave period in seconds (tp = 20.0).
- **Gamma –** Spectral peak dispersion factor (igamma = 3.3).
- **nn –** Directional or peak dispersion factor, must be even integer (inn = 4).
- **Gauge Depth (m) –** Water depth in meters (d = 5.0). This parameter is available if the *Gauge Depth* option above is set to Specify for each spectrum.

Spreadsheet Options

- **Import/Export –** Import/export an ASCII, space delimited text file with the spectral generator parameters. The file format is:

SPECTRAL TABLE	Values
Method Option Time Index Angle Hs(1) Tp(1) Gamma (1) Hs (1) Tp(1) Gamma(1) Hs(2) Tp(2) Gamma(2) Wind Fetch nn StdDev Depth	Headers
0 -1 999.0 None 25.0 1.0 20.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	1st row of values
0 -1 999.0 None 30.0 1.0 16.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	2nd row of values

Additional format description is found in CMS-Wave Spectral Table File.

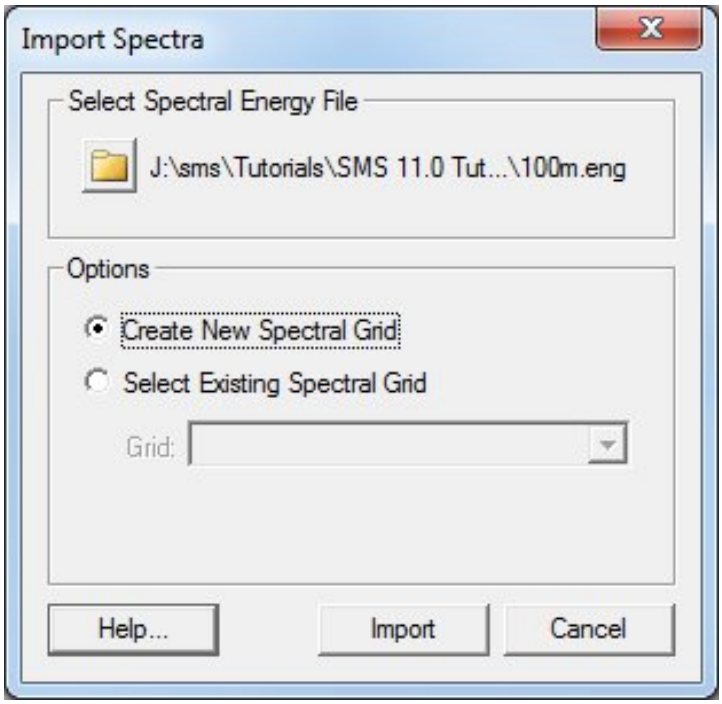
- **Import from GenCade –** Imports wave parameters from the *Filtered Ocean Conditions* dialog. This button is only available if data has been filtered in GenCade.
- **Spectral Defaults –** This opens a section of the dialog that lists the approximate spectral parameters. Double-click a row of values to replace the selected spreadsheet row(s). The period (T), gamma, and nn will be replaced for the row. If the period in the spreadsheet does not match a period in the table, the spreadsheet period is rounded to the nearest table period.

Related Topics

- Spectral Energy

Import Spectra

The *Import Spectra* dialog is accessed through the *Spectral Energy* dialog by clicking on the **Import Spectra** button. This dialog is used to import existing spectra into the project.



Import Spectra dialog

File types

The following types of files can be imported through this dialog.

- ***.eng –** Spectral energy file. You will be prompted to specify if the file is in the CMS-Wave or STWAVE format.
- ***.dws –**- The BOUSS-2D spectra file format.

Obsolete Options

The following options are no longer available as of SMS version 11.2.

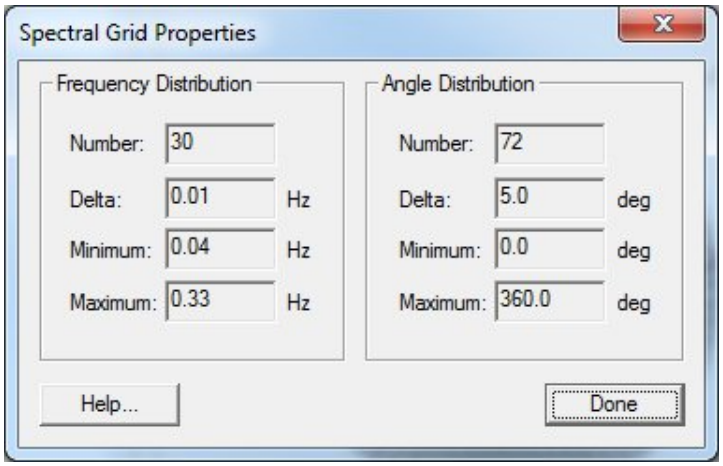
- **Select Spectral Energy File –** Click on the folder icon to browse to the spectral energy file you want to import.
- **Create New Spectral Grid –** If this option is selected, SMS will import the spectra with the grid definition contained in the spectral energy file.
- **Select Existing Spectral Grid –** If this option is selected the datasets are imported as datasets of the selected grid (the grid selected in the combo box below).
- **Import as Time Steps –** Imports the datasets using the identifier as a time value.

Related Topics

- Spectral Energy Dialog

Spectral Grid Properties

The *Spectral Grid Properties* dialog is accessed through the *Spectral Energy* dialog by right-clicking on a grid in the Spectral Manager tree and clicking on **Properties**. This dialog only displays information. The values cannot be changed. If different values are desired, a new grid must be created through the *Create Spectral Energy Grid* dialog.



Spectral Grid Property dialog

Frequency Distribution

- **Number –** View the number of frequency bands.
- **Delta –** View the step size in Hz.
- **Minimum –** View the minimum frequency in Hz.
- **Maximum –** View the maximum in Hz.

Angle Distribution

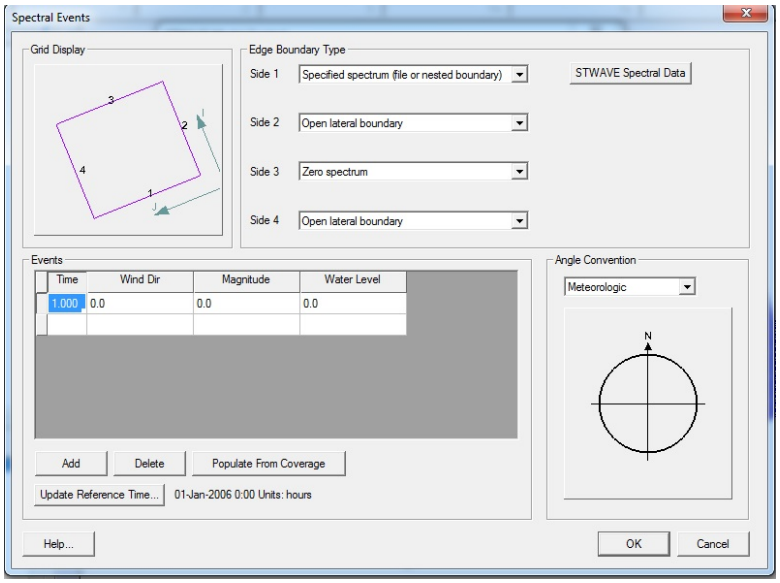
- **Number –** View the number of angle bands.
- **Delta –** View the step size in degrees.
- **Minimum –** View the minimum angle in degrees.
- **Maximum –** View the maximum angle in degrees.

Related Topics

- Spectral Energy

Spectral Events

The external boundary condition for STWAVE and CMS-Wave consists of one or more energy spectra entering on one or more open edges of the grid. The *Spectral Events* dialog specifies the boundary conditions and locations of these boundary conditions. This dialog is reached through the **Boundary Control...** button in the *STWAVE Model Control* dialog or through the **Define Cases...** button in the *CSM-WAVE Model Control* dialog.



Spectral Events dialog

Grid Display

This graphic shows the orientation of the grid and labels the sides of the grids which is used in other controls within the dialog.

Edge Boundary Type

The type of boundary condition applied to each edge of the STWAVE grid is shown and in some cases edited in this section of the dialog.

The types of boundary conditions include:

- Specified spectrum – This may come from a parent grid if using nesting. Otherwise, a button to the right will be used to assign the spectral coverage to the boundary.
- 1D transformed spectrum – This boundary type allows energy to propagate along the boundary without interference. The cells would have the same energy if the grid was extended and the boundary became interior to a larger grid.
- Zero spectrum – The boundary doesn't have any spectral energy applied.

For half-plane models, the boundary condition types are fixed and energy travels in the positive I direction of the grid. In this case, the user specifies spectra that will be introduced at side 1. Sides 2 and 4 will be treated as 1D transformed spectra.

For full-plane models, the boundary condition types may be specified. You cannot have specified spectra on two adjacent boundaries. Therefore, you can have specified spectra on a maximum of two boundaries and these must be on opposite boundaries.

Events Spreadsheet

This spreadsheet defines the time step or cases that will be used and the input boundary conditions for each.

The first column is the time offset value. This number represents how much later the time is than the specified reference time. Hence, if 5 is entered for the time offset, and the time units is hours, then it is the case of 5 hours later. When nesting is used, the case ids, and the number of cases, will be determined by the parent simulation.

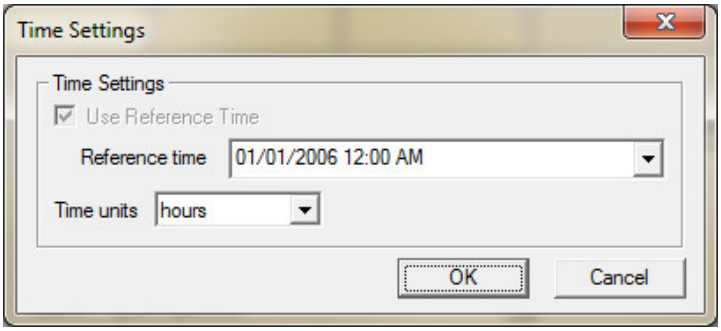
In addition to the external condition, the engine can simulate distributed forces over the domain including wind, surge and currents. The currents applied to a simulation are specified in the *Model Control* dialog. Wind and surge values are specified in the spread sheet in columns 2 through 4.

The spectra to be used will be matched up or interpolated with the model timesteps. The button **Populate From Coverage** can be used to generate events for each time found in the spectral coverage(s) that have been assigned to the model.

The number and use of the remaining columns in the spreadsheet will depend upon the options used for the STWAVE simulation. For example, if a constant value is used for wind and/or surge columns will appear that represent the wind direction, magnitude and/or tidal elevation as applicable.

Reference Time

The reference time controls allow you to set a reference time and the units to be used when defining the time for each case. Clicking on the **Update Reference Time** button will bring up a *Time Settings* dialog. In this dialog the "Reference Time" and "Time units" can be set.



The spectral energy *Time Settings* dialog.

Angle Convention

The angle convention controls allow you to choose the convention that will be used for the wind direction field in the events spreadsheet. The direction represented by the wind angle of the active row of the spreadsheet is plotted on the direction graph.

Related Topics

- Spectral Coverage

2.12. Datasets(VTK)

Datasets VTK

Datasets

A dataset is a set of values associated with each node or cell in a geometric object. Datasets:

- Can be scalar (1 value) per object or vector (2+ values per object).
- Can be steady state (constant through time) or transient (values change at specified times).
- Can have active information to specify that specific nodes or cells are inactive in a model (generally used by solutions to indicate dry areas). Activity can be represented by NULL values in the dataset or as separate on/off values for nodes or cells.
- Control how contours, vectors, and functional surfaces are displayed.

Generating Datasets

Datasets can be generated in a variety of ways such as:

- Output from a numeric engine (water level, velocity, concentration, transport, etc.)
- Tabular values in a text file entered by the user or exported from another application such as a GIS
- Created by interpolating from a scatter point set to a grid, or mesh
- Generated by performing mathematical operations on existing datasets with the Dataset Calculator

Project Explorer

Datasets are displayed and managed in the Project Explorer. See the Project Explorer article for more information.

Active Dataset

Each geometric object has a dataset that is termed the "active dataset." The active dataset is used for contour and vector display and may be used for functional surfaces, 2D plots, or other functionalities keying off this dataset.

Project Explorer Icons

Different icons are used to represent datasets in the project explorer including the active/inactive state of the dataset. These icons are below:

Dataset Type	Inactive Icon	Active Icon
Elevation		
Scalar		
Vector		

Time Information

Transient datasets (those that change with time) have information and functionality that is not available for steady state datasets. A time step represents a specific time and its values in a transient dataset. A dataset may use absolute times meaning the dataset has full date/time information. Alternatively, a dataset can use relatives times which means they know how much time has elapsed since some non-specified zero time (generally the beginning of the simulation). When transient datasets are present, the Time step window may be present depending upon the current time settings as specified in the preferences dialog.

Folder

The datasets and solutions are organized by folders. The user can create new folders and move datasets, solutions, and folders to other folders anywhere on the *Project Explorer*. Folders can be created by right-clicking on the certain items in the *Project Explorer* and selecting *New Folder* in the menu. A dataset or folder can be deleted simply by selecting the folder and selecting the Delete key or by right-clicking on the item and selecting the Delete option in the corresponding pop-up menu.

Datasets on VTK Objects

Datasets as used on VTK objects have different functionalities then those used in the original geometric representations in SMS. Some of the differences for VTK datasets include:

- Each dataset can be mapped either to nodes or cells and the same geometric object can have both types of datasets at the same.
- A new dataset calculator has been created. The new calculator has additional functionalities and options. For more information see the Dataset Calculator VTK topic.

Related Topics

- Layout of the Graphical Interface

Conversions Scalar↔Vector

Conversions Scalar↔Vector

Datasets can be converted from scalar datasets to vector dataset or converted from vector datasets to scalar datasets.

Scalar to Vector

Converts two scalar datasets to a single vector dataset. The specified scalar datasets can be either magnitude and direction or x and y components.

To convert to vector do the following:

1. In the *project explorer*, select two scalar datasets. Click on the first dataset, press and hold down the *CTRL* key, then click the second dataset.
2. Right-click on the selected datasets and select **Scalars to Vector**.
3. Select whether the datasets are magnitude and direction or x and y components.
4. Give the new vector dataset a name in the *Dataset Name* field.
5. Click **Ok**.

Vector to Scalar

Converts a single vector dataset into one or more scalar datasets. The resulting scalar datasets can be magnitude, direction, x or y components.

To convert to scalar do the following:

1. In the *project explorer*, right-click on a vector dataset and select **Vector to Scalars**.
2. Right-click on the selected datasets and select **Scalars to Vector**.
3. Place checkbox's next to the datasets that you want to create.
4. Give the new scalar datasets a prefix in the *Prefix* field.
5. Click **Ok**.

Dataset Calculator VTK

The dataset calculator allows you to create new datasets using mathematical expressions. The inputs for the mathematical expressions can be user defined constants, existing datasets, data derived from the geometry, or data derived from datasets such as gradients or activity information (0/1).

The dataset calculator expressions can include:

- standard operations: + - * / ^ .
- build unit vectors: iHat, jHat, kHat (ie (1,0,0), (0,1,0), (0,0,1))
- abs
- acos
- asin
- atan
- ceil
- cos
- cosh
- exp
- floor
- log
- mag
- min
- max
- norm
- sign
- sin
- sinh
- sqrt
- tan
- tanh

Expressions

In the top left of the dialog, there are fields for name and expression. The name specified will become the dataset name. The expression defines the mathematical operation that will take place. You can insert a predefined function into the expression field using the "Insert f(x)" combo-box. The specified function will be pasted into the current cursor location in the expression field and will include an indication of the number of values expected.

SMS maintains a list of expressions currently defined. Specifying a name and expression and clicking "Add" will add an expression to the list of expressions. If you select an expression from the list of expressions, the name and expression fields will be populated. You can make the current list of expressions the default list by saving settings ("File | Save settings" from the main menu). You can move your expressions to another computer or user by using the save and load buttons.

Variables

When the expression field is modified, SMS will parse the expression to find variable names in the expression. A variable must start with a character but can include digits after the first character. "Var1" would be valid but "1Var" would not. The list of variables determined by SMS will appear in a spreadsheet in the top right of the dialog. Each variable has a name, type, and source information. The name is parsed from the expression field and cannot be edited in the spreadsheet. The type can be chosen from the spreadsheet. The source information defines the options used which vary by variable type and cannot be edited in the spreadsheet. Beneath the variables spreadsheet are controls to specify the source information for the currently selected variable. The controls that will be available will depend upon the variable type.

The simplest variable type is a value. While it is possible to include numeric values in the expression field, it can be more clear to name specific variables. For example, you may want to have an expression use a variable named "gravity" that is specified as a value variable. This makes the expression more readable and helps remind people to change this if working in a different set of units or similar situation.

Another type of variable is a dataset. This will use the specified dataset anywhere the variable shows up in the expression. Note that both scalar and vector datasets can be used in an expression.

The final type of variable are "Derived" variables. Derived variables can be based upon one of three different sources. They can be based upon node geometry, cell geometry, or datasets. Derived nodal options include location (3D vector) and nodal spacing (average spacing to neighboring nodes). Derived cell options include area, centroid (3D vector), extents minimum (3D vector), extents maximum (3D vector), and perimeter. Derived dataset values include activity (0/1 for each node/cell for each timestep), directional derivative (vector representation of the gradient), and vector angle (in the cartesian coordinate system used in math).

Output Location

Datasets in VTK can be associated with nodes or cells. When using the dataset calculator, you can specify whether you want the dataset created to be associated with nodes or cells. If you don't specify the output location, SMS will decide based upon the variables used in your expression. With the exception of values, each variable will have an affinity to nodes or cells. If any variable has affinity to nodes, SMS will default the output to be based upon nodes. If all the input variables have an affinity to cells, the output will be a cell based dataset. The variable affinity is generally intuitive: dataset variables use dataset, derived variables come from whether they are node, cell, or dataset derived. One exception to the intuitive rule is "directional derivative" which is derived from datasets. This variable will have an affinity opposite of the input dataset. If the input dataset is nodes, the directional derivative will be computed on cells.

Whenever input data has an affinity opposite to the output location, the data is converted before the expression is evaluated. Generally, this means that the value for each node or cell (whichever conversion is taking place) is determined by averaging the surrounding values (connected cells or nodes belong to the cell). The exception to this rule is activity which is never averaged. A node is considered active if any of the surrounding cells is active.

Working with Scalars/Vectors

As mentioned previously, an expression may contain both scalar and vector components. The result of an expression may also be either a scalar or vector dataset. Some sub-expressions can be used with either type of dataset. For example, you can multiply a value by either a scalar or vector dataset. Each component in the vector dataset will have the multiplication applied. Other sub-expressions only make sense when dealing with a certain type of input dataset. The "." operator can be used to compute the dot product of two vectors and doesn't make sense for scalar datasets. The identifier iHat can be used to create a unit vector in the x direction (1.0, 0.0, 0.0). You can use the dot operator to extract the x component of a vector using an expression like: "myvector.iHat". jHat and kHat can be used

similarly to extract y and z components. You can also convert components into vectors using statements similar to: "vx*iHat + vy*jHat." The mag function extracts the magnitude of a vector and also does not apply to scalar datasets.

Output Times

By default the output dataset will have times corresponding to any time used in any of the input datasets. If all the input datasets are steady state datasets, the output dataset will be steady state. You can specify specific output times for the output dataset. This can be useful to reduce the number of timesteps that would be generated or focus on a range of times. If all the input datasets are steady state and output times are specified, the resulting dataset will have multiple timesteps each with the exact same values. If an input dataset doesn't have a timestep at an output time, the data will be interpolated between the nearest timesteps. If the time occurs outside the range covered by the dataset, the nearest dataset time will be used.

Errors

If there is a problem with the expression entered, SMS will give an error message and allow you to correct the problem. If the problem is with the expression itself, the message should indicate about where the problem exists. The position may be off slightly so examine the whole expression carefully if the cause isn't immediately apparent.

Interpolation VTK

VTK Dataset Interpolation

VTK datasets can be interpolated to create functions or datasets on mesh2d, cgrid, scatter, VTK mesh or curvilinear geometric objects. The interpolation is invoked in the project explore by right-clicking on the VTK mesh or Curvilinear geometric object (from which the source datasets will be obtained) and then selecting the *Interpolate to...* option.

Interpolation Dialog

When the user selects an interpolation command, the *Interpolation Option* dialog. The user selects the appropriate options and once the **OK** button is selected, the interpolation procedure is performed. The user specified options include:

- Interpolation Method – The user selects a current method that is used for all interpolation until the user selects another method. The supported methods include:
  - Standard (Linear)
  - Inverse Distance Weighted (IDW)
- Extrapolation Method – If the VTK dataset does not bound the data being interpolated to, an extrapolation value is used for each location outside the boundary or if the location has be marked as inactive. The user selects the method to be used to generate the extrapolation value from the following:
  - Inactive – The dataset value is set to a null indicator and is not displayed.

    (Note: scatter sets do not support inactive cell so this option is not available when they are the selected target.)
  - Inverse Distance Weighted (IDW)
  - Constant Value – A single value is applied to all extrapolated locations in the dataset.
  - Existing Dataset Value – The corresponding value from a specified existing dataset can be used for locations outside of the bounds of the source VTK dataset. The dataset must be from the same object being interpolated

to and must be of the same type (i.e. scalar, vector).

- Target objects – The use selects a target for interpolation from this tree list of geometrics objects.
- Source object info – This section contains a tree list of the source datasets and information regarding their usage as follows
  - Datasets to interpolate – The user selects the datasets to interpolate from by marking their check box. Multiple datasets may be interpolated at a time.
  - New name – The user may rename the dataset created as a result of the interpolation.
  - Map Z – The user may designate the dataset created as the elevation or "Z" dataset by marking the appropriate check box.
  - Extrapolation Constant Value – The column is displayed when the "Constant Value" Extrapolation Method is selected. The user must enter the single value to apply to the extrapolation locations for each dataset to be interpolated from.
  - Extrapolation Dataset – This column is displayed when the "Existing Dataset Value" Extrapolation Method is selected. The user must select an extrapolation dataset for each dataset to be interpolated from.

Related Topics

- Scatter Interpolation

Article Sources and Contributors

What is SMS? *Source*: http://www.xmswiki.com/xms/index.php?oldid=41243 *Contributors*: Aanderson, Jcreer

Tutorials *Source*: http://www.xmswiki.com/xms/index.php?oldid=64319 *Contributors*: Admin, Jcreer, Jdhowlett, Morelandtom

Sample Problems *Source*: http://www.xmswiki.com/xms/index.php?oldid=64902 *Contributors*: Jcreer, Jdhowlett

Highlights *Source*: http://www.xmswiki.com/xms/index.php?oldid=63947 *Contributors*: Jcreer, Rjones

What's New in SMS 11.2 *Source*: http://www.xmswiki.com/xms/index.php?oldid=66166 *Contributors*: Azundel, Jcreer

General Interface Features *Source*: http://www.xmswiki.com/xms/index.php?oldid=50867 *Contributors*: Jcreer, Rjones

Keyboard Shortcuts *Source*: http://www.xmswiki.com/xms/index.php?oldid=61992 *Contributors*: Jcreer, Jdhowlett

Publications *Source*: http://www.xmswiki.com/xms/index.php?oldid=63939 *Contributors*: Jcreer, Jdhowlett

SMS Screen *Source*: http://www.xmswiki.com/xms/index.php?oldid=63940 *Contributors*: Jcreer

Layout *Source*: http://www.xmswiki.com/xms/index.php?oldid=41612 *Contributors*: Jcreer, Jdhowlett

Data Toolbar *Source*: http://www.xmswiki.com/xms/index.php?oldid=61943 *Contributors*: Jcreer, Sramiah

Dynamic Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=51492 *Contributors*: Jcreer, Jdhowlett

Edit Window *Source*: http://www.xmswiki.com/xms/index.php?oldid=58163 *Contributors*: Jcreer, Jdhowlett

Graphics Window *Source*: http://www.xmswiki.com/xms/index.php?oldid=41017 *Contributors*: Jcreer, Jdhowlett

Help or Status Window *Source*: http://www.xmswiki.com/xms/index.php?oldid=61936 *Contributors*: Jcreer, Jdhowlett

Macros *Source*: http://www.xmswiki.com/xms/index.php?oldid=61939 *Contributors*: Azundel, Jcreer, Jdhowlett

Menu Bar *Source*: http://www.xmswiki.com/xms/index.php?oldid=61808 *Contributors*: Azundel, Jcreer, Jdhowlett

Project Explorer *Source*: http://www.xmswiki.com/xms/index.php?oldid=47786 *Contributors*: Azundel, Jcreer

Project Explorer Right-Click Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=62023 *Contributors*: Azundel, Jcreer, Sramiah

SMS Menus *Source*: http://www.xmswiki.com/xms/index.php?oldid=63941 *Contributors*: Jcreer

Static Tools *Source*: http://www.xmswiki.com/xms/index.php?oldid=47383 *Contributors*: Jcreer, Jdhowlett

Time Step Window *Source*: http://www.xmswiki.com/xms/index.php?oldid=63942 *Contributors*: Jcreer, Jdhowlett

Toolbars *Source*: http://www.xmswiki.com/xms/index.php?oldid=63936 *Contributors*: Jcreer, Rjones

Support *Source*: http://www.xmswiki.com/xms/index.php?oldid=63943 *Contributors*: Jcreer

Downloads *Source*: http://www.xmswiki.com/xms/index.php?oldid=63944 *Contributors*: Jcreer, Rjones

FTP Site Info *Source*: http://www.xmswiki.com/xms/index.php?oldid=63945 *Contributors*: Abarney, Jcreer

License Agreement *Source*: http://www.xmswiki.com/xms/index.php?oldid=47410 *Contributors*: Jcreer, Ksmith

Registering SMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=47407 *Contributors*: Jcreer

Hardware Locks *Source*: http://www.xmswiki.com/xms/index.php?oldid=43476 *Contributors*: Jcreer

Graphics Card Troubleshooting *Source*: http://www.xmswiki.com/xms/index.php?oldid=61102 *Contributors*: Jcreer, Ksmith

Introduction to Setting up SMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=63946 *Contributors*: Jcreer, Jdhowlett

System Requirements *Source*: http://www.xmswiki.com/xms/index.php?oldid=47116 *Contributors*: Jcreer, Ksmith, Morelandtom, Wood

Breaklines *Source*: http://www.xmswiki.com/xms/index.php?oldid=61948 *Contributors*: Azundel, Jcreer

Object Info *Source*: http://www.xmswiki.com/xms/index.php?oldid=63854 *Contributors*: Jcreer

Materials Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=64882 *Contributors*: Jcreer, Jdhowlett

Get Online Maps *Source*: http://www.xmswiki.com/xms/index.php?oldid=64372 *Contributors*: Cmsmemoe, Jcreer, Unjedai

CAD Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=66049 *Contributors*: Azundel, Jcreer

Plot Window *Source*: http://www.xmswiki.com/xms/index.php?oldid=51434 *Contributors*: Azundel, Jcreer, Rjones

Computed vs. Observed Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=47326 *Contributors*: Jcreer, Jdhowlett

Error Summary *Source*: http://www.xmswiki.com/xms/index.php?oldid=61951 *Contributors*: Jcreer, Jdhowlett

Error vs. Simulation Plot *Source*: http://www.xmswiki.com/xms/index.php?oldid=41980 *Contributors*: Jcreer, Jdhowlett

Error vs. Time Step Plot *Source*: http://www.xmswiki.com/xms/index.php?oldid=47513 *Contributors*: Jcreer, Jdhowlett

Observation Profile *Source*: http://www.xmswiki.com/xms/index.php?oldid=61953 *Contributors*: Jcreer, Jdhowlett

Profile Customization Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=63852 *Contributors*: Jcreer, Jdhowlett

Residual vs. Observed Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=47512 *Contributors*: Jcreer, Jdhowlett

Time Series *Source*: http://www.xmswiki.com/xms/index.php?oldid=64233 *Contributors*: Jcreer, Jdhowlett

Time Series Data File *Source*: http://www.xmswiki.com/xms/index.php?oldid=63847 *Contributors*: Jcreer, Jdhowlett

Time Series Plot *Source*: http://www.xmswiki.com/xms/index.php?oldid=51696 *Contributors*: Jcreer, Jdhowlett

View Values Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=63851 *Contributors*: Jcreer, Jdhowlett

ARR Mesh Quality Assessment Plot *Source*: http://www.xmswiki.com/xms/index.php?oldid=63848 *Contributors*: Jcreer

Animations *Source*: http://www.xmswiki.com/xms/index.php?oldid=63948 *Contributors*: Jcreer, Rjones

Film Loop Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64048 *Contributors*: Bmerrell, Jcreer

Film Loop Drogue Plot Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=63951 *Contributors*: Bmerrell, Jcreer

Film Loop Flow Trace Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=63952 *Contributors*: Bmerrell, Jcreer

Film Loop General Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=63958 *Contributors*: Jcreer, Jdhowlett

Film Loop Multiple Views *Source*: http://www.xmswiki.com/xms/index.php?oldid=63957 *Contributors*: Bmerrell, Jcreer

Film Loop Time Step Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=63956 *Contributors*: Jcreer, Jdhowlett

Coordinate Systems *Source*: http://www.xmswiki.com/xms/index.php?oldid=41675 *Contributors*: Jcreer, Ksmith

Coordinate Conversions *Source*: http://www.xmswiki.com/xms/index.php?oldid=55766 *Contributors*: Jcreer, Ksmith

CPP Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=62596 *Contributors*: Jcreer, Ksmith

Geographic Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=36074 *Contributors*: Ksmith, Unjedai

Local Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=38290 *Contributors*: Jcreer, Ksmith

Single Point Conversion *Source*: http://www.xmswiki.com/xms/index.php?oldid=41732 *Contributors*: Jcreer, Unjedai

Projections *Source*: http://www.xmswiki.com/xms/index.php?oldid=65661 *Contributors*: Jcreer, Ksmith, Unjedai

UTM Africa *Source*: http://www.xmswiki.com/xms/index.php?oldid=28045 *Contributors*: Jdhowlett

UTM Asia *Source*: http://www.xmswiki.com/xms/index.php?oldid=28049 *Contributors*: Jdhowlett

UTM Australia *Source*: http://www.xmswiki.com/xms/index.php?oldid=28050 *Contributors*: Jdhowlett

UTM Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=45396 *Contributors*: Jdhowlett, Unjedai

UTM Europe *Source*: http://www.xmswiki.com/xms/index.php?oldid=28051 *Contributors*: Jdhowlett

UTM North America *Source*: http://www.xmswiki.com/xms/index.php?oldid=28052 *Contributors*: Jdhowlett

UTM South American *Source*: http://www.xmswiki.com/xms/index.php?oldid=28053 *Contributors*: Jdhowlett

State Plane Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=28041 *Contributors*: Jdhowlett

Alaska State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28054 *Contributors*: Jdhowlett

Hawaii State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28055 *Contributors*: Jdhowlett

Mideast State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28056 *Contributors*: Jdhowlett

Midwest State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28059 *Contributors*: Jdhowlett

New England State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28057 *Contributors*: Jdhowlett

Northwest State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28060 *Contributors*: Jdhowlett

South Central State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28061 *Contributors*: Jdhowlett

South East State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28062 *Contributors*: Jdhowlett

Southwest State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=28063 *Contributors*: Jdhowlett

Virginia Area State Plane *Source*: http://www.xmswiki.com/xms/index.php?oldid=38291 *Contributors*: Jcreer

Datasets *Source*: http://www.xmswiki.com/xms/index.php?oldid=65261 *Contributors*: Azundel, Jcreer, Jdhowlett

Create Datasets *Source*: http://www.xmswiki.com/xms/index.php?oldid=65262 *Contributors*: Azundel, Jcreer

Dataset Toolbox *Source*: http://www.xmswiki.com/xms/index.php?oldid=65263 *Contributors*: Jcreer, Jdhowlett

Data Calculator *Source*: http://www.xmswiki.com/xms/index.php?oldid=65264 *Contributors*: Jcreer, Jdhowlett, Ksmith

Size Dataset *Source*: http://www.xmswiki.com/xms/index.php?oldid=65265 *Contributors*: Jcreer, Jdhowlett

Smooth Dataset *Source*: http://www.xmswiki.com/xms/index.php?oldid=65266 *Contributors*: Jcreer, Twood, Wood

Metadata *Source*: http://www.xmswiki.com/xms/index.php?oldid=65267 *Contributors*: Azundel, Jcreer

Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64041 *Contributors*: Jcreer, Sramiah

Color Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64647 *Contributors*: Jcreer, Jdhowlett

Functional Surfaces *Source*: http://www.xmswiki.com/xms/index.php?oldid=61983 *Contributors*: Jcreer, Jdhowlett

Lighting Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64043 *Contributors*: Cdickson, Jcreer, Ksmith

Raster Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64059 *Contributors*: Cdickson, Jcreer, Wood

General Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64055 *Contributors*: Jcreer, Jdhowlett

Z Magnification *Source*: http://www.xmswiki.com/xms/index.php?oldid=46714 *Contributors*: Jcreer

Contour Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64191 *Contributors*: Jcreer, Jdhowlett

Vector Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=60483 *Contributors*: Jcreer, Jdhowlett

Visualization for 3D Solutions *Source*: http://www.xmswiki.com/xms/index.php?oldid=47466 *Contributors*: Jcreer

File Import Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=64374 *Contributors*: Jcreer, Jdhowlett

File Import Filter Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=64376 *Contributors*: Jcreer, Jdhowlett

File Import Wizard Supported File Formats *Source*: http://www.xmswiki.com/xms/index.php?oldid=64377 *Contributors*: Jcreer, Jdhowlett

Export Tabular File *Source*: http://www.xmswiki.com/xms/index.php?oldid=61987 *Contributors*: Azundel, Jcreer

Exporting Profile Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64035 *Contributors*: Jcreer, Jdhowlett

Export Dataset Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=64036 *Contributors*: Jcreer, Jdhowlett

Export Map Data in Shape Format *Source*: http://www.xmswiki.com/xms/index.php?oldid=47451 *Contributors*: Jcreer

Data Transform *Source*: http://www.xmswiki.com/xms/index.php?oldid=64112 *Contributors*: Jcreer, Jdhowlett

Zonal Classification *Source*: http://www.xmswiki.com/xms/index.php?oldid=64113 *Contributors*: Jcreer, Rjones

Images *Source*: http://www.xmswiki.com/xms/index.php?oldid=65691 *Contributors*: Jcreer, Wood

Image Pyramids *Source*: http://www.xmswiki.com/xms/index.php?oldid=65692 *Contributors*: Jcreer, Jdhowlett

Import from Web *Source*: http://www.xmswiki.com/xms/index.php?oldid=62616 *Contributors*: Azundel, Cmsmemoe, Jcreer, Ksmith, Unjedai, Wood

Registering an Image *Source*: http://www.xmswiki.com/xms/index.php?oldid=65690 *Contributors*: Jcreer, Ksmith

Save as Image *Source*: http://www.xmswiki.com/xms/index.php?oldid=65693 *Contributors*: Jcreer, Sramiah

Web Service for Background Imagery *Source*: http://www.xmswiki.com/xms/index.php?oldid=65694 *Contributors*: Jcreer

Preferences *Source*: http://www.xmswiki.com/xms/index.php?oldid=64037 *Contributors*: Jcreer, Rjones

Time Settings *Source*: http://www.xmswiki.com/xms/index.php?oldid=64040 *Contributors*: Jcreer, Jdhowlett

Spectral Energy *Source*: http://www.xmswiki.com/xms/index.php?oldid=66298 *Contributors*: Jcreer, Jdhowlett

Create Spectral Energy Grid *Source*: http://www.xmswiki.com/xms/index.php?oldid=64136 *Contributors*: Abarney, Jcreer

Generate/Edit Spectra *Source*: http://www.xmswiki.com/xms/index.php?oldid=64138 *Contributors*: Jcreer, Jschultze

Import Spectra *Source*: http://www.xmswiki.com/xms/index.php?oldid=64380 *Contributors*: Jcreer, Jdhowlett

Spectral Grid Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=64134 *Contributors*: Abarney, Jcreer

Spectral Events *Source*: http://www.xmswiki.com/xms/index.php?oldid=66295 *Contributors*: Jcreer

Datasets VTK *Source*: http://www.xmswiki.com/xms/index.php?oldid=47791 *Contributors*: Jcreer

Conversions Scalar↔Vector *Source*: http://www.xmswiki.com/xms/index.php?oldid=62002 *Contributors*: Jcreer

Dataset Calculator VTK *Source*: http://www.xmswiki.com/xms/index.php?oldid=62004 *Contributors*: Jcreer

Interpolation VTK *Source*: http://www.xmswiki.com/xms/index.php?oldid=66100 *Contributors*: Jcreer

Image Sources, Licenses and Contributors

Image:SMS Layout.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Layout.png *License*: unknown *Contributors*: Rjones
Image:Module toolbar.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Module_toolbar.png *License*: unknown *Contributors*: Rjones
Image:mesh_module.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mesh_module.png *License*: unknown *Contributors*: Rjones
Image:Cartesian Grid.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cartesian_Grid.png *License*: unknown *Contributors*: Rjones
Image:Scatter Set Example.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scatter_Set_Example.png *License*: unknown *Contributors*: Rjones
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
Image:Map_Data_Set.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map_Data_Set.png *License*: unknown *Contributors*: Rjones
file:Ptm particle set.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ptm_particle_set.jpg *License*: unknown *Contributors*: Jcreer
Image:Generic Model Interface.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Generic_Model_Interface.jpg *License*: unknown *Contributors*: Azundel
Image:SMS color filled contours.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_color_filled_contours.jpg *License*: unknown *Contributors*: Rjones
file:Vectorvisual.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Vectorvisual.png *License*: unknown *Contributors*: Jcreer
Image:ProfilePlot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ProfilePlot.gif *License*: unknown *Contributors*: Abarney
file:Flow_trace.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Flow_trace.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS_Data_Calculator.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Data_Calculator.jpg *License*: unknown *Contributors*: Jcreer, Jdhowlett
Image:CreateDS.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CreateDS.jpg *License*: unknown *Contributors*: Jhworth
Image:Transform Feature Objects.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Transform_Feature_Objects.jpg *License*: unknown *Contributors*: Jcreer
Image:CoordTrans.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CoordTrans.jpg *License*: unknown *Contributors*: Morelandtom
Image:ImportWizard2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ImportWizard2.jpg *License*: unknown *Contributors*: Morelandtom
Image:SMS image transparency.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_image_transparency.jpg *License*: unknown *Contributors*: Morelandtom
Image:CAD_Data.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:CAD_Data.jpg *License*: unknown *Contributors*: Morelandtom
Image:GoogleEarth.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GoogleEarth.jpg *License*: unknown *Contributors*: Morelandtom
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:Terraserver.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Terraserver.jpg *License*: unknown *Contributors*: Morelandtom
Image:ZonalClass.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ZonalClass.jpg *License*: unknown *Contributors*: Morelandtom
Image:SMSscreen.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMSscreen.jpg *License*: unknown *Contributors*: Jcreer
Image:Statusbar1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Statusbar1.gif *License*: unknown *Contributors*: Abarney
Image:Statusbar2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Statusbar2.gif *License*: unknown *Contributors*: Abarney
Image:Optional Macros Toolbar.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Optional_Macros_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:File Toolbar.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:File_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Print.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Print.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Display Toolbar.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
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
Image:dispopts.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dispopts.gif *License*: unknown *Contributors*: Jdhowlett
Image:PlanView.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:PlanView.jpg *License*: unknown *Contributors*: Abarney
Image:SMS Selection Info.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Selection_Info.jpg *License*: unknown *Contributors*: Jcreer
Image:Static Tools Toolbar.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Static_Tools_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Tool_Pan.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Pan.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Zoom.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Zoom.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Rotate.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Rotate.GIF *License*: unknown *Contributors*: Jhandy
File:Time Step Window.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Time_Step_Window.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:Breakline.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Breakline.gif *License*: unknown *Contributors*: Cballard
Image:SMS Info Carte.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Info_Carte.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Get Info Map.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Get_Info_Map.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Info Mesh.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Info_Mesh.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Info Scatter.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Info_Scatter.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Info Raster.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Info_Raster.jpg *License*: unknown *Contributors*: Jcreer
File:SMS Materials Data.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Materials_Data.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Materials Legend.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Materials_Legend.jpg *License*: unknown *Contributors*: Jcreer
File:GetOnlineMapsDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetOnlineMapsDialog.png *License*: unknown *Contributors*: Cmsmemoe
Image:ComputedVsObserved.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ComputedVsObserved.gif *License*: unknown *Contributors*: Abarney
Image:ErrorSummary.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ErrorSummary.gif *License*: unknown *Contributors*: Abarney
Image:ErrorVsSimulation.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ErrorVsSimulation.gif *License*: unknown *Contributors*: Abarney
Image:ErrorVsTimestep.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ErrorVsTimestep.gif *License*: unknown *Contributors*: Abarney
Image:Plot3.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Plot3.JPG *License*: unknown *Contributors*: Astrassburg
Image:Plot4.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Plot4.JPG *License*: unknown *Contributors*: Astrassburg
Image:SMS Profile Customization.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Profile_Customization.jpg *License*: unknown *Contributors*: Jcreer
Image:ResidualVsObserved.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ResidualVsObserved.gif *License*: unknown *Contributors*: Abarney
Image:TimeSeriesEditor.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TimeSeriesEditor.jpg *License*: unknown *Contributors*: Jdhowlett
Image:TimeSeries.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:TimeSeries.gif *License*: unknown *Contributors*: Abarney
Image:SMS View Values.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_View_Values.jpg *License*: unknown *Contributors*: Jcreer
File:elem_angles.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_angles.png *License*: unknown *Contributors*: Jcreer
File:BaryDomain.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:BaryDomain.png *License*: unknown *Contributors*: Jcreer
File:elem_angles_ordered.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_angles_ordered.png *License*: unknown *Contributors*: Jcreer
File:ARRdomain.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ARRdomain.png *License*: unknown *Contributors*: Jcreer
File:Contours.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Contours.png *License*: unknown *Contributors*: Jcreer
Image:Bama.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Bama.gif *License*: unknown *Contributors*: Cballard

Image:Noyo.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Noyo.gif *License*: unknown *Contributors*: Cballard
Image:SMS Film Loop Display Opt.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Film_Loop_Display_Opt.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Film Loop general.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Film_Loop_general.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Film Loop Multi.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Film_Loop_Multi.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Film Loop Time.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Film_Loop_Time.jpg *License*: unknown *Contributors*: Jcreer
Image:ellipsiodcoordsys.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ellipsiodcoordsys.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys1.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys2.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys3.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys4.jpg *License*: unknown *Contributors*: Ksmith
Image:Africa.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Africa.gif *License*: unknown *Contributors*: Cballard
Image:asia.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Asia.gif *License*: unknown *Contributors*: Cballard
Image:Australia.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Australia.gif *License*: unknown *Contributors*: Cballard
Image:Usutm.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Usutm.gif *License*: unknown *Contributors*: Jdhowlett
Image:UTM_world_no_Image_Map.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:UTM_world_no_Image_Map.jpg *License*: unknown *Contributors*: Abarney
Image:Europe.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Europe.gif *License*: unknown *Contributors*: Cballard
Image:Northamerica.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Northamerica.gif *License*: unknown *Contributors*: Cballard
Image:Southamerica.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Southamerica.gif *License*: unknown *Contributors*: Cballard
Image:USstate_plane_no_image_map.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:USstate_plane_no_image_map.jpg *License*: unknown *Contributors*: Abarney
Image:Ak27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ak27.gif *License*: unknown *Contributors*: Cballard
Image:Ha27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ha27.gif *License*: unknown *Contributors*: Cballard
Image:Me27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Me27.gif *License*: unknown *Contributors*: Cballard
Image:Mw27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mw27.gif *License*: unknown *Contributors*: Cballard
Image:Ne27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ne27.gif *License*: unknown *Contributors*: Cballard
Image:Nw27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Nw27.gif *License*: unknown *Contributors*: Cballard
Image:Sc27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sc27.gif *License*: unknown *Contributors*: Cballard
Image:Se27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Se27.gif *License*: unknown *Contributors*: Cballard
Image:Sw27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sw27.gif *License*: unknown *Contributors*: Cballard
Image:Virg27.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Virg27.gif *License*: unknown *Contributors*: Cballard
Image:SMS Dataset Info.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Dataset_Info.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Dataset Toolbox.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Dataset_Toolbox.jpg *License*: unknown *Contributors*: Jcreer
Image:Smooth_Data_Sets.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Smooth_Data_Sets.jpg *License*: unknown *Contributors*: Jcreer, Jdhowlett
Image:SMS Metadata.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Metadata.jpg *License*: unknown *Contributors*: Jcreer
Image:Metadata Profile Info.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Metadata_Profile_Info.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Spatial Metadata.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Spatial_Metadata.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Source Metadata.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Source_Metadata.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Dataset Metadata.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Dataset_Metadata.jpg *License*: unknown *Contributors*: Jcreer
Image:Dispopts.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dispopts.gif *License*: unknown *Contributors*: Jdhowlett
Image:Color Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Color_Options.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS New Palette.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_New_Palette.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Functional Surface Opt.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Functional_Surface_Opt.jpg *License*: unknown *Contributors*: Jcreer
Image:LightingDialog.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:LightingDialog.jpg *License*: unknown *Contributors*: Ksmith
Image:RasterDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:RasterDialog.png *License*: unknown *Contributors*: Cdickson
Image:SMS Display Opts General.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Display_Opts_General.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Displaying Opts View.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Displaying_Opts_View.jpg *License*: unknown *Contributors*: Jcreer
image:Contour Label.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Contour_Label.jpg *License*: unknown *Contributors*: Jcreer
Image:Contout Legend.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Contout_Legend.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Display Opt Vector.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Display_Opt_Vector.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Vector Legend Opt.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Vector_Legend_Opt.jpg *License*: unknown *Contributors*: Jcreer
Image:File Import Wizard - Step 1 of 2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:File_Import_Wizard_-_Step_1_of_2.jpg *License*: unknown *Contributors*: Jdhowlett
Image:File Import Wizard - Step 2 of 2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:File_Import_Wizard_-_Step_2_of_2.jpg *License*: unknown *Contributors*: Jdhowlett
Image:File Import Filter Options.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:File_Import_Filter_Options.jpg *License*: unknown *Contributors*: Jdhowlett
Image:SMS Exporting Plot Data.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Exporting_Plot_Data.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Export dataset.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Export_dataset.jpg *License*: unknown *Contributors*: Jcreer
Image:step33f.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Step33f.gif *License*: unknown *Contributors*: Trabadi
Image:step44.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Step44.gif *License*: unknown *Contributors*: Trabadi
Image:step55.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Step55.gif *License*: unknown *Contributors*: Trabadi
Image:step66.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Step66.gif *License*: unknown *Contributors*: Trabadi
Image:step77.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Step77.gif *License*: unknown *Contributors*: Trabadi
Image:ImportFromWeb-VirtualEarthMapLocator.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ImportFromWeb-VirtualEarthMapLocator.png *License*: unknown *Contributors*: Unjedai
Image:SelectRegistrationPoint.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectRegistrationPoint.jpg *License*: unknown *Contributors*: Abarney
Image:Zoom.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Zoom.JPG *License*: unknown *Contributors*: Jdhowlett
Image:Pan.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Pan.JPG *License*: unknown *Contributors*: Jdhowlett
Image:SMS Pref General.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_General.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Pref Startup.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_Startup.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Pref Images.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_Images.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Pref File.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_File.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Pref Project.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_Project.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Pref Toolbars.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_Toolbars.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Pref Time.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_Time.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Pref Map.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_Map.jpg *License*: unknown *Contributors*: Jcreer

Image:SMS Pref Graphics.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Pref_Graphics.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Time Settings.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Time_Settings.jpg *License*: unknown *Contributors*: Jcreer
File:SpectralEnergy.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SpectralEnergy.jpg *License*: unknown *Contributors*: Jcreer
Image:SelectCGridCell.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectCGridCell.JPG *License*: unknown *Contributors*: Abarney
Image:Rotate.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Rotate.JPG *License*: unknown *Contributors*: Jdhowlett
Image:contours.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Contours.gif *License*: unknown *Contributors*: Abarney
File:SpectralTimeSettings.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SpectralTimeSettings.jpg *License*: unknown *Contributors*: Jcreer
File:Create Spectral Energy.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Create_Spectral_Energy.jpg *License*: unknown *Contributors*: Jcreer
File:GenerateEditSpectra.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GenerateEditSpectra.jpg *License*: unknown *Contributors*: Jcreer
File:Import Spectra.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Import_Spectra.jpg *License*: unknown *Contributors*: Jcreer
File:Spectral Grid Properties.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Spectral_Grid_Properties.jpg *License*: unknown *Contributors*: Jcreer
File:SpectralEvents.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SpectralEvents.jpg *License*: unknown *Contributors*: Jcreer