SMS User Manual (v12.2)

The Surface Water Modeling System

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 in the late 1980s on Unix workstations. The development of SMS was funded primarily by The United States Army Corps of Engineers . 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 . and continue to develop SMS and other software products, such as WMS (Watershed Modeling System) and GMS (Groundwater Modeling System) .

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: download the tutorials and files individually by subject matter in the SMS learning center or 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 a subset of the tutorials is often all that is needed, 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 needed to learn how to use SMS. It is there that all of the SMS tutorial documents and files can be accessed via download. To access the SMS Learning Center and tutorials click here , or follow these steps:

- Go to www.aquaveo.com
- Click on the *SMS logo* found on the right side of the screen. This will go 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.
- Choose the directory in which to run and save the tutorial files.
- Once the files have finished downloading, select the **Open** option.
- When the data files come up, extract the files by clicking on the **Extract all files** option.
- Once the files are extracted, begin the tutorial.

Available Tutorials

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

SMS Tutorials		
Data Visualization	Feature Stamping	GIS
Google Earth	Import From Web	Import Spectral Data
Mesh Editing	Observation	Overview
Scattered Datasets	Sensitivity	Size Function
SMS Model Tutorials		
ADCIRC	ADCIRC LTEA Meshing	ADCIRC Symmetric Hurricane Simulation
BOUSS2D	CGWAVE	Additional CGWAVE
CMS Flow	CMS Wave	FESWMS
FESWMS Steering	FESWMS Weirs	Generic Mesh Model
PTM	RMA2	RMA2 Steering
RMA4	SRH-2D	SRH-2D Culvert Structures
SRH-2D Culvert Structures HY-8	SRH-2D Gates	SRH-2D Obstructions
SRH-2D Bridge Pressure Flow	SRH-2D Simulations	SRH-2D Weir Flow

STWAVE	TUFLOW 1D	TUFLOW 2D
TUFLOW AD	TUFLOW FV	Additional TUFLOW
WAM		

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

Related Topics

- Sample Problems
- SMS Tutorial Archive
- SMS Tutorial History

Sample Problems

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

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 searching for case studies containing particular attributes. Performing a search with no selections will allow browsing all studies contained in the repository.

Test cases can be added to the repository . In order to add studies to the repository an account is required. The provided contact information will not be released to anyone. Everyone is invited 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
- Hollingsworth, Jason M (2008). Foundational Data Repository for Numeric Engine Validation. Thesis, Brigham Young University. [0]

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...

Modules

- Data is divided into modules based upon the data type
- 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...

Cartesian Grid Module

- Used to create, edit, and visualize rectilinear grids
- Datasets can have values at cells, corners, and midsides
- Can use cell-centered or mesh-centered grids

More Info...

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...

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...

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...

Particle Module

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

More Info...

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...

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...

Vectors

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

More Info...

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 (flow rate) data at a point or across an arc
- 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

- 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

- 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...

Data Transform

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

More Info...

Projections

- Associate a projection with data
- Ability to work with global, local, and no projection
- Convert data from one projection to another
- Projections include Geographic, UTM, and State Plane coordinate systems
- Reproject data on the fly align multiple projections

More Info...

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...

Miscellaneous Tools

Image Support

- Multiple images can be read/viewed at the same time (tiled or overlay)

- Independent transparency specified for each image
- Images can be loaded from web services as either static or dynamic images
- Images can be draped over mesh or scatter data
- Many image formats are supported including JPG, TIFF, PNG, MrSID, and ECW
- Local images can be geo-referenced to view images along with other data
- Image pyramids can be created for very large images

More Info...

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...

Export Options

- Graphics window can be copied to the clipboard
- Current view can be exported in KML format for visualization 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 *Dataset Toolbox* , or LTEA (linear truncation error analysis) (ADCIRC)

More Info...

Import from Web

- Easy to use navigation tool that allows choosing a model location

- Image data is downloaded from USGS terraserver
- Image options include aerial photos, topographic charts, and urban (higher resolution color)

More Info...

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...

What's New in SMS 12.2

Community Edition

The SMS Community Edition is an unlicensed version that allows using a basic interface to:

- Open and view files.
- Generate a mesh (limited to one mesh).
- Use the SRH-2D interface (limited to one simulation).

General Features

- Options added to Smooth Arc tool.
- Size functions can be used to redistribute vertices along an arc.

Module Features

Curvilinear Grid

- SMS can now use multiple curvilinear grids.
- Curvilinear grids can now be duplicated.

UGrid Module

- UGrid module added to SMS.

Model Features

CMS-Flow

- Structures coverage added.
- Monitoring stations coverage added.

HEC-RAS

- Now uses the simulation process similar to that used in SRH-2D.
- Can be used to export geometry generated in SMS for use in HEC-RAS.

SHR-2D

- New sediment materials coverage.
- Pressure zones with arched ceiling elevations.
- Weir flow over pressure zones.
- Internal links to connect internal source and sinks.

- Internal source/sinks.
- Option to use energy total head instead of water-surface elevation for culverts.

1.1. Support

Support

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

Support Forum

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

Related Topics

- GMS
- SMS
- WMS
- Tech Support Agreement

External Links

- Aquaveo Technical Support

System Requirements

System requirements for GMS, SMS and WMS.

Windows 10

Windows 10 is supported in GMS 10.0, SMS 12.1, WMS 10.0 and greater versions only.

Component	Minimum Required	Recommended
RAM	1 GB	8 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 or higher 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 8

Windows 8 may have compatibility issues with XMS software. It is recommended to upgrade to Windows 10.

Component	Minimum Required	Recommended
RAM	1 GB	8 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 7Windows 7 is supported in GMS 7.0, SMS 10.1 and greater versions only.

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

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	8 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 compatible with GMS 7.0, SMS 10.0, WMS 8.1 and some greater versions, but may not be compatible with current versions. There is limited technical support for Windows XP, and some limitations using certain fonts and display options.

Component	Minimum Required	Recommended
RAM	1 GB	8 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

Notes

- There may be display problems when running over a 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 continuing to experience problems after updating a graphics card drivers, contact support .

External Links

- Description of the DirectX Diagnostic Tool

Downloads

The Download Center at www.aquaveo.com has 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
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

Related Topics

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

FTP Site Info

Technical support may request uploading files to the ftp server in order to determine the source of issues.

To upload files:

1) Zip up the files to upload.
2) Go to this link:
    1. | Upload Link

3) Paste files in window.

Related Topics

- XMS FTP Site Info

License Agreement

XMS License Agreement

XMS 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. We agree that these programs may send information back to Aquaveo in order to enable a program on the computer in use. Additionally, parts of the software may be covered by one or more of the following agreements.

Boost Software License

Boost Software License - Version 1.0 - August 17th, 2003

Permission is hereby granted, free of charge, to any person or organization obtaining a copy of the software and accompanying documentation covered by this license (the "Software") to use, reproduce, display, distribute, execute, and transmit the Software, and to prepare derivative works of the Software, and to permit third-parties to whom the Software is furnished to do so, all subject to the following:

The copyright notices in the Software and this entire statement, including the above license grant, this restriction and the following disclaimer, must be included in all copies of the Software, in whole or in part, and all derivative works of the Software, unless such copies or derivative works are solely in the form of machine-executable object code generated by a source language processor.

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, TITLE AND NON-INFRINGEMENT. IN NO EVENT SHALL THE COPYRIGHT HOLDERS OR ANYONE DISTRIBUTING THE SOFTWARE BE LIABLE FOR ANY DAMAGES OR OTHER LIABILITY, WHETHER IN CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

FTGL License Agreement

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.

Bugtrap

http://www.codeproject.com/Articles/14618/Catch-All-Bugs-with-BugTrap

The above URL states that Bugtrap is licensed as follows:

Microsoft Public License (MS-PL)

[OSI Approved License]

GNU Lesser General Public License

GNU LESSER GENERAL PUBLIC LICENSE

Version 2.1, February 1999

[This is the first released version of the Lesser GPL. It also counts as the successor of the GNU Library Public License, version 2, hence the version number 2.1.]

Preamble

The licenses for most software are designed to take away your freedom to share and change it. By contrast, the GNU General Public Licenses are intended to guarantee your freedom to share and change free software--to make sure the software is free for all its users.

This license, the Lesser General Public License, applies to some specially designated software packages--typically libraries--of the Free Software Foundation and other authors who decide to use it. You can use it too, but we suggest you first think carefully about whether this license or the ordinary General Public License is the better strategy to use in any particular case, based on the explanations below.

When we speak of free software, we are referring to freedom of use, not price. Our General Public Licenses are designed to make sure that you have the freedom to distribute copies of free software (and charge for this service if you wish); that you receive source code or can get it if you want it; that you can change the software and use pieces of it in new free programs; and that you are informed that you can do these things.

To protect your rights, we need to make restrictions that forbid distributors to deny you these rights or to ask you to surrender these rights. These restrictions translate to certain responsibilities for you if you distribute copies of the library or if you modify it.

For example, if you distribute copies of the library, whether gratis or for a fee, you must give the recipients all the rights that we gave you. You must make sure that they, too, receive or can get the source code. If you link other code with the library, you must provide complete object files to the recipients, so that they can relink them with the library after making changes to the library and recompiling it. And you must show them these terms so they know their rights.

We protect your rights with a two-step method: (1) we copyright the library, and (2) we offer you this license, which gives you legal permission to copy, distribute and/or modify the library.

To protect each distributor, we want to make it very clear that there is no warranty for the free library. Also, if the library is modified by someone else and passed on, the recipients should know that what they have is not the original version, so that the original author's reputation will not be affected by problems that might be introduced by others.

Finally, software patents pose a constant threat to the existence of any free program. We wish to make sure that a company cannot effectively restrict the users of a free program by obtaining a restrictive license from a patent holder. Therefore, we insist that any patent license obtained for a version of the library must be consistent with the full freedom of use specified in this license.

Most GNU software, including some libraries, is covered by the ordinary GNU General Public License. This license, the GNU Lesser General Public License, applies to certain designated libraries, and is quite different from the ordinary General Public License. We use this license for certain libraries in order to permit linking those libraries into non-free programs.

When a program is linked with a library, whether statically or using a shared library, the combination of the two is legally speaking a combined work, a derivative of the original library. The ordinary General Public License therefore permits such linking only if the entire combination fits its criteria of freedom. The Lesser General Public License permits more lax criteria for linking other code with the library.

We call this license the "Lesser" General Public License because it does Less to protect the user's freedom than the ordinary General Public License. It also provides other free software developers Less of an advantage over competing non-free programs. These disadvantages are the reason we use the ordinary General Public License for many libraries. However, the Lesser license provides advantages in certain special circumstances.

For example, on rare occasions, there may be a special need to encourage the widest possible use of a certain library, so that it becomes a de-facto standard. To achieve this, non-free programs must be allowed to use the library. A more frequent case is that a free library does the same job as widely used non-free libraries. In this case, there is little to gain by limiting the free library to free software only, so we use the Lesser General Public License.

In other cases, permission to use a particular library in non-free programs enables a greater number of people to use a large body of free software. For example, permission to use the GNU C Library in non-free programs enables many more people to use the whole GNU operating system, as well as its variant, the GNU/Linux operating system.

Although the Lesser General Public License is Less protective of the users' freedom, it does ensure that the user of a program that is linked with the Library has the freedom and the wherewithal to run that program using a modified version of the Library.

The precise terms and conditions for copying, distribution and modification follow. Pay close attention to the difference between a "work based on the library" and a "work that uses the library". The former contains code derived from the library, whereas the latter must be combined with the library in order to run.

Terms and Conditions for Copying, Distribution and Modification

GNU LESSER GENERAL PUBLIC LICENSE

TERMS AND CONDITIONS FOR COPYING, DISTRIBUTION AND MODIFICATION

0. This License Agreement applies to any software library or other program which contains a notice placed by the copyright holder or other authorized party saying it may be distributed under the terms of this Lesser General Public License (also called "this License"). Each licensee is addressed as "you".

A "library" means a collection of software functions and/or data prepared so as to be conveniently linked with application programs (which use some of those functions and data) to form executables.

The "Library", below, refers to any such software library or work which has been distributed under these terms. A "work based on the Library" means either the Library or any derivative work under copyright law: that is to say, a work containing the Library or a portion of it, either verbatim or with modifications and/or translated straightforwardly into another language. (Hereinafter, translation is included without limitation in the term "modification".)

"Source code" for a work means the preferred form of the work for making modifications to it. For a library, complete source code means all the source code for all modules it contains, plus any associated interface definition files, plus the scripts used to control compilation and installation of the library.

Activities other than copying, distribution and modification are not covered by this License; they are outside its scope. The act of running a program using the Library is not restricted, and output from such a program is covered only if its contents constitute a work based on the Library (independent of the use of the Library in a tool for writing it). Whether that is true depends on what the Library does and what the program that uses the Library does.

1. You may copy and distribute verbatim copies of the Library's complete source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice and disclaimer of warranty; keep intact all the notices that refer to this License and to the absence of any warranty; and distribute a copy of this License along with the Library.

You may charge a fee for the physical act of transferring a copy, and you may at your option offer warranty protection in exchange for a fee.

2. You may modify your copy or copies of the Library or any portion of it, thus forming a work based on the Library, and copy and distribute such modifications or work under the terms of Section 1 above, provided that you also meet all of these conditions:

a) The modified work must itself be a software library.

b) You must cause the files modified to carry prominent notices stating that you changed the files and the date of any change.

c) You must cause the whole of the work to be licensed at no charge to all third parties under the terms of this License.

d) If a facility in the modified Library refers to a function or a table of data to be supplied by an application program that uses the facility, other than as an argument passed when the facility is invoked, then you must make a good faith effort to ensure that, in the event an application does not supply such function or table, the facility still operates, and performs whatever part of its purpose remains meaningful.

(For example, a function in a library to compute square roots has a purpose that is entirely well-defined independent of the application. Therefore, Subsection 2d requires that any application-supplied function or table used by this function must be optional: if the application does not supply it, the square root function must still compute square roots.)

These requirements apply to the modified work as a whole. If identifiable sections of that work are not derived from the Library, and can be reasonably considered independent and separate works in themselves, then this License, and its terms, do not apply to those sections when you distribute them as separate works. But when you distribute the same sections as part of a whole which is a work based on the Library, the distribution of the whole must be on the terms of this License, whose permissions for other licensees extend to the entire whole, and thus to each and every part regardless of who wrote it.

Thus, it is not the intent of this section to claim rights or contest your rights to work written entirely by you; rather, the intent is to exercise the right to control the distribution of derivative or collective works based on the Library.

In addition, mere aggregation of another work not based on the Library with the Library (or with a work based on the Library) on a volume of a storage or distribution medium does not bring the other work under the scope of this License.

3. You may opt to apply the terms of the ordinary GNU General Public License instead of this License to a given copy of the Library. To do this, you must alter all the notices that refer to this License, so that they refer to the ordinary GNU General Public License, version 2, instead of to this License. (If a newer version than version 2 of the ordinary GNU General Public License has appeared, then you can specify that version instead if you wish.) Do not make any other change in these notices.

Once this change is made in a given copy, it is irreversible for that copy, so the ordinary GNU General Public License applies to all subsequent copies and derivative works made from that copy.

This option is useful when you wish to copy part of the code of the Library into a program that is not a library.

4. You may copy and distribute the Library (or a portion or derivative of it, under Section 2) in object code or executable form under the terms of Sections 1 and 2 above provided that you accompany it with the complete corresponding machine-readable source code, which must be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange.

If distribution of object code is made by offering access to copy from a designated place, then offering equivalent access to copy the source code from the same place satisfies the requirement to distribute the source code, even though third parties are not compelled to copy the source along with the object code.

5. A program that contains no derivative of any portion of the Library, but is designed to work with the Library by being compiled or linked with it, is called a "work that uses the Library". Such a work, in isolation, is not a derivative work of the Library, and therefore falls outside the scope of this License.

However, linking a "work that uses the Library" with the Library creates an executable that is a derivative of the Library (because it contains portions of the Library), rather than a "work that uses the library". The executable is therefore covered by this License. Section 6 states terms for distribution of such executables.

When a "work that uses the Library" uses material from a header file that is part of the Library, the object code for the work may be a derivative work of the Library even though the source code is not. Whether this is true is especially significant if the work can be linked without the Library, or if the work is itself a library. The threshold for this to be true is not precisely defined by law.

If such an object file uses only numerical parameters, data structure layouts and accessors, and small macros and small inline functions (ten lines or less in length), then the use of the object file is unrestricted, regardless of whether it is legally a derivative work. (Executables containing this object code plus portions of the Library will still fall under Section 6.)

Otherwise, if the work is a derivative of the Library, you may distribute the object code for the work under the terms of Section 6. Any executables containing that work also fall under Section 6, whether or not they are linked directly with the Library itself.

6. As an exception to the Sections above, you may also combine or link a "work that uses the Library" with the Library to produce a work containing portions of the Library, and distribute that work under terms of your choice, provided that the terms permit modification of the work for the customer's own use and reverse engineering for debugging such modifications.

You must give prominent notice with each copy of the work that the Library is used in it and that the Library and its use are covered by this License. You must supply a copy of this License. If the work during execution displays copyright notices, you must include the copyright notice for the Library among them, as well as a reference directing the user to the copy of this License. Also, you must do one of these things:

a) Accompany the work with the complete corresponding machine-readable source code for the Library including whatever changes were used in the work (which must be distributed under Sections 1 and 2 above); and, if the work is an executable linked with the Library, with the complete machine-readable "work that uses the Library", as object code and/or source code, so that the user can modify the Library and then relink to produce a modified executable containing the modified Library. (It is understood that the user who changes the contents of definitions files in the Library will not necessarily be able to recompile the application to use the modified definitions.)

b) Use a suitable shared library mechanism for linking with the Library. A suitable mechanism is one that (1) uses at run time a copy of the library already present on the user's computer system, rather than copying library functions into the executable, and (2) will operate properly with a modified version of the library, if the user installs one, as long as the modified version is interface-compatible with the version that the work was made with.

c) Accompany the work with a written offer, valid for at least three years, to give the same user the materials specified in Subsection 6a, above, for a charge no more than the cost of performing this distribution.

d) If distribution of the work is made by offering access to copy from a designated place, offer equivalent access to copy the above specified materials from the same place.

e) Verify that the user has already received a copy of these materials or that you have already sent this user a copy.

For an executable, the required form of the "work that uses the Library" must include any data and utility programs needed for reproducing the executable from it. However, as a special exception, the materials to be distributed need not include anything that is normally distributed (in either source or binary form) with the major components (compiler, kernel, and so on) of the operating system on which the executable runs, unless that component itself accompanies the executable.

It may happen that this requirement contradicts the license restrictions of other proprietary libraries that do not normally accompany the operating system. Such a contradiction means you cannot use both them and the Library together in an executable that you distribute.

7. You may place library facilities that are a work based on the Library side-by-side in a single library together with other library facilities not covered by this License, and distribute such a combined library, provided that the separate distribution of the work based on the Library and of the other library facilities is otherwise permitted, and provided that you do these two things:

a) Accompany the combined library with a copy of the same work based on the Library, uncombined with any other library facilities. This must be distributed under the terms of the Sections above.

b) Give prominent notice with the combined library of the fact that part of it is a work based on the Library, and explaining where to find the accompanying uncombined form of the same work.

8. You may not copy, modify, sublicense, link with, or distribute the Library except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense, link with, or distribute the Library is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

9. You are not required to accept this License, since you have not signed it. However, nothing else grants you permission to modify or distribute the Library or its derivative works. These actions are prohibited by law if you do not accept this License. Therefore, by modifying or distributing the Library (or any work based on the Library), you indicate your acceptance of this License to do so, and all its terms and conditions for copying, distributing or modifying the Library or works based on it.

10. Each time you redistribute the Library (or any work based on the Library), the recipient automatically receives a license from the original licensor to copy, distribute, link with or modify the Library subject to these terms and conditions. You may not impose any further restrictions on the recipients' exercise of the rights granted herein. You are not responsible for enforcing compliance by third parties with this License.

11. If, as a consequence of a court judgment or allegation of patent infringement or for any other reason (not limited to patent issues), conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot distribute so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not distribute the Library at all. For example, if a patent license would not permit royalty-free redistribution of the Library by all those who receive copies directly or indirectly through you, then the only way you could satisfy both it and this License would be to refrain entirely from distribution of the Library.

If any portion of this section is held invalid or unenforceable under any particular circumstance, the balance of the section is intended to apply, and the section as a whole is intended to apply in other circumstances.

It is not the purpose of this section to induce you to infringe any patents or other property right claims or to contest validity of any such claims; this section has the sole purpose of protecting the integrity of the free software distribution system which is implemented by public license practices. Many people have made generous contributions to the wide range of software distributed through that system in reliance on consistent application of that system; it is up to the author/donor to decide if he or she is willing to distribute software through any other system and a licensee cannot impose that choice.

This section is intended to make thoroughly clear what is believed to be a consequence of the rest of this License.

12. If the distribution and/or use of the Library is restricted in certain countries either by patents or by copyrighted interfaces, the original copyright holder who places the Library under this License may add an explicit geographical distribution limitation excluding those countries, so that distribution is permitted only in or among countries not thus excluded. In such case, this License incorporates the limitation as if written in the body of this License.

13. The Free Software Foundation may publish revised and/or new versions of the Lesser General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Library specifies a version number of this License which applies to it and "any later version", you have the option of following the terms and conditions either of that version or of any later version published by the Free Software Foundation. If the Library does not specify a license version number, you may choose any version ever published by the Free Software Foundation.

14. If you wish to incorporate parts of the Library into other free programs whose distribution conditions are incompatible with these, write to the author to ask for permission. For software which is copyrighted by the Free Software Foundation, write to the Free Software Foundation; we sometimes make exceptions for this. Our decision will be guided by the two goals of preserving the free status of all derivatives of our free software and of promoting the sharing and reuse of software generally.

No Warranty

NO WARRANTY

15. BECAUSE THE LIBRARY IS LICENSED FREE OF CHARGE, THERE IS NO WARRANTY FOR THE LIBRARY, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE LIBRARY "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE LIBRARY IS WITH YOU. SHOULD THE LIBRARY PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

16. IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MAY MODIFY AND/OR REDISTRIBUTE THE LIBRARY AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE LIBRARY (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE LIBRARY TO OPERATE WITH ANY OTHER SOFTWARE), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

END OF TERMS AND CONDITIONS

How to Apply These Terms to Your New Libraries

If you develop a new library, and you want it to be of the greatest possible use to the public, we recommend making it free software that everyone can redistribute and change. You can do so by permitting redistribution under these terms (or, alternatively, under the terms of the ordinary General Public License).

To apply these terms, attach the following notices to the library. It is safest to attach them to the start of each source file to most effectively convey the exclusion of warranty; and each file should have at least the "copyright" line and a pointer to where the full notice is found.

```
<one line to give the library's name and a brief idea of what it does.>
Copyright (C) <year> <name of author>
This library is free software; you can redistribute it and/or modify it
under the terms of the GNU Lesser General Public License as published
by the Free Software Foundation; either version 2.1 of the License, or
(at your option) any later version.
This library is distributed in the hope that it will be useful, but
WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY
or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public
License for more details.
You should have received a copy of the GNU Lesser General Public
License along with this library; if not, write to the Free Software
Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301
USA
```

Also add information on how to contact you by electronic and paper mail.

You should also get your employer (if you work as a programmer) or your school, if any, to sign a "copyright disclaimer" for the library, if necessary. Here is a sample; alter the names:

Yoyodyne, Inc., hereby disclaims all copyright interest in the library `Frob' (a library for tweaking knobs) written by James Random Hacker.

<signature of Ty Coon>, 1 April 1990

Ty Coon, President of Vice

That's all there is to it!

Qt

Other Licenses Used in Qt

Qt contains some code that is not provided under the GNU General Public License (GPL), GNU Lesser General Public License (LGPL) or the Qt Commercial License Agreement, but rather under specific licenses from the original authors. Some pieces of code were developed by Nokia and others originated from third parties. This page lists the licenses used, names the authors, and links to the places where it is used.

Nokia gratefully acknowledges these and other contributions to Qt. We recommend that programs that use Qt also acknowledge these contributions, and quote these license statements in an appendix to the documentation.

See also: Licenses for Fonts Used in Qt for Embedded Linux

Copyright (c) 2001 Robert Penner All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
- Neither the name of the author nor the names of contributors may be used to endorse or promote products derived from this software without specific prior written permission.

Easing Equations by Robert Penner

OpenGL is a trademark of Silicon Graphics, Inc. in the United States and other countries.

QGLFormat

QGLFormat

QPlatformWindowFormat

QtScript is licensed under the GNU Library General Public License. Individual contributor names and copyright dates can be found inline in the code.

This library is free software; you can redistribute it and/or modify it under the terms of the GNU Library General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.

This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Library General Public License for more details.

You should have received a copy of the GNU Library General Public License along with this library; see the file COPYING.LIB. If not, write to the Free Software Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301, USA.

QtScript Module

QtScript Module

WebKit is licensed under the GNU Library General Public License. Individual contributor names and copyright dates can be found inline in the code.

This library is free software; you can redistribute it and/or modify it under the terms of the GNU Library General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.

This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Library General Public License for more details.

You should have received a copy of the GNU Library General Public License along with this library; see the file COPYING.LIB. If not, write to the Free Software Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301, USA.

WebKit in Qt

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR AND CONTRIBUTORS "AS IS". ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

- Shift-JIS Text Codec
- ISO 2022-JP (JIS) Text Codec
- EUC-JP Text Codec

Copyright (C) 1999-2000 Mizi Research Inc. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

EUC-KR Text Codec

Copyright (C) 2000 TurboLinux, Inc. Written by Justin Yu and Sean Chen.

Copyright (C) 2001, 2002 Turbolinux, Inc. Written by James Su.

Copyright (C) 2001, 2002 ThizLinux Laboratory Ltd. Written by Anthony Fok.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

GBK Text Codec

Big5-HKSCS Text Codec

THIS SOFTWARE IS PROVIDED BY THE AUTHOR AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Big5 Text Codec

Copyright (C) 2003-2004 immodule for Qt Project. All rights reserved.

This file is written to contribute to Nokia Corporation and/or its subsidiary(-ies) under their own license. You may use this file under your Qt license. Following description is copied from their original file headers. Contact immodule-qt@freedesktop.org if any conditions of this licensing are not clear to you.

QInputContextPlugin, QInputContext, QInputContextFactory

QInputContextPlugin

QInputContext

QInputContextFactory

Copyright (C) 2003-2006 Ben van Klinken and the CLucene Team Changes are Copyright (C) 2012 Nokia Corporation and/or its subsidiary(-ies).

This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version.

This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details.

You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301 USA

QtHelp Module

Copyright (C) 2004, 2005 Daniel M. Duley

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR ``AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHOR BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

QImage

Contributions to the Following QtGui Files: qapplication_cocoa_p.h, qapplication_mac.mm, qdesktopwidget_mac.mm qeventdispatcher_mac.mm qeventdispatcher_mac_p.h qmacincludes_mac.h qt_cocoa_helpers.mm qt_cocoa_helpers_p.h qwidget_mac.mm qsystemtrayicon_mac.mm

QAxServer Module, QAxContainer Module

QAxServer Module

QAxContainer Module

The qtmain Library

Implementation of the Recursive Shadow Casting Algorithm in Qt Designer

Contributions to the Following qt3to4 Files

Contributions to the Following qt3to4 Files: treewalker.h, treedump.cpp, treedump.h, treewalker.cpp

Torrent Example

QRegion

QDate::weekNumber()

This product includes software developed by the University of California, Berkeley and its contributors.

QLocale

TSCII Text Codec

QTestLib Manual

Drag and Drop

QtSvg Module

Phonon Module

The name and trademarks of copyright holders may NOT be used in advertising or publicity pertaining to the software without specific, written prior permission. Title to copyright in this software and any associated documentation will at all times remain with copyright holders.

QtXmlPatterns Module

Copyright (C) 2000-2004, International Business Machines Corporation and others. All Rights Reserved. Copyright (C) 2007 Apple Inc. All rights reserved.

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, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, provided that the above copyright notice(s) and this permission notice appear in all copies of the Software and that both the above copyright notice(s) and this permission notice appear in supporting documentation.

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 OF THIRD PARTY RIGHTS. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR HOLDERS INCLUDED IN THIS NOTICE BE LIABLE FOR ANY CLAIM, OR ANY SPECIAL INDIRECT OR CONSEQUENTIAL DAMAGES, OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

Except as contained in this notice, the name of a copyright holder shall not be used in advertising or otherwise to promote the sale, use or other dealings in this Software without prior written authorization of the copyright holder.

Parts of WebKit used by the QtWebKit module

Copyright (c) 1998 by Bjorn Reese <breese@imada.ou.dk>

Permission to use, copy, modify, and distribute this software for any purpose with or without fee is hereby granted, provided that the above copyright notice and this permission notice appear in all copies.

THIS SOFTWARE IS PROVIDED ``AS IS *AND WITHOUT ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, WITHOUT LIMITATION, THE IMPLIED WARRANTIES OF MERCHANTIBILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE AUTHORS AND CONTRIBUTORS ACCEPT NO RESPONSIBILITY IN ANY CONCEIVABLE MANNER.*

Parts of the QCrashHandler class

Parts of the FreeType projects have been modified and put into Qt for use in the painting subsystem. These files are ftraster.h, ftraster.c, ftgrays.h and ftgrays.c. The following modifications has been made to these files:

```
Renamed FT_ and ft_ symbols to QT_FT_ and qt_ft_ to avoid name conflicts
in qrasterdefs_p.h.
Removed parts of code not relevant when compiled with _STANDALONE_ defined.
Changed behavior in ftraster.c to follow X polygon filling rules.
Implemented support in ftraster.c for winding / odd even polygon fill
rules.
Replaced bitmap generation with span generation in ftraster.c.
Renamed ftraster.h as qblackraster_p.h.
Renamed ftraster.c as qblackraster.c.
Renamed ftgrays.h as qgrayraster_p.h.
Renamed ftgrays.c as qgrayraster.c.
```

See src/3rdparty/freetype/docs/FTL.txt and src/3rdparty/freetype/docs/GPL.txt for license details.

Copyright (c) 1985, 1986, 1987 X Consortium

Parts of the Q3PolygonScanner class used in Qt for Embedded Linux

Parts of the Q3PolygonScanner class used in Qt for Embedded Linux

Parts of the internal QKeyMapper class on X11 platforms

pnmscale.c - read a portable anymap and scale it

Parts of the internal QImageSmoothScaler

Parts of the internal QImageSmoothScaler::scale() function use code based on pnmscale.c by Jef Poskanzer.

jQuery JavaScript Library v1.3.2 http://jquery.com/


```
src/corelib/io/qurl.cpp
```

- Neither the name of Apple, Inc. nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of WebKit used by the QtWebKit module

Copyright (C) 1999 Serika Kurusugawa, All rights reserved.

Copyright (C) 1999-2000 Mizi Research Inc. All rights reserved.

Copyright (C) 2004, 2005 Daniel M. Duley

Copyright (C) 2000 Ming-Che Chuang

Copyright (C) 2001, 2002 James Su, Turbolinux Inc.

Copyright (C) 2002 WU Yi, HancomLinux Inc.

Copyright (C) 2001, 2002 Anthony Fok, ThizLinux Laboratory Ltd.

Copyright (c) 2000 Hans Petter Bieker. All rights reserved.

Copyright (C) 2001, 2002 ThizLinux Laboratory Ltd.

Copyright (C) 2001, 2002 Turbolinux, Inc.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR AND CONTRIBUTORS ``AS IS *AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.*

Parts of the codecs implemented by Qt

Copyright (c) 1992, 1993 The Regents of the University of California. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. 3. All advertising materials mentioning features or use of this software must display the following acknowledgment: This product includes software developed by the University of California, Berkeley and its contributors. 4. Neither the name of the University nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE REGENTS AND CONTRIBUTORS ``AS IS *AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.*

```
src/corelib/tools/qlocale.cpp
```

Copyright (C) 1997 - 2002, Makoto Matsumoto and Takuji Nishimura, All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
- Neither the name of the author nor the names of contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of WebKit used by the QtWebKit module

Copyright (C) 2005, 2007, 2008 by George Williams

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

The name of the author may not be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR “AS IS” AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHOR BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of the FreeType library

Copyright 2001, 2002 Catharon Productions Inc.

This file is part of the Catharon Typography Project and shall only be used, modified, and distributed under the terms of the Catharon Open Source License that should come with this file under the name `CatharonLicense.txt'. By continuing to use, modify, or distribute this file you indicate that you have read the license and understand and accept it fully.

Note that this license is compatible with the FreeType license.

Included in the build system of the FreeType library

See CatharonLicense.txt for more information

Copyright (C) 2006 Apple Computer, Inc. All rights reserved.
Copyright (C) 2007 Eric Seidel <eric@webkit.org>
Copyright (C) 2008 Kelvin W Sherlock (ksherlock@gmail.com)
Copyright (C) 2008 Alp Toker <alp@atoker.com>
Copyright (C) 2009 University of Szeged
Copyright (C) 2007 Alexey Proskuryakov (ap@nypop.com)
Copyright (C) 2009 Daniel Bates (dbates@intudata.com)
Copyright (C) 2008 Nikolas Zimmermann <zimmermann@kde.org>
Copyright (C) 2006 Michael Emmel mike.emmel@gmail.com
Copyright (C) 2007 Holger Hans Peter Freyther
Copyright (C) 2008 Collabora Ltd. All rights reserved.
Copyright (C) 2006 Dirk Mueller <mueller@kde.org>
Copyright (C) 2006 Zack Rusin <zack@kde.org>
Copyright (C) 2006 George Staikos <staikos@kde.org>
Copyright (C) 2006 Simon Hausmann <hausmann@kde.org>
Copyright (C) 2006 Rob Buis <buis@kde.org>
Copyright (C) 2008 Julien Chaffraix <jchaffraix@webkit.org>
Copyright (C) 2007 Henry Mason (hmason@mac.com)
Copyright (C) 1999 Lars Knoll (knoll@kde.org)
Copyright (C) 1999 Antti Koivisto (koivisto@kde.org)
Copyright (c) 2009 The Android Open Source Project
Copyright (C) 2008 Dirk Schulze <krit@webkit.org>
Copyright (C) 2008 Nokia Corporation and/or its subsidiary(-ies)
Copyright (C) 2008 Matt Lilek <webkit@mattlilek.com>
Copyright (C) 2009 280 North Inc. All Rights Reserved.
Copyright (C) 2009 Joseph Pecoraro
Copyright (C) 2008 Anthony Ricaud (rik24d@gmail.com)
Copyright (C) 2006 Samuel Weinig <sam.weinig@gmail.com>
Copyright (C) 2008 Christian Dywan <christian@imendio.com>
Copyright (C) 2006 Michael Emmel mike.emmel@gmail.com
Copyright (C) 2009 Holger Hans Peter Freyther
Copyright (C) 2008 Google Inc. All rights reserved.
Copyright (C) 2006 Friedemann Kleint <fkleint@trolltech.com>
Copyright (C) 2007 Nicholas Shanks <webkit@nickshanks.com>
Copyright (C) 2008 Collin Jackson <collinj@webkit.org>
Copyright (C) 2007 Staikos Computing Services Inc. <info@staikos.net>
Copyright (C) 2008 Kevin Ollivier <kevino@theolliviers.com> All Rights Reserved.
Copyright (C) 2005 Frerich Raabe <raabe@kde.org>
Copyright (C) 2005 Maksim Orlovich <maksim@kde.org>
Copyright (C) 2005, 2006 Kimmo Kinnunen <kimmo.t.kinnunen@nokia.com>.
Copyright (C) 2007-2009 Torch Mobile, Inc.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY APPLE COMPUTER, INC. “AS IS” AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL APPLE COMPUTER, INC. OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of WebKit used by the QtWebKit module

Copyright (C) 2009 University of Szeged All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY UNIVERSITY OF SZEGED “AS IS” AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL UNIVERSITY OF SZEGED OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of WebKit used by the QtWebKit module

Copyright (C) 2002 Michael Ringgaard. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. 3. Neither the name of the project nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

```
src/3rdparty/ce-compat/ce_time.c
```

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
- Neither the name of the University of Cambridge nor the name of Apple Inc. nor the names of their contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of WebKit used by the QtWebKit module

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
- Neither the name of the University of Cambridge nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Parts of WebKit used by the QtWebKit module

Parts of WebKit used by the QtWebKit module

Included in util/unicode/data, tests/auto/qtextboundaryfinder/data and tests/auto/qchar Parts of the makeqpf tool

HDF5

NETCDF

NETCDF

http://www.unidata.ucar.edu/software/netcdf/copyright.html

Ticpp

Ticpp

The MIT License (MIT)

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.

VTK

VTK is an open-source toolkit licensed under the BSD license.

Copyright (c) 1993-2008 Ken Martin, Will Schroeder, Bill Lorensen

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

Neither name of Ken Martin, Will Schroeder, or Bill Lorensen nor the names of any contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS *AS IS* AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHORS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Introduction to Setting up SMS

Installing

The installation wizard will guide through the installation process. There will be the option to install different parts of the SMS program including the executable files, tutorial files, documentation files, etc. If 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 starting 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 creating new projects. Modify the default settings for a new project by choosing **Save Settings** from the *File* menu. This will replace the default settings with the 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 and Network Locks .

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 .

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

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 running SMS in demo mode. The second, **Enable** is used to enable the program. This is described below.

Password

If using a password to enable SMS, send information to the vendor about the user's machine to get the password. There are several ways to send this information.

Register SMS with a Password

1) Start SMS and select the **Register…** button when the welcome screen appears.
    1) If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS. Then select the **Change Registration** button in the *Register SMS* dialog.

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.

See also: Register_SMS_with_a_Password_I.pdf

Hardware Locks

Follow the instructions received with the hardware lock to install the hardware lock and accompanying drivers. If hardware lock instructions were not received, 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 a 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 evaluating 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. A connection to a web utility will be made and a valid registration code will be sent via email. After receiving the registration code, enter it into the dialogue box, and select register. After evaluating SMS, please contact an SMS vendor with any questions or to purchase.

Related Topics

- SMS System Requirements
- Hardware Locks
- SMS Community Edition

Community Edition

Starting at version 12.2 there is a free version of SMS called "Community Edition". It is limited to include only the 2D mesh module and the SRH-2D model interface. It is also restricted in the size of the mesh and the number of simulations. Any size model can be imported, but if the mesh exceeds 5000 elements or the number of simulations is more than one, the project cannot be saved and a watermark is displayed in the Graphics Window. The community edition must still be registered using a license code which can be obtained via the internet from the *Registration Wizard* (*Help* | *Register* | *Change Registration* | **Get Community Edition License**).

Check to see if running in Community Edition mode by going to the *Registration* dialog. The size limits are displayed in the *About* dialog which is accessed through the **About** command in the *Help* menu.

The Community Edition capabilities are as follows:

Included Feature	**Limitations**
2D Mesh Module	Limited to one mesh.
Conceptual/Map Module Tool	
SRH-2D Interface	Limited to one simulation and cannot be saved it there is more than one mesh.
Import CAD/GIS Data	
Display Options	
Import Images	
Online Maps	
3D Viewing	
Lighting Model	

Technical support is not provided for the Community Edition.

Related Topics

- Registering SMS

Hardware Locks

Single User USB Hardware Lock

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

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

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 .
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

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

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

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 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 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. This trade off can be made 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 to determine your computer's hardware, operating system, and graphics card. To use the dxdiag 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 , HP or Compaq , Toshiba , Sony , 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
- ATI
- Diamond
- Elsa
- Intel
- Matrox
- nVidia
- S3 – Not all S3 card support OpenGL 1.5 which is required for all display options to be enabled.
- SIS – Not all SIS card support OpenGL 1.5 which is required for all display options to be enabled.
- VIA – 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 . 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 .

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 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.

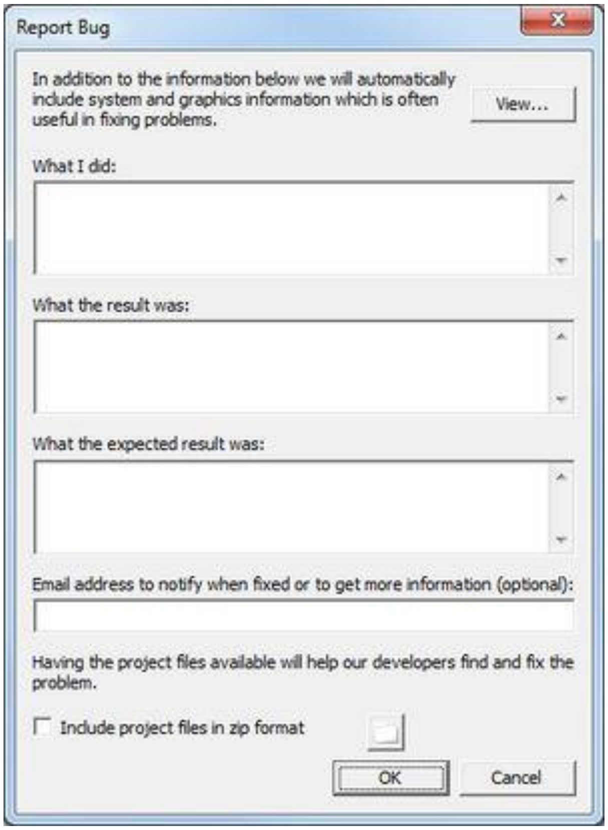
Contacting Support

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

External Links

- Aquaveo Technical Support

Report Bug



While Aquaveo and its developers work hard to keep problems in SMS to a minimum, some bugs or defects may occasionally surface. Reporting bugs helps Aquaveo and its developers resolve these issues. SMS must be connected to the internet in order to report a bug.

Some bugs in SMS are reported automatically to Aquaveo. This is done through bug trap software included in SMS. Automatic bug reports typically happen whenever SMS crashes.

Bugs can also be reported. It is advisable to first check Bugfixes SMS and the Aquaveo's Support Forums to see if the bug has already been reported. To report a new bug, go to the *Help* menu and select the **Report Bug** command. Activating this command will bring up the *Report Bug* dialog.

Report Bug Dialog

When reporting a bug, complete as many of the sections of the *Report Bug* dialog as possible. The more information the developers have the more likely the situation can be resolved in a timely manner.

- *System Information*

  SMS will deliver a text file that gives the configuration of SMS on the computer where the bug was reported. Clicking on the **View** button will bring up this text file showing what information is being sent.

- *What I did*

  Write in this section a brief description of the actions that were being done when the bug occurred.

- *What the result was*

  In this section, briefly explain the evidence of the bug in SMS.

- *What the expected result was*

  It is best to not assume that the developers will understand what should have appeared instead of the bug. Briefly state what should have occurred had the bug not occurred.

- *Email address*

  Provide an email address so that Aquaveo can follow up on the reported bug. Emails in bug reports are kept private and are not sold or used in marketing.

- *Project files*

  It can be helpful to include project files of the project being worked on when the bug occurred. This will help determine if the bug is related to SMS or if it is a problem with the project files. Include all files in a single ZIP file. Use the browser button to attach the ZIP file to the bug report.

After completing this dialog, pressing **OK** will send the information to Aquaveo, LLC. An internet connection must be available and active for this information to be received.

After Submitting a Bug

After a bug has been submitted, Aquaveo will review the reported bug. Whenever possible, the bug will be resolved as quickly as possible. There is no time frame for when a bug will be resolved—some are resolved within hours while others may not be resolved for many months.

In some cases, a bug cannot be resolved by Aquaveo, this is particularly true in cases where the bug has been caused by user error or if the bug is caused by third-party software used by SMS.

Not all users will be contacted once a bug is resolved. It is recommended to contact Aquaveo's technical support if there are further concerns.

Related Topics

- Bugfixes SMS
- Support

1.2. General Information

General Interface Features

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

Display

Editing the View with the Mouse

It can be easier to navigate with mouse controls. The three controls include:

- If the mouse has a middle button (or a mouse wheel), scroll the wheel to zoom in and out.
- If the mouse has a middle button (or a mouse wheel), hold it down and drag to pan the view.
- 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

- Projections
- File Formats

Keyboard Shortcuts

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

Standard Menu Shortcuts

Shortcuts for standard menu commands are listed in the table below.

Keyboard Shortcuts

Modifier	Key	Command
	F	*Display* | **Frame Image**
	SPACE	*Display* | **Refresh**
CTRL	*D*	*Display* | **Display Options**
	DELETE	*Edit* | **Delete**
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**
CTRL	*N*	*File* | **Delete All**
CTRL	*X*	*File* | **Exit**
	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**
SHIFT	*Q*	Toggle snapping on and off.

ALT Key Navigation

Commands and menus in SMS can be accessed without the use of a mouse by pressing the *ALT* and then pressing the corresponding key for the menu and command. The keys are underlined in SMS after pressing the *ALT* key. When pressing the key to menu command, the command should activate. If it does not activate, press the *ENTER* key.

For more information see the article ALT Key Navigation .

Related Topics

- Right-Click Menus
- Layout of the Graphical Interface

Publications

The following is a partial list of publications and reports related to the use of SMS. Please feel free to make additions to the list.

SMS Related Publications and Reports

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.

2015

- Pinho, José, Ferreira. R., Vieira, L., and Schwanenberg, D. (2015, January). "Comparison Between Two Hydrodynamic Models for Flooding Simulations at River Lima Basin" in *Water Resources Management* : Volume 29, Issue 2, pp. 431-444. [1]
- Vimeris, May Trio, Sugianto, D. H., and Wahyu Budi S (2015, January) "Kajian Refraksi-Difraksi dan Transformasi Penjalaran Gelombang Laut di Perairan Pantai Tapak Paderi Kota Bengkulu" in *Journal of Oceanography* volume 4, number 1, pp. 270-279. [2]
- Li, Honghai, Lin, L., Lu, C., Reed, C. W., and Shak, A. T. (2015, May). "Modeling study of Dana Point Harbor, California: littoral sediment transport around a semi-permeable breakwater" in *Journal of Ocean Engineering and Marine Energy* : Volume 1, Issue 2, pp. 181-192. [3]
- Siadatmousavia, S. Mostafa, Joseb, F., da Silvac, G. M. (2015, September). "Sensitivity of a third generation wave model to wind and boundary condition sources and model physics: A case study from the South Atlantic Ocean off Brazil coast" in *Computers & Geosciences* . DOI: 10.1016/j.cageo.2015.09.025. [4]
- Bilskie, Matthew V., Cogginb, D., Hagena, S. C., Medeiros, S. C. (2015, December). "Terrain-driven unstructured mesh development through semi-automatic vertical feature extraction" in *Advances in Water Resources* : Volume 86, Part A, pp. 102–118. [5]
- Marusic, Galina, Sandu, I., Vasilache, V., Filote, C., Sevcenco, N., and Cretu, M. (2015) "Modeling of Spacio-temporal Evolution of Fluoride Dispersion in 'River-type' Systems" in *Rev. Chim.* (Bucharest), volume 66, number 4, pp. 503-506. [6]

- 罗全胜, 曹明伟, and 谢龙 (2015) "三峡大坝175 m 蓄水运行后猪儿碛河段水动力条件变化分析" in *长江科学院院报*, volume 32, number 5, pp. 1-5. [7]

2014

- Stanev, Emil V., Zhang, J. Y., Grashorn, S., Koch, W., Pein, J., and Jacob, B. (2014, October). "Downscaling to Study Straits, Inlets and Tidal-Bays Dynamics: Unstructured Grid Model Simulations in the North and Baltic Seas." [8]
- Gerstner, N., Belzner, F., & Thorenz, C. (2014) "Simulation of Flood Scenarios with Combined 2D/3D Numerical Models." [9]
- Savant, G., & McAlpin, T. O. (2014). "Tidal Hydrodynamics in the Lower Columbia River Estuary through Depth Averaged Adaptive Hydraulics Modeling." *Journal of Engineering, 2014.* [10]
- Bilskie, M. V., Akhavian, R., and Hagen, S. (2014). "Bare Earth LiDAR to Gridded Topography for the Pascagoula River, MS: An Accuracy Assessment." *Bridges* , 10, 9780784412411-00018.
- Fulton, J. W. and Wagner, C. R. (2014). "Calibration of a Two-Dimensional Hydrodynamic Model for Parts of the Allegheny, Monongahela, and Ohio Rivers, Allegheny County, Pennsylvania" (No. SIR-2013-5145, p. 53). *United States Geological Survey*. [11]
- Huang, J. and Russell, K. (2014) "Two-Dimensional Numerical Simulation of a Grand Canyon Sandbar during a High Flow Experiment." *World Environmental and Water Resources Congress 2014:* pp. 1183-1190.

2013

- Li, H., Sanchez, A., Wu, W., & Reed, C. (2013, August). "Implementation of Structures in the CMS: Part 1, Rubble Mound" (No. ERDC/CHL-CHETN-IV-93). *Engineer Research and Development Center Vicksburg MS Coastal and Hydraulics Lab.* [12]
- Marusic, G., and Ciufudean, C. (2013, June). "Current state of research on water quality of Prut River." In *Proceedings of the 11th WSEAS International Conference on Environment, Ecosystems and Development (EED13)* , pp. 1-3. [13]
- Dobroliubov, S., Arkhipkin, V., Koltermann, P., Surkova, G., & Bublik, D. (2013, April). "High-resolution retrospective analysis of storm surges in the North Caspian Sea based on numerical simulations." In *EGU General Assembly Conference Abstracts* (Vol. 15, p. 7572).
- Lyubimova, T., Lepikhin, A., Parshakova, Y., Tiunov, A., Konovalov, V., & Shumilova, N. (2013, March). "Numerical modelling of admixture transport in a turbulent flow at river confluence." In *Journal of Physics: Conference Series* (Vol. 416, No. 1, p. 012028). IOP Publishing. [14]
- Petrescu, V., Ianus, L., & Sirbu, N. (2013) "Mathematical modeling of selective withdrawal from heliothermic stratified lakes. Case study" *U.P.B. Sci. Bull., Series D.* (Vol. 75, Issue 2) [15]
- McAlpin, T. O., Sharp, J. A., Scott, S. H., & Savant, G. (2013). "Habitat Restoration and Flood Control Protection in the Kissimmee River." *Wetlands* , 33(3), 551-560.

2012

- Marusic, G., Sandu, I., Moraru, V., Vasilache, V., Cretu, A., Filote, C., & Ciufudean, C. (2012, May) "Software for modeling spatial and temporal evolution of river-type systems." In *Proceedings of the 11th International Conference on Development and Application Systems* (pp. 17-19). [16]
- Li, H., & MacDonald, N. (2012, April). "Use of the PTM with CMS Quadtree Grids" (No. ERDC/CHL-CHETN-IV-82). *Engineer Research and Development Center Vicksburg MS Coastal and Hydraulics Lab.* [17]
- Ortiz, J. C., Salcedo, B., & Otero, L. J. (2012). "Investigating the Collapse of the Puerto Colombia Pier (Colombian Caribbean Coast) in March 2009: Methodology for the Reconstruction of Extreme Events and the Evaluation of their Impact on the Coastal Infrastructure." *Journal of Coastal Research* , 30(2), 291-300.
- Miot da Silva, G., Siadat Mousavi, S. M., & Jose, F. (2012). "Wave-driven sediment transport and beach-dune dynamics in a headland bay beach." *Marine Geology* , 323, 29-46.

- Wagner, D. M. (2012) "Two-Dimensional Simulation of the June 11, 2010, Flood of the Little Missouri River at Albert Pike Recreation Area, Ouachita National Forest, Arkansas." *Scientific Investigations Report 2012–5274*, USGS. [18]
- Jiang, J., Wang, P., Lung, W. S., Guo, L., & Li, M. (2012). "A GIS-based generic real-time risk assessment framework and decision tools for chemical spills in the river basin." *Journal of hazardous materials* , 227, 280-291.

2011

- Smith, T. J., Piotrowski, J. A., Young, N. C., Schnoebelen, D. J., & Weber, L. J. (2011). "Simulation of Spatial and Temporal Trends in Upper Mississippi River Physical Habitat." In *Proceedings of the 34th World Congress of the International Association for Hydro-Environment Research and Engineering: 33rd Hydrology and Water Resources Symposium and 10th Conference on Hydraulics in Water Engineering* (p. 3952). Engineers Australia.
- Massey, T. C., Anderson, M. E., Smith, J. M., Gomez, J., & Jones, R. (2011, September). "STWAVE: Steady-State Spectral Wave Model User's Manual for STWAVE, Version 6.0" (No. ERDC/CHL-SR-11-1). *Engineer Research and Development Center Vicksburg MS Coastal and Hydraulics Lab.* [19]
- Anderson, M. E., Lin, L., & Demirbilek, Z. (2011, April). "CMS-Wave Model: Part 4. An Automated Procedure for CMS-Wave in Resource-Demanding Applications" (No. ERDC/CHL-CHETN-IV-79). *Engineer Research and Development Center Vicksburg MS Coastal and Hydraulics Lab.* [20]
- Li, H., Lin, L., & Brown, M. E. (2011, January). "Applying Particle Tracking Model in the Coastal Modeling System" (No. ERDC/CHL-CHETN-IV-78). *Engineer Research and Development Center Vicksburg MS Coastal and Hydraulics Lab.* [21]
- Sasaki, J., Komatsu, Y., Matsumaru, R., & Wiyono, R. U. A. (2011). "Unstructured model investigation of 2004 Indian Ocean tsunami inundation in Banda Aceh, Indonesia." *J. Coast. Res* , 941-945. [22]

2010

- Pasquale, N., Perona, P., Schneider, P., Shrestha, J., Wombacher, A., & Burlando, P. (2010, November). "Modern comprehensive approach to monitor the morphodynamic evolution of restored river corridors." *Hydrology and Earth System Sciences Discussions,* 7(6), 8873-8912. [23]
- Glenn, J. S., & Bartell, E. M. (2010). "Evaluating Short-Circuiting Potential of Stormwater Ponds." In *World Environmental and Water Resources Congress 2010@ sChallenges of Change* (pp. 3942-3951). ASCE.

2009

- Cook, A., & Merwade, V. (2009). "Effect of topographic data, geometric configuration and modeling approach on flood inundation mapping." *Journal of Hydrology* , 377(1), 131-142.

2008

- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [24]
- Jul 2008 ERDC/CHL CHETN-IV-71 Particle Tracking Model (PTM) in the SMS 10: IV. Link to Coastal Modeling System [25]

2007

- SRH-2D Training Presentation [26]
- Aug 2007 ERDC/CHL CHETN-I-76 Modeling Nearshore Waves for Hurricane Katrina [27]
- Aug 2007 ERDC/CHL CHETN-I-75 Full-Plane STWAVE with Bottom Friction: II. Model Overview [28]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [29]
- May 2007 ERDC/CHL CHETN-I-73 Infra-Gravity Wave Input Toolbox (IGWT): User’s Guide [30]
- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User’s Guide [31]

- May 2007 ERDC/CHL CHETN-I-74 WABED Model in the SMS: Part 2. Graphical Interface [32]
- Lai, Y.G. and Bountry, J.A. (2007). "Numerical modeling study of levee setback alternatives for lower Dungeness River, Washington" [33]

2006

- Sep 2006 9th International Workshop On Wave Hindcasting and Forecasting Jane McKee Smith Modeling Nearshore Waves For Hurricane Katrina [34]
- Sep 2006 ERDC/CHL TR-06-20 PTM: Particle Tracking Model [35]
- 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 [36]
- Jul 2006 ERDC/CHL CHETN-III-73 Wave-Action Balance Equation Diffraction (WABED) Model: Tests of Wave Diffraction and Reflection at Inlets [37]
- 2006 Short Course Presentation FISC [38]
- Mar 2006 ERDC/CHL CHETN-I-71 Full Plane STWAVE: SMS Graphical Interface [39]
- 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) [40]
- Lai, Y.G. and Bountry, J.A. (2006). "Numerical hydraulic modeling and assessment in support of Elwha Surface Diversion Project." [41]
- Lai, Y.G., Holburn, E.R., and Bauer, T.R. (2006)."Analysis of sediment transport following removal of the Sandy River Delta Dam." [42]
- Bountry J.A. and Lai, Y.G. (2006)."Numerical modeling of flow hydraulics in support of the Savage Rapids Dam removal." [43]

2005

- Jul 2005 ERDC TN-DOER-D4 Particle Tracking Model (PTM) in the SMS: I. Graphical Interface [44]
- Jul 2005 ERDC TN-DOER-D5 Particle Tracking Model (PTM): II. Overview of Features and Capabilities [45]
- Jul 2005 ERDC TN-DOER-D6 Particle Tracking Model (PTM) in the SMS: III. Tutorial with Examples [46]
- May 2005 ERDC/CHL CHETN-I-70 BOUSS-2D Wave Model in SMS: 2. Tutorial with Examples [47]
- Mar 2005 ERDC/CHL CHETN-I-69 BOUSS-2D Wave Model in the SMS: 1. Graphical Interface [48]
- May 2005 ERDC/CHL CHETN-IV-63 Representation of Nonerodible (Hard) Bottom in Two-Dimensional Morphology Change Models [49]
- 2005 US-China Workshop Paper [50]

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 [51]
- Mar 2004 ERDC/CHL CHETN-I-68 How to Use CGWAVE with SMS:An Example for Tedious Creek Small Craft Harbor [52]

2003

- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D and STWAVE [53]
- Jun 2003 ERDC/CHL CHETN-I-67 Tedious Creek Small Craft Harbor:CGWAVE Model Comparisons Between Existing and Authorized Breakwater Configurations [54]

2002

- Jun 2002 ERDC/CHL CHETN-I-66 Grid Nesting with STWAVE [55]

- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [56]
- 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 [57]
- Mar 2002 ERDC/CHL CHETN-II-45 Wave Transmission at Detached Breakwaters for Shoreline Response Modeling [58]

2001

- Sep 2001 ERDC/CHL CHETN-I-64 Modeling Nearshor Wave Transformation with STWAVE [59]
- Sep 2001 ERDC/CHL TR-1-25 BOUSS-2D: A Boussinesq Wave Model for Coastal Regions and Harbors [60]
- Jun 2001 ERDC/CHL CHETN-IV-32 Leaky Internal-Barrier Normal-Flow Boundaries in the ADCIRC Coastal Hydrodynamics Code [61]
- 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 [62]

1999

- Dec 1999 Coastal Engineering Technical Note IV-21 Surface-Water Modeling System Tidal Constituents Toolbox for ADCIRC [63] [64]

1998

- Aug 1998 Technical Report CHL-98-xx CGWAVE: A Coastal Surface Water Wave Model of the Mild Slope Equation [65]

1990

- Mar 1990 CETN-II-21 Computer Program: Genesis Version 2 [66]

External Lists of Articles

- ADCIRC publications [67]
- Journal articles using SMS by Prof. Greg Pasternack, UC Davis [68]

1.3. 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.

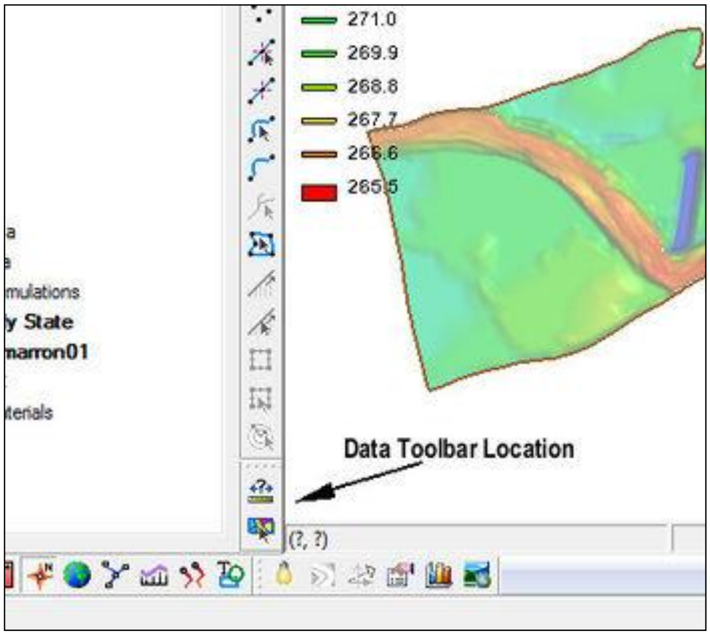
The interface to SMS has been designed in a modular fashion. Separate modules are used for each data type. As switching from one module to another, the available menus and tools change. Inside the modules, a numeric model can be associated with a mesh or grid. When that grid is active, the tools and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

The SMS screen includes several toolbars , edit fields , and menus . Some of these change when switching modules or numerical models . The main 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

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 toolbars, project explorer, time steps window and edit window are dockable windows. Dockable windows may be positioned anywhere on the screen.

Data Toolbar



The *Data Toolbar* contains tools to query or obtain data. Whether the toolbar appears at startup is set in the *Preferences* dialog.

The following tools are available:

Measure Tool

The **Measure Tool** is used to measure distances interactively. When clicking in the graphics window, a line will show the distance being measured and the *Coordinates Bar* will show the total numerical value. The units used to report the measured distance is specified on the *Toolbars* tab of the *Preferences* dialog—either the project unit as set in the *Projections* dialog or user-specified units.

Get Data Tool

The **Get Data Tool** is used to specify the location of interest for obtaining data using the import from web feature. Click and drag in the main graphics window to specify the area to be downloaded. The *Data Service Options* dialog will then appear and the web service or catalog can be selected for downloading the data. SMS will ask to save the data file before downloading the data and importing it into SMS. The data will appear in the GIS module.

This tool is not available if a display projection has not been specified. The tool will determine the data location based on the display projection.

Related Topics

- Layout of the Graphical Interface

Toolbars

There are several toolbars that can be displayed. Most toolbars are organized in the layout with set default positions. The *Preferences* dialog can be used to specify which toolbars are displayed. By clicking on the top of the toolbar, it can be detached from their default position and moved as floating palettes in the interface.

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 clicking 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

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 switching to a different module. Activating a module simply changes the set of available tools and menu commands. See Modules for more Information.

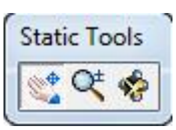
Data Toolbar

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

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 clicking in the Graphics Window depends on the current tool. The following describes the tools in the *Static Tool* palette.

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 not wanting to switch tools, pan by holding down the *F2* key and clicking and dragging with the mouse.
- If the mouse has a middle button (or a mouse wheel), 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 wanting to switch tools, zoom by holding down the *F3* key and clicking and dragging with the mouse.
- If the mouse has a middle button (or a mouse wheel), 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 not wanting to switch tools, 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

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 selecting 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 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 quickly modeling 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	Not editable	Not editable

				dataset		
Cartesian Grid Cellstring	N/A	N/A	N/A	N/A	N/A	N/A
Scatter Point	Editable for single selection	Editable for single selection	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 most interaction happens 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.

What data appears in the Graphics Window, and how each data type is formatted, can be controlled. 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, the help bar, at the bottom of the SMS application window. The second, the coordinates bar, is attached to the *Main Graphics Window* .

Help Bar

The status bar, also called the help bar, attached to the main application window shows help messages when the mouse hovers over a tool or an item in a dialog box. This bar is usually empty when not hovering over a tool or item and when not performing an action.

At times, it also may display a message in red text to prompt for specific actions, such as that shown in the figure below. Typically, messages in red appear when in the process of completing an action. Red messages disappear once the process has been completed.

Coordinates Bar

The second status bar, also called the coordinates bar, is attached to the Main Graphics Window. This bar is split into two separate panes.

The left shows the mouse coordinates when the model is in plan view. Coordinates are displayed in standard X,Y,Z coordinates. The units for these coordinates will match the units set in the object projection .

The right pane shows information for selected entities. Typically, this will show the number of objects selected. If one object is selected, it will give the ID for that object. When an arc is selected, the arc length and number of segments is shown. When multiple arcs are selected, the combined lenght of all selected arcs along with the combined number of segments is displayed. For polygons, the bar will show the area of the area of the polygon. When multiple polygons are selected, the combined area of all polygons will be shown. If two nodes or two vertices are selected, then the distance between the two points will be given.

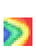
Related Topics

- 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** – Opens the *Lighting* tab of the *Display Options* dialog. See Lighting Options .
-  **Contour Options** – Opens the *Contours* tab of the *Display Options* dialog. See Contour Options .
-  **Vector Options** – Opens the *Vectors* tab of the *Display Options* dialog. See Vector Options .
-  **Get Module Info** – Opens the *Information* dialog. See Object Info .

- **Plot Wizard** – Starts the *Plot Wizard* . See Plot Wizard .
- **Dynamic Imagery** – Opens the *Get Online Maps* dialog. See Get Online Maps .

File Toolbar

- **Open** – Starts the *Open* browser. See Open .
- **Save Project** – Saves changes to the SMS project file. See Save Project .
- **Print** – Opens the *Print* dialog. See Print .
- **Delete** – Delete the selected items. If none are selected, delete all items.

Display Toolbar

- **Refresh** – Forces the display to update. See Refresh .
- **Frame** – Centers displayed data. See Frame .
- **Display Options** – Opens the *Display Options* dialog. See Display Options .
- **Plan View** – Change the view in the Graphics Window to a plan view .

Related Topics

- Layout of the Graphical Interface

Project Explorer

The Project Explorer (which is sometimes 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
- Scatter Module
- Map Module
- GIS 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. New folders can be created. It is possible to move datasets, solutions, and folders into 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 geometric object (mesh, grid, scatter set). These are displayed below the object in the Project Explorer and can be arranged into folders.

Geometric items can often be dragged to be linked to simulations or other objects.

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.

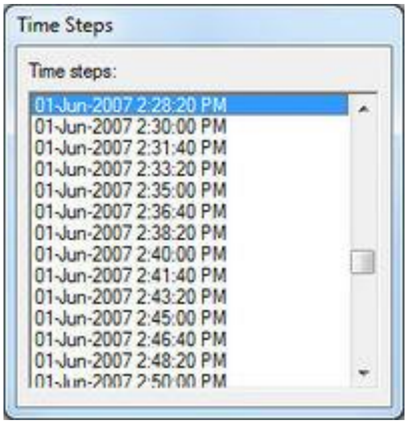
Right-Click Menus

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

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 changing the how time is displayed.
- Time Preferences – Opens the *SMS Preferences* dialog.

Related Topics

- Layout of the Graphical Interface

1.3.1. SMS Menus

SMS Menus

Menu commands in SMS are accessed through one of two ways. The first is through the menus located in menu bar. The second is by clicking the right mouse button to bring up a right-click menu.

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 standard menus, such as the *File* , *Edit* , and *Display* menus, 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
- 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 Mesh Model
- GenCade
- PTM
- STWAVE
- TABS
    - RMA2
    - RMA4
- TUFLOW

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

- Layout of the Graphical Interface
- Keyboard Shortcuts

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

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 with the option to specify another format type to use when reading in the file. Data from the file is added to the current data base and SMS updates the display.

Save Project

Used to save an SMS Project File (file extension *.sms). The first time this command is invoked, prompt will ask for a file name (unless a project file has already been opened in 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, prompt will ask for a file name (unless a project file has already been opened in 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...

Allows saving 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. Specify the *Save as type* in the *Save* dialog to select 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 map module must be active).

Delete All

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 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 by bring up the *View Data File* dialog. 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

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, there is the option of displaying the information to a separate window and echoing the information into a file.

The *Selection Information* dialog allows turning 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, prompt will ask for a file name. The *Display Echo Window* option opens a window where information is also displayed.

Save Settings

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

Print

Used to set printing options. The dialog contains three tabs:

Margins

This page allows specifying 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 defining 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 defining a scale to be added to the bottom of the printed image. If changing 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 the data is in. This could be feet, meters, lat/lon, etc. If, after the scale is set, zooming in or zooming 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

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 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.

Layout

Launches the *XMS Layout* dialog for defining a print layout.

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

Used to exit the program. If the data has not been saved, SMS gives a warning before exiting.

Related Topics

SMS Menus

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

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, SMS will not ask to confirm the deletion of selected entities.

Select All

Selects all items associated with the current selection tool.

Select With Poly

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, it is possible to 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

Brings up a submenu with the following options:

Material Type

Selects all items of the current selection tool of a specified material. This command opens the *Materials Data* dialog with a list of the defined materials and waits for a material type to be selected. This enables all nodes or elements that reference a specific material to be selected together.

Dataset Value

Opens a dialog that asks to specify a range. 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

Opens a dialog that asks to specify a range. 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

Opens a dialog that asks to specify a range. 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.

Ambiguous Gradient

Selects all elements in a mesh or cells in a grid where the directional flow is difficult to determine due to variation in the elevation at each node.

BC Type

Brings up a *Select Arc Type* dialog if the active coverage is a boundary conditions coverage (CMS-Flow , SRH-2D , TUFLOW , etc.). Allows selecting arcs by assigned boundary condition type.

Time Settings...

Opens the *Time Settings* dialog. For more information, see the Time Settings article.

Materials Data

See the Materials Data article.

Project Metadata

Allows defining 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

Sets program preferences. For more information, see the article Preferences .

Obsolete Commands

Confirm Deletions

By default, whenever a set of selected objects is about to be deleted, prompt will ask 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. Moved to Preferences dialog.

Current Coordinates

Tells SMS what coordinate system the data is to reference. SMS supports several different global systems as well as a user defined local system. Replaced with the Projections command.

Coordinate Conversions

Converts the current data from on coordinate system to another. For more information, see the Coordinate Conversions article. Replaced with the Reproject command.

Single Point Conversion

Opens a dialog which acts as a stand-alone coordinate converter. Specify a to and from coordinate system and a location. The location is converted to the new system within the dialog. Replaced with the Single Point Projection.. command.

Projection...

Brings up the *Current Projection* dialog. See Projections for more information. Moved to the *Display* menu.

Reproject...

Reprojecting means to convert data from one coordinate system to another. See Reproject for more information. Moved to the *Display* menu.

Single Point Projection...

Allows entering the XYZ coordinates for a point in one projection and seeing what the new coordinates would be if the point was reprojected to a different projection. See Projections for more information. Moved to the *Display* menu.

Related Topics

SMS Menus

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

Brings up the *Display Options* dialog. 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

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 All...

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

Single Point Projection...

Allows entering the XYZ coordinates for a point in one projection and seeing what the new coordinates would be if the point was reprojected to a different projection. See Projections for more information.

Related Topics

SMS Menus

Web Menu

The *Web* menu is one of the standard menus available regardless of the current module and model. The menu primarily provides ways to import data into SMS from online databases.

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 Maps ...

Brings up the *Get Online Maps* dialogue allowing selection of various online data. See Get Online Maps article for details.

Find Data

This sub-menu includes options to open to the Geo-Spatial Data Acquisition page on the XMS wiki. Each wiki article provides links to online databases where uses can find and download data.

Image

Brings up the GSDA Imagery article.

Bathymetry

Opens the article section GSDA Bathymetric Digital Elevation .

Coastline

Brings up the GSDA Oceanic Data article.

Tidal

Opens the article section GSDA Tidal Data .

Current

Opens the article section GSDA Current Data .

Wave

Opens the article section GSDA Wave Data .

Obsolete Commands

The following commands are no longer included in current versions of SMS. *Tidal Data* :

NOAA Hourly Verified...

NOAA 6-Minute Raw...

Related Topics

- SMS Menus
- Get Online Maps
- Import from Web

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.

Related Topics

SMS Menus

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:

SMS Help

Launches the Help File or brings up the XMS wiki depending which has been specified in the *Preferences* dialog.

Register

Brings up the *Register* window. Shows which components have been registered and changes can be made to the registration. See the article Registering SMS for more information.

About

Brings up the *About SMS* dialog that displays the version, build date, contact information, etc.

Report Bug

Allows for reporting issues with SMS. Activating this command will bring up the *Report Bug* dialog. See the article Report Bug for more information.

Check for Updates

Searches for updates to the current version. This command requires an internet connection to function. If updates are found, the option to install the latest version will be given.

Related Topics

SMS Menus

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** – Sets 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
- Scatter Module
- Map Module
- GIS 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 to 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

- SMS Menus

2. Functionalities

Breaklines

A breakline is a feature or polyline representing a ridge, thalweg, or other shape 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.

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
- Generate Contour Breaklines

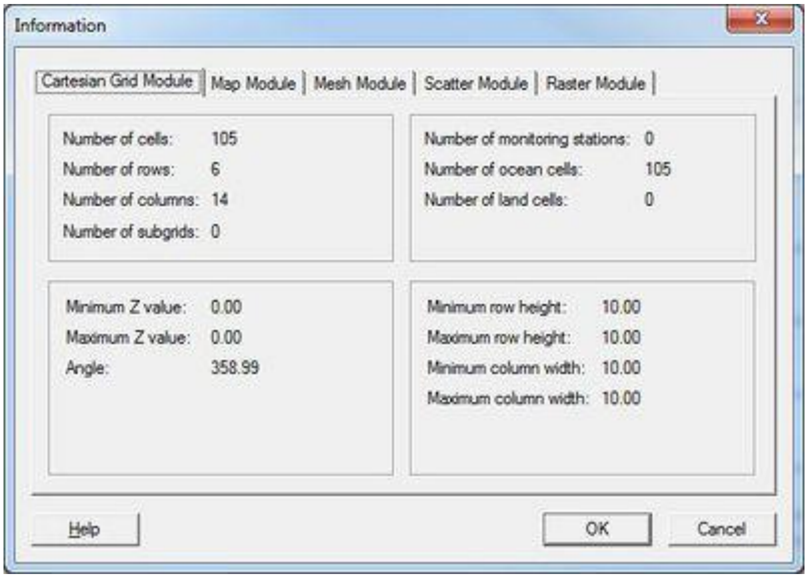
Object Info

Object information for a module can be found by clicking on the  **Get Module Info** macro or using the **Get Info** command in the *File* menu.

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

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:

  The drop down menu will list all available coverages. Select a coverage to view information.

- The type of the current coverage
- Number of points
- Number of nodes / vertices
- Number of arcs
- Number of polygons

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

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

  The drop down menu will list all available scatter sets. Select a scatter set to view information.

    - Scatter set ID
    - Number of points
    - Number of triangles

Raster Module Information

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

  The drop down menu will list all available raster sets. Select a raster set to view information.

    - 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 Data** . 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 Data** 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 Option* s dialog. These options include:
  - *Legend Title* – The name to be displayed on the legend.
  - *Legend Location* – The specification of where on the screen the legend will appear. Options include "Top Left Corner", "Bottom Left Corner", "Top Right Corner", "Bottom Right Corner", "Screen Location", and "World Location". The "Screen Location" and "World Location" will require the X location and *Y location* to be indicated. "World Location" also require the *Z location* to be specified.
  - *Font* – The font to be used in the legend. Clicking on the button will open the *Font* dialog. The down arrow can be used to specify a font color without opening the *Font* dialog.
  - *Size* – The size of each entry in the legend as indicated by *Width* and *Height*

  Note that only active materials are included in the legend.

Related Topics

- Area Property Coverage
- Edit Menu

Get Online Maps

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

Online maps are raster datasets that can contain imagery, elevation, or land use information. When online maps is available, right-click on each of the map items in the Project Explorer to convert them to static images that can be saved to the local hard drive. It is possible to convert or interpolate online maps containing elevation data to various elevation formats.

Note that online data sources are on external servers that the XMS software has no control over. The data may draw/export very slowly or become unavailable at any time. The XMS software has no control over this.

The **Advanced** button allows selecting 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 adding 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
- ASTER and SRTM data – USGS & NASA
- NLCD and CORINE (European) Land Cover data
- World Imagery More Info
- World Street Maps More Info
- World Topo Maps More Info
- MapQuest OpenStreetMap Worldwide Street Maps
- USA Topo Maps More Info
- 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. Do this if wanting 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.

Related Topics

- GIS Conversion and Editing

XMS Print Layout

The *Layout Editor* dialog, accessible by selecting *File* | **Layout...** , allows information from the XMS Main Graphics Window to be assembled and exported for use in reports and presentations.

Layout Editor Description

Below is a brief explanation of the macros, tools, and menus found in the *Layout Editor* dialog.

Layout Editor Menus

File menu items:

- **Import...** – Brings up the *Open* dialog, allowing importing of a saved layout (*.mwl).
- **Export...** – Brings up the *Save As* dialog, allowing savings of the layout in a user-specified folder.
- **Page Setup...** – Brings up the *Page Setup* dialog to allow setting of the paper size, orientation, and margins.
- **Print...** – Brings up the *Print* dialog, allowing printing of the layout to the desired printer.

View menu items:

- **Zoom in** – Magnifies for a closer, more-detailed view.
- **Zoom out** – Shrinks the view to be farther out and less-detailed.
- **Fit to screen** – Zooms in or out to show the entire page in the main layout window.
- **Show Margin** – Shows or hides the margin guide in the main layout window.
- *Toolbars* – Opens a submenu with the following options:
  - **Document** – Shows or hides the Document Toolbar .
  - **Tool** – Shows or hides the Tool Toolbar .
  - **Zoom** – Shows or hides the Zoom Toolbar .
- **Refresh** – Redraws the main layout window to the current settings.

Document Toolbar

- **Import...** – Brings up the *Open* dialog, allowing importing of a saved layout (*.mwl).
- **Export...** – Brings up the *Save As* dialog, allowing savings of the layout in a user-specified folder.
- **Print...** – Brings up the *Print* dialog, allowing printing of the layout to the desired printer.

Tool Toolbar

Each of these tools places objects in the main layout window by selecting the desired tool, then clicking and dragging to select the area where the object is to be placed. Once the mouse button is released after clicking and dragging, the desired object is immediately placed within the selected area.

- **Insert map** – Inserts the current content of the XMS Main Graphics Window into the selected area in the main layout window.
- **Insert north arrow** – Inserts a north arrow into the selected area in the main layout window.
- **Insert scale bar** – Inserts a scale bar into the selected area in the main layout window.
- **Insert text:** – Inserts a text box into the selected area in the main layout window.
- **Insert rectangle** – Inserts a rectangle into the selected area in the main layout window.
- **Insert bitmap** – Inserts a bitmap into the selected area in the main layout window.
- **Select tool** – Used to deselect the current tool.

- **Update Current View** – Updates the selected map object based on the current view from the XMS Main Graphics Window.
- **Zoom map to extent of data view** – Adjusts the size of the map image to fit within the extents of the map object box containing the image.

Zoom Toolbar

- **Zoom in** – Magnifies for a closer, more-detailed view.
- **Zoom out** – Shrinks the view to be farther out and less-detailed.
- **Fit to screen** – Zooms in or out to show the entire page in the main layout window.
- *Percentage* field – Changes the the given zoom level. This is populated with common percentages, but can also be manually changed by clicking in the white area and entering a positive integer.

When the *Layout Editor* dialog is closed, the layout is saved in its current state to a temporary folder. When the project is saved, the temporary layout file is saved as a part of the project file. If a project has a layout associated with it, that layout will be loaded into the *Layout Editor* dialog when it is opened. Otherwise, a blank layout will be shown.

Objects List

After inserting any object into the *Layout Editor* dialog, the object can be selected using the objects list section in the upper right portion of the window. The objects list displays all objects that have been inserted into the layout. Select an object to make it active by clicking directly on the listed object or by using the *Up* and *Down* keys on the keyboard to cycle through each object in the list. The display order of the objects can be adjusted using the **Up** or **Down** arrow buttons. Clicking the delete button will immediately remove the object from the object list.

Object Properties

The lower right portion of the objects list shows the properties of the active (or selected) object. The properties can be sorted using the following command buttons:

- **Categorize** – Places the properties in categories such as "Layout", "Map", and "Symbol". The options in each category relate to the category title.
- **Alphabetical** – Displays all properties alphabetically from A–Z without grouping them into categories.

General Properties

All objects have the following properties in common:

- *Location* – This field present two editable numbers. The first number is the *X* -axis location of the object. Increasing the *X* number moves the object to the right. The second number is the *Y* -axis location of the object. Increasing the *Y* number moves the object down. This property can be expanded to more clearly see the *X* and *Y* numbers.
- *Name* – This editable field shows the currently-selected object's name.
- *Size* – This field present two editable numbers. The first number is the width of the object. Increasing the *Width* number expands the object to the right and decreasing the number shrinks the the size of the object toward the left edge. The second number is the height of the object. Increasing the *Height* number expands the object down and decreasing the number shrinks the size of the object toward the top edge. This property can be expanded to more clearly see the *Width* and *Height* numbers.
- *Background* – Clicking on the ... button in this field, the *Polygon Symbolizer Properties* dialog is brought up.

Map

The **Insert map** tool is used to place the current image in the XMS Main Graphics Window into the *Layout Editor* dialog. The tool is used by clicking and dragging in the *Layout Editor* to define the area where the map will be displayed. The editor will automatically resize and scale the image to the defined area.

The display of the editor can further be edited by adjusting the map properties. The map objects use general properties and the follow map object specific properties:

- *Scale* – Adjusts the size of the image inside of the map object. Increasing this value with decrease the size of the image. Decreasing the value will increase the image size.
- *Bearing* – The degree away from North of the original image in the XMS Main Graphics Window.
- *Dip* – The angle of descent relative to a horizontal plane of the image in the XMS Main Graphics Window.
- *Height* – The original height of the image in the XMS Main Graphics Window.
- *Width* – The original width of the image in the XMS Main Graphics Window.

North Arrow

The **Insert North Arrow** tool inserts in the selected location a north arrow associated with a specific map object. When a map gets rotated, the north arrow changes its rotation angle based on the map's bearing angle.

The following are the properties of the north arrow:

- *Color* – Contains a drop-down list of colors. Selecting a color will change the color of the north arrow object.
- *Map* – This field contains a dropdown list of all map objects in the Layout Editor. Selecting a map object assigns the north arrow to that map. When assigned, the arrow will rotate to match show north on the map.
- *North Arrow Style* – A drop-down list of north arrow styles. Styles include: "Default", "Black Arrow", "Center Star", "Triangle N", "Triangle Hat", and "Arrow N".

- *Rotation* – Changes the rotation of the north arrow. Normally, the arrow rotation matches the map bearing.

Scale Bar

The **Insert Scale Bar** tool inserts in the selected location a scale bar associated with a specific map object. The scale of the scale bar and the map controls are user-defined.

The following are the properties of the scale bar:

- *Break Before Zero* – Default is "False". If set to "True", the center of the scale bare will be "0" and the bar will extend out equal lengths on each side of the "0".
- *Color* – Contains a drop-down list of colors. Selecting a color will change the color of the scale bar object.
- *Font* – Displays the current font style and size for text in the scale bar object. Clicking on the ... button in this field brings up a *Font* dialog where the font type, style, size, script, and any effects can be selected.
- *Map* – This field contains a drop-down list of all map objects in the Layout Editor. Selecting a map object assigns the scale bar to that map. When assigned, the scale bar will adjust to fit the scale of the map.
- *Number of Breaks* – Indicates how many interval marks will be displayed on the scale bar. Requires a minimum value of "1".
- *Text Hint* – Rasterization options for how the text will be rendered. Options include: "System Default", "Single Bit Per Pixel Grid Fit", "Single Bit Per Pixel", "Anti-Alias Grid Fit", "Anti-Alias", and "Clear Type Grid Fit".
- *Unit* – A drop-down menu where the scale bar measurement units can be selected. Units options include: "Kilometers", "Meters", "Centimeters", "Millimeters", "Miles,", Yards", "Feet", and "Inches".
- *Unit text* – Indicates how the units are referred to on the scale bar. Currently, this is not updated when the *Units* are changed.

The scale bar doesn't support geographic degrees. Of the dip is not equal to 0° or 90°, the scale bar doesn't show any scales.

Text

The **Insert Text** tool inserts text in the selected location.

The following are the properties of the inserted text:

- *Color* – Contains a drop-down list of colors. Selecting a color will change the color of the text.
- *Continent Alignment* – Determines the horizontal and vertical alignment of the text inside of the text object. The default is to align the text to the upper left side of the text object.
- *Font* – Displays the current font style and size for the text. Clicking on the ... button in this field brings up a *Font* dialog where the font type, style, size, script, and any effects can be selected.
- *Text* – Field where the text displayed in the text object can be edited.
- *Text Hint* – Rasterization options for how the text will be rendered. Options include: "System Default", "Single Bit Per Pixel Grid Fit", "Single Bit Per Pixel", "Anti-Alias Grid Fit", "Anti-Alias", and "Clear Type Grid Fit".

Rectangle

The **Insert Rectangle** tool creates a rectangle in the location specified. Rectangles created in the *Layout Editor* use general properties only and do not have specific properties.

Image

The **Insert Bitmap** tool brings up a dialog allowing a bitmap image to be imported into the layout. This is often used for images such as a logo file.

The following are the properties of the inserted bitmap:

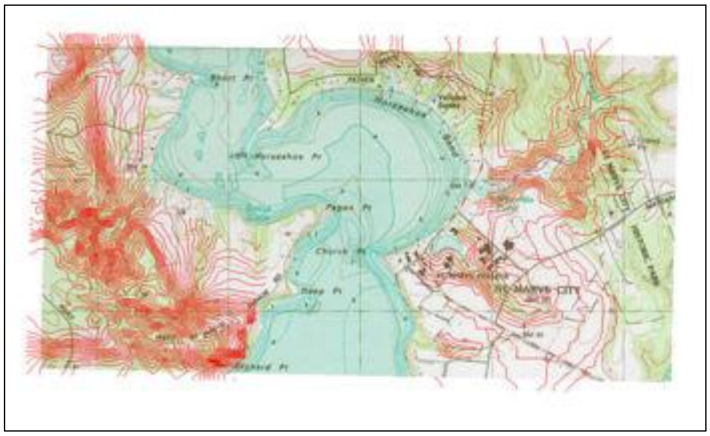
- *Brightness* – Increases how light the image appears. Value can be from 0–255 with the default value being "0" (no additional lightening).
- *Contrast* – Increases the difference in color in the image. Value can be from 0–255 with the default value being "0" (no additional contrast).
- *File Name* – Displays the pathname and file name of the imported image.

- *Preserve Aspect Ratio* – A drop-down menu with the options "True" or "False". Selecting "True" will keep the image constrained to the dimensions of the imported image. Selecting "False" will change the image dimensions to fit the image object box.

Related Topics

- GMS File Menu
- SMS File Menu

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.

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. If wanting to turn off the display of all CAD data, then uncheck the box next to the CAD folder .

Creating CAD data from SMS data

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 *Clean Option* dialog will appear just after using the command to help resolve and potential problems in the conversion.

The feature objects are added to a new 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 CAD 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 CAD 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

- 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 (flow rate) 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 in extracting data from two or three dimensional objects, model verification, and defining one-dimensional river models. By selecting *Display* | **Plot Wizard** , a step by step process is given to create a variety of plots. The wizard can also be reached by using the **Plot Wizard** macro. The list of currently available plots in the *Plot Wizard* is shown below. All of the plots listed below are associated with observation coverages .

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 guide through creating a plot.

Step 1 – Select 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 indicate if appropriate data exists 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 editing 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.

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:

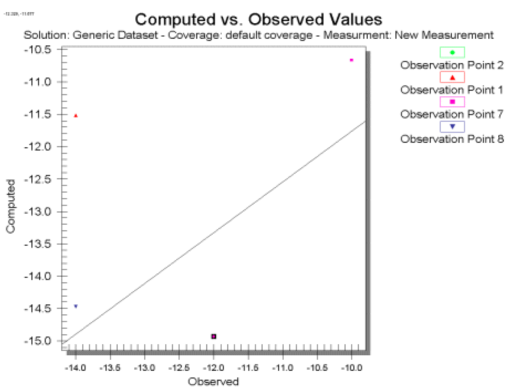
1) Select the cells of interest from the spreadsheet
2) Press *CTRL* + *C* – cell contents are now in the clipboard
3) Select the paste destination
4) Press *CTRL* + *V*

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 below.



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

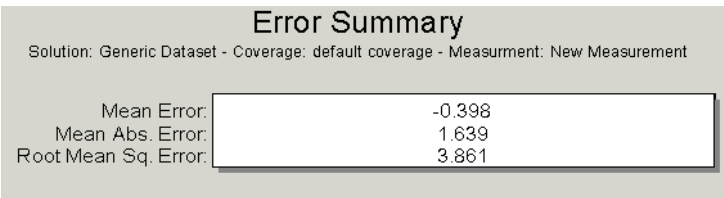
An *Error Summary* plot is used to display a text listing of the mean error, mean absolute error, and root mean squared error for a dataset and the observed values associated with a mesh or grid on observation points in the Observation coverage . The errors shown are the mean errors for all observation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

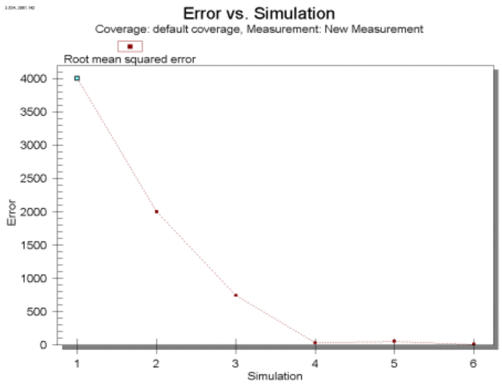
- *Coverage* – Displays the name of the coverage where the current data for the plot is coming from.
- *Measurement* – This is the name of the current measurement, created in the *Feature Objects* | **Attributes** dialog, being plotted.
- *Feature Objects* – Displays which feature object is utilized 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.

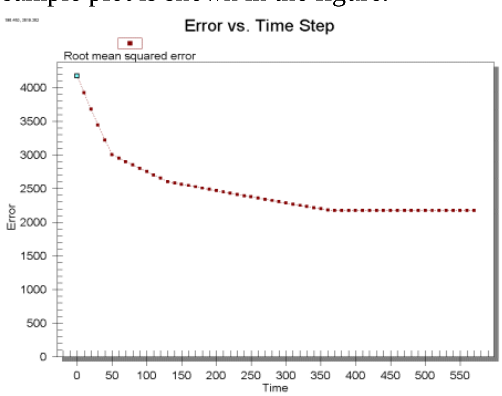
- *Solutions* – This box lists all available solutions for the simulation. Select the desired solution to use its datasets in the plot.
- *Move Up/Move Down* – SMS initially shows the solutions in the order they were opened. However, this is not necessarily the order in which they were run. To change 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 most 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 below.

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.

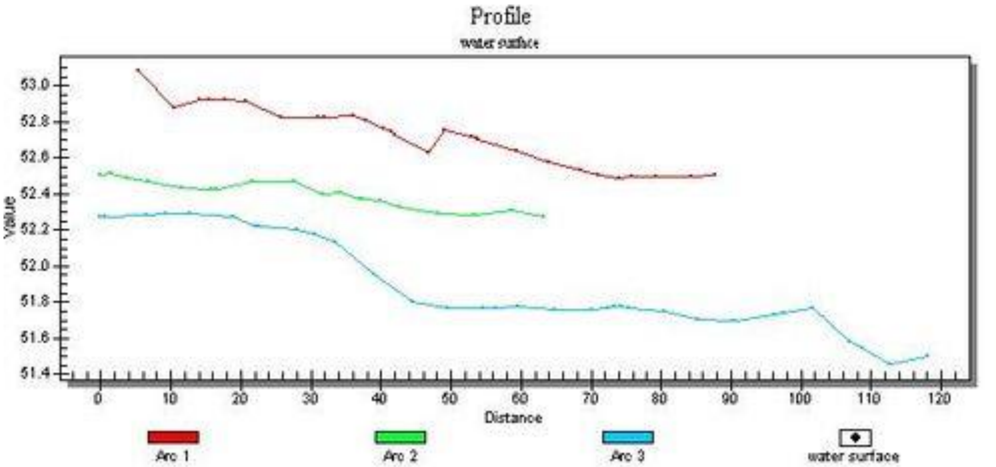
- *Specified dataset(s)* – Profile plot points are based on the specified dataset(s). Datasets from different modules can be specified.

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

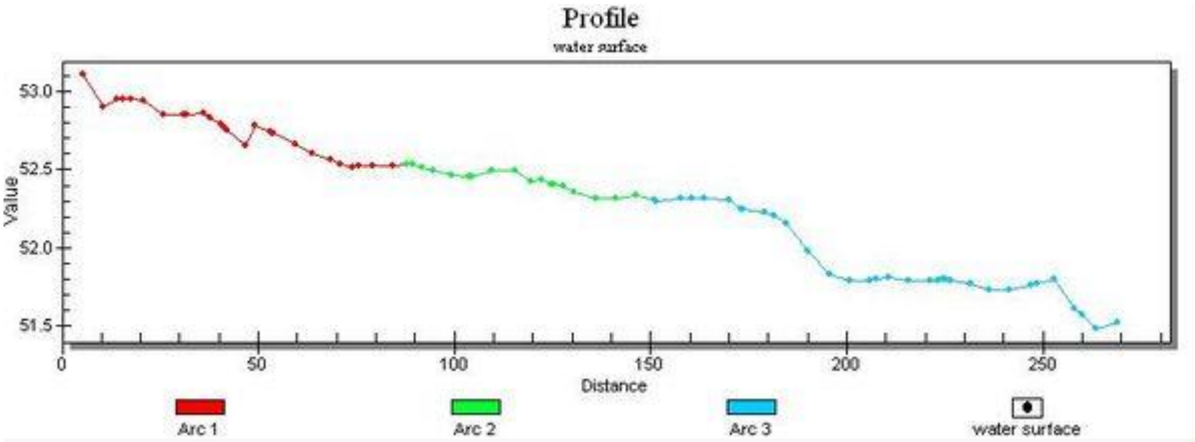
There are two ways in which an observation profile can be created when multiple arcs will be graphed. Multiple arcs can be graphed on a single plot was so they appear in separate segments as shown below.



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 editing many plot properties. The plot options are organized onto the following tabs in the dialog:

- **General Tab** – Plot title, border style, viewing style, font size, numeric precision, and grid line style can be changed.
  - Main Title – Allows changing the title of the plot.
  - Subtitle – Allows changing the subtitle of the plot.
  - 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 exporting 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.

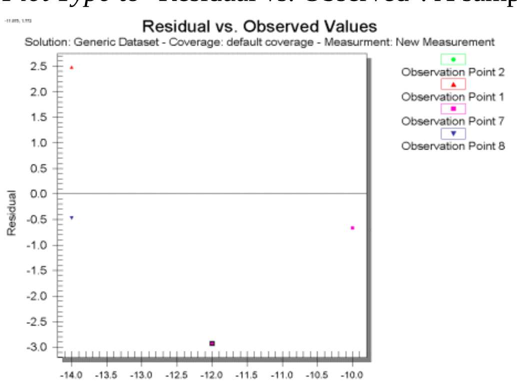
To open the *Profile Customization* dialog, select *Display* | **Plot Display Options** or right-click on the plot and select **Display Options** .

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.

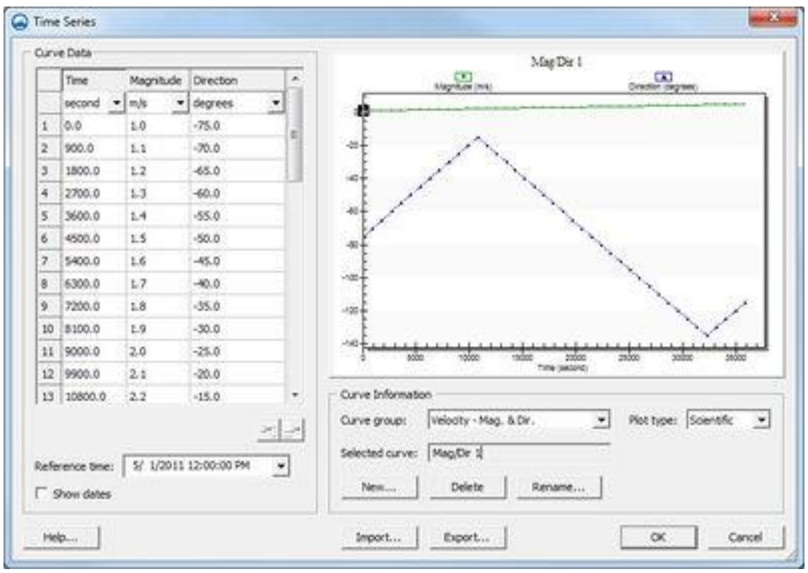
1) Times series can support more than 2 columns of data.
2) The time series group can be specified (i.e. velocity, xy), when available.
3) When using times, the reference time can be specified.
4) The units of each column can be specified (from a list) and the column data will be converted when switching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

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"

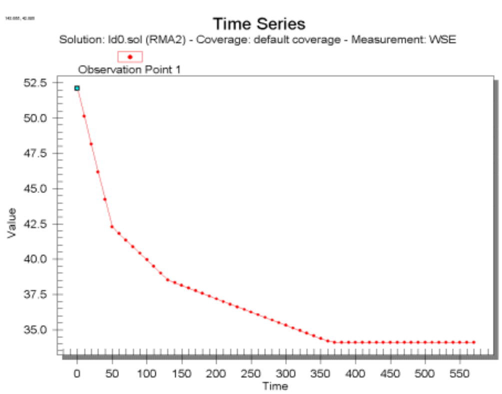
```
0.0                     1.0              -75.0
900.0                   1.1              -70.0
1800.0                  1.2              -65.0
2700.0                  1.3              -60.0
3600.0                  1.4              -55.0
4500.0                  1.5              -50.0
5400.0                  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* , define 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 displaying the calibration curve defined for each point. If there is no 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

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.

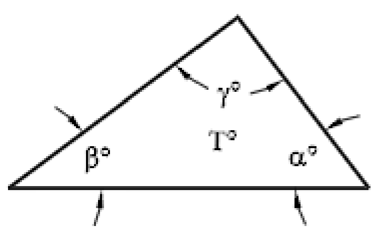
The plot includes the ARR region (defined below), a point for each element in the mesh, and three contour lines (0.3 in red, 0.45 in yellow and 0.6 in green) of the currently selected element quality measure (also defined below). As a general rule, elements with quality lower than 0.3 should be reviewed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

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.

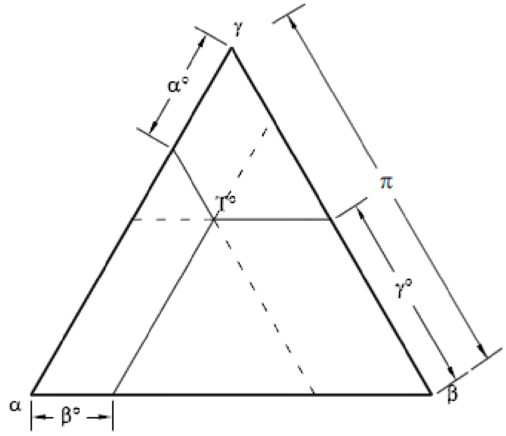
Once the mesh is edited in any way, update the ARR plot by right-clicking in the plot and selecting **Refresh** . Until this is done, the plot will continue to reflect the mesh that existed when it was generated (or most recently refreshed).

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:



Plot these three angles into an equilateral triangle.



If ordering the three angles so that α > β > γ as shown below, 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:

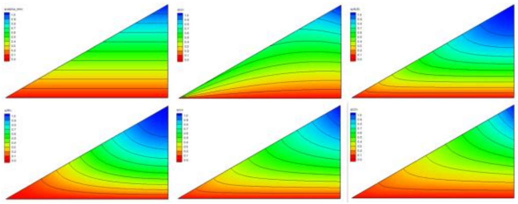
- The minimum interior angle α_{min} (γ from previous figure).
- The lengths of edges.
- The triangle area.
- The inner and outer radius.
- The minimum distance through the triangle (h_{min}).

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:

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

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

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

- 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 controlling 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 specifies whether the flow trace should be created for a steady state or dynamic system. Below this, 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. Start by defining 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. 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.

Animation Playback

Once a new animation has been generated, SMS launches the AVI player and plays the animation. The speed of playback can be adjusted using the *Speed* scroll bar. The maximum speed depends on the speed of the computer and the size of the image being animated. The smaller the image, the faster the maximum playback speed.

Related Topics

- Visualization

Film Loop Display Options

This page of the *Film Loop Setup* wizard allows setting up the film loop clock options. 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, additional options can be set for the clock position, progress bar and clock style.

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

AVI Codecs

Starting in SMS 10.0, it s possible to choose what Codec to use to create a AVI movie. SMS will search the 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 to choose from. 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, first download ffdshow. If the computer has installed the 32-bit version of SMS, ffdshow was installed unless during installation the option was turned off.

If ffdshow is not installed, download it. To download ffdshow, go to http://ffdshow-tryout.sourceforge.net .

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

Using FFDSHOW

There are two steps to using the ffdshow codecs for animations generated in SMS. First, use the ffdshow program to set the options for the video encoder. Secondly, 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 is desired. (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 is used needs to be on the computer that will be playing the animations. If wanting to move the AVI's to a different computer, make sure that the target computer has ffdshow. Otherwise, it's necessary to download ffdshow to play the animations.

Alternate Video Players

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

Related Topics

- Animations

Film Loop Drogue Plot Options

This page of the *Film Loop Setup* wizard allows setting 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. It may be necessary to experiment with this maximum value to get something that is desirable 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 it's not desirable to have a tail, then set the fade time to zero (0.0).

It's possible to specify the background to be either a solid color or an image that has been already opened and registered. If there isn't 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.

Particle Report

When creating a drogue plot, SMS can write out the following statistical information for each particle:

- The particle’s starting and ending location.
- The start and end times for each particle in decimals days.
- The total distance it traveled over the length of the animation.
- The minimum, maximum, and average velocity at which it traveled.

Be aware that when a particle leaves the domain, it can no longer be tracked so the ending location will be the point at which it left the domain. Click the **Browse** button to set the name of the report file.

Related Topics

- Animations

Film Loop Flow Trace Options

This page of the *Film Loop Setup* wizard allows setting 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, 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

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

- AVI, or audio video interleave (*.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/ for information on obtaining Google Earth©.

The *General Options* page allows specifying the following:

- File name for each video animation file type being exported
- Film loop type

- **Transient Data Animation** – To use this option, there must be 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, there must be an available vector dataset, such as velocity. This animation randomly distributes particles throughout the domain and shows their path through time.

- **Drogue Plot** – To use this option, there must be an available vector dataset, such as velocity, and there must have been 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 animation of 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. See the article Play AVI Application .

Related Topics

- Animations

Film Loop Multiple Views

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

Initial View – This section has options for the bearing and dip at the start of the animation run.

- *Bearing* – Degrees rotated away from North for the initial start of the animation.
- *Dip* – The angle above or below horizontal elevation for the initial start of the animation.

Animation – This field shows a list of all views that will be used during the animation run. Views will be displayed in the order presented in this field from the top down.

- **Add View** – Brings up the *Add View* dialog where additional the bearing, dip, and duration can be set for an additional view during the animation run.
- **Remove** – Deletes the selected view from the animation list.

- *Display Bearing/Dip in each frame* – When this option is checked on, the bearing and dip will displayed for each frame during the animation run. This is checked on by default.

Add View Dialog

This dialog will created addition views during the animation run. The following options are available for each view:

- *Bearing* – Degrees rotated away from North for this view in the animation run.
- *Dip* – The angle above or below horizontal elevation this view in the animation run.
- *Number of steps to this view* – How many animation frames will use this view.

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 – Setting the *Run Simulation From* option requires selecting a start time from the available time steps.
- Film loop duration – Setting the *Run Simulation To* option requires selecting an ending time from the available time steps. This time step must be after the start time.
- *Specify Number of Frames* – Indicates the number of frames in the final animation. The default number equals the number of time steps in the transient data. The number of animation frames can be different from the number of time steps to make the animation shorter or longer.
- *Specify Time Step Size* – Changes the time step increments to the indicated size.

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. Projections

Projections

Related Versions	
GMS	v9.1
SMS	v11.1
WMS	v9.1
version note	

Projection refers to a map projection like UTM . In XMS software, a projections are associated with the project and individual data objects.

Starting with SMS 11.1 and GMS 9.1, the XMS software works on the concept of a "Display Projection". This is the projection being worked in. Each geometric object loaded into the XMS package, such as a Scatterset, DEM, grid or mesh, also has an associated projection. These two projections may not be the same. If the two projections are compatible (i.e. it is a viable option to convert from one projection to the other), the XMS package will convert the data from the object projection to the display projection, just for display purposes. This is referred to as "Project on the fly". The data itself maintains the values associated with the object projection so when the XMS package saves a project, the data files are not modified.

Alternatively, data can be reprojected from one projection to another, actually changing the data values that will be saved as part of a project. This is done by right-clicking on the geometric object in the project explorer (data tree) and selecting the **Reproject...** command. The SMS package includes a feature to reproject all data from whatever projection the data is currently in to a single projection using the **Reproject All...** command in the *Display* menu.

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

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

Display Projection

The display projection, or the projection currently associated with the project, can be specified via the *Display* | **Projection** menu command. This setting controls how the XMS application displays (or interprets) data. Data defining objects with a specified projection are converted to the display projection (if it is different from the object projection) for display purposes only. This is referred to as "projection on the fly". The data saved to files as part of a project, or exported for a simulation are exported in the object projection. Display projection only affects the display.

Objects without a specified projection are assumed by the XMS application to be referenced to the display projection.

The display projection is saved as part of a project file. Specify a display projection as part of the system settings for new sessions/projects.

When a data object is read into an XMS application, for which no data has yet been loaded and no projection has been specified, and the data object has its own projection, the display projection is reset to the object projection. This allows defining a working projection simply by loading data that uses that projection (and has a projection definition such as a *.prj file to define it.)

No Projection (Previously Local Projection)

Many numerical models work in local or model space, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). XMS software allows for this using a Display projection set to local or no projection option. This is standard practice when building a numerical model of a flume test. The units of the model are also specified as part of the projection. If the display projection is in this mode, no global projections are allowed on individual objects.

(Note: when the display projection is set to *No Projection* or *Local Projection* , the data may still be referenced to a projection. The display projection can be changed to reflect that projection if desired.)

Global Projection

Data referenced to a global projection can be easily correlated and used with other applications that utilize projections including GIS and CAD. When the display projection is specified as a global projection, the XMS application can export georeferenced images, shapefiles, and KMZ files that may be directly imported to other applications.

Selecting the *Global Projection* option will automatically bring up the *Select Projection* dialog where a global projection can be chosen. If the *Select Projection* dialog does not automatically appear, or desiring to change the current global projection, then the **Select Projection** button in the projection dialog can be used to access the dialog.

Object Projection

Each geometric object loaded into a session can have an associated projection. When an object is loaded from a file, the XMS application looks for a projection either in the object data file or in an associated *.prj file. If no projection is found, the object is left with no projection or floating. In this case, the object is assumed to be related to the display projection, regardless of what that projection is. The object projection can be specified by right-clicking on the object in the project explorer and selecting the **Projection...** command. The default projection displayed in the dialog that appears is the object's projection if it has one, and no projection otherwise. In the case of no projection, the display projection is filled in as the default global projection should that option be selected.

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 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 projection or coordinate system.

There are four basic reprojection tasks:

- Reproject on the fly, which just displays all data in a specified projection without changing the base values
- Reprojecting the entire project from one system to another
- Reprojecting one geometric object (i.e. mesh or grid) from one coordinate system to another
- Single point reprojection, which allows entering 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.

Reproject on the Fly

When data from multiple projections are loaded into and XMS application, without a defined projection, they do not overlay and the display shows data clusters at two distinct locations. With project on the fly, if the data object has a defined projection (such as a *.prj file), this data would be reprojected on the fly to the display projection.

If data does not line up due to incorrect or incomplete projection specification, specify different object projections to attempt to align the data correctly. Object projection is specified by right-clicking on the object in the project explorer.

Reproject everything

Reprojecting everything can be done by selecting the *Display* | **Reproject All...** menu command. This will convert all the data loaded into the XMS application from the object projection(s) to a specified projection. This operation brings up a dialog which allows specifying the desired projection. The default value is the display projection currently specified for the project.

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 and sececting **Reproject...** . If the object does not have a specified projection, this command is not available. It can be accessed by selection the **Projection...** command for the object in the same right-click menu and defining a projection for the object.

When the object has a projection, this command reprojects from one projection to another. The command brings up a dialog with a "from" projection specified on the left and a "to" projection specified on the right. The "from" projection is defaulted to the object projection. The "to" projection is defaulted to the display projection.

Single Point Reprojection

Single Point Reprojection command is found in the *Display* menu and allows entering 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 allows creating a feature point at the new location.

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) library which supports hundreds of standard projections.

Related Topics

- Projection Dialogs

Projection Dialogs

There are two main projection dialogs used in SMS, GMS, and WMS: The *Display Projection* dialog, and the *Object Projection* dialog. From each of these, the *Select Projection* dialog can be accessed. More detailed information about each projection and the information in these dialogs can be found in the Projections article.

Display Projection Dialog

The *Display Projection* dialog contains settings which are applied to the project as a whole.

Horizontal section

The *Horizontal* section of the dialog has two options available via radio buttons:

- *No projection* – This option doesn't set a projection, and only allows adjusting the horizontal *Units* used in the project. The available units include:
  - " U. S. Survey Feet ". Equal to $^{1200}/_{3937}$ meters, approximately 0.3048006096 meters.
  - " International Feet ". Equal to 0.3048 meters.
  - " Meters ". Equal to the distance traveled by light in vacuum within $^{1}/_{299792458}$ of a second.
  - " Inches ". Equal to $^{1}/_{39.37}$ of a meter.

- " Centimeters ". Equal to $^1/_{100}$ of a meter.

- *Global projection* – Clicking on the **Set Projection** button allows more specific projections to be set. These are listed below in the Select Projection Dialog section.

Vertical section

The Vertical section has two drop-down boxes:

- *Projection* , giving the following options:
- "Local"
  - " NGVD 29 (US) "
  - " NAVD 88 (US) "
- *Units* , giving the following options:
  - "U. S. Survey Feet"
  - "International Feet"
  - "Meters"
  - "Inches"
  - "Centimeters"

Object Projection Dialog

The *Object Projection* dialog is the same as the *Display Projection* dialog, but only applies to one specific object (e.g., a coverage or a mesh). It can be accessed by selecting the object in the Project Explorer, then right-clicking on it and selecting **Projection...** from the menu.

Select Projection Dialog

The *Select Projection* dialog is accessed through the **Set Projection...** button in either the *Display Projection* dialog or the *Object Projection* dialog. It allows setting global projections for the project or for a specific object. It contains a single *Projection* tab with several options and sections.

To the right of the *Projection* drop-down are three buttons:

- **Load From File...** : Allows a projection to be loaded from an external file. The accepted file formats are PRJ and SPR (PRJ is preferable).
- **Save to File...** : Allows the projection information set in this dialog to be saved as a PRJ file.
- **Init from EPSG...** : Opens a dialog where an EPSG Projection Code can be entered.

Datum

Datums in *italics* are not actual options. The available options for that entry are listed in the indented list directly below it.

Create New Datum Dialog

The *Create New Datum* dialog (right) allows for the creation of a new datum if the available datums (the list above) do not cover the needs of the project. There are a number of different fields and options in this dialog. These are detailed below:

- *Datum Name* : Enter the desired name for the new datum.
- *Abbreviation (Optional)* : An optional field for a shortened version of the *Datum Name* .
- *Prime Meridian (Degrees)* : Enter the prime meridian .
- *Ellipsoid (Spheroid) Selection* : This section consists of a drop-down list and two buttons:
    - **Add Ellipsoid...** : This button brings up the *Custom Ellipsoid Setup* dialog. It has the following fields:
        - *Ellipsoid Name* : Enter the desired name of the ellipsoid.
        - *Semi-Major Axis (meters)* : Enter the length of the semi-major axis .
        - The two radio button options are:
            - *Use Semi-Minor Axis of* : Enter the length of the the semi-minor axis .
            - *Use flattening of* : Enter a number representing the flattening .
    - **Edit Ellipsoid...** : Allows editing of a custom ellipsoid (one created with the **Add Ellipsoid...** button).
- *Datum Transformation Method* : This section consists of four radio button options:
    - *3-parameter (Molodensky) Transformation* : Uses the Molodensky transformation method.
    - *7-parameter (Bursa-Wolfe) Transformation (Position Vector Rotation)* : Uses seven parameters : "three translations, three rotations, and a scale correction factor".
    - *7-parameter (Bursa-Wolfe) Transformation (Coordinate Frame Rotation)* : Variation of the above.
    - *Custom Shift Based on Control Point File* : Selecting this options brings up an *Open* dialog which allows choosing a Control Point File with the necessary data to set the datum transformation method. This file is a text file with entries (one per line) in the following format:

```
deg_longitude_in_new_datum,deg_latitude_in_new_datum,deg_longitude_in_WGS84,deg_
latitude_in_WGS84
```

- *Shifts to WGS84 (meters)* : This section allows the *X Shift* , *Y Shift* , and *Z Shift* to be manually set for the new datum.
- *Rotation to WGS84* : This section allows setting the X, Y, and Z in for the new datum by selecting from the *Units* drop-down:
  - arc-seconds
  - radians
  - micro-radians
- *Scale Correction to WGS84 (parts per million)* : This section allows setting the *Scale (ppm)* for the new datum.

Planar Units

The planar unit is simply the measuring format used in the projection. Select the appropriate one from the list.

Parameters

In the *Parameters* section, the *Attribute* and *Value* columns contain information specific to the *Projection* and *Zone* selected.

Related Topics

- Projections

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).

Related Topics

- Projections
- Projection Dialogs

2.3.a. UTM Coordinate System

UTM Coordinate System

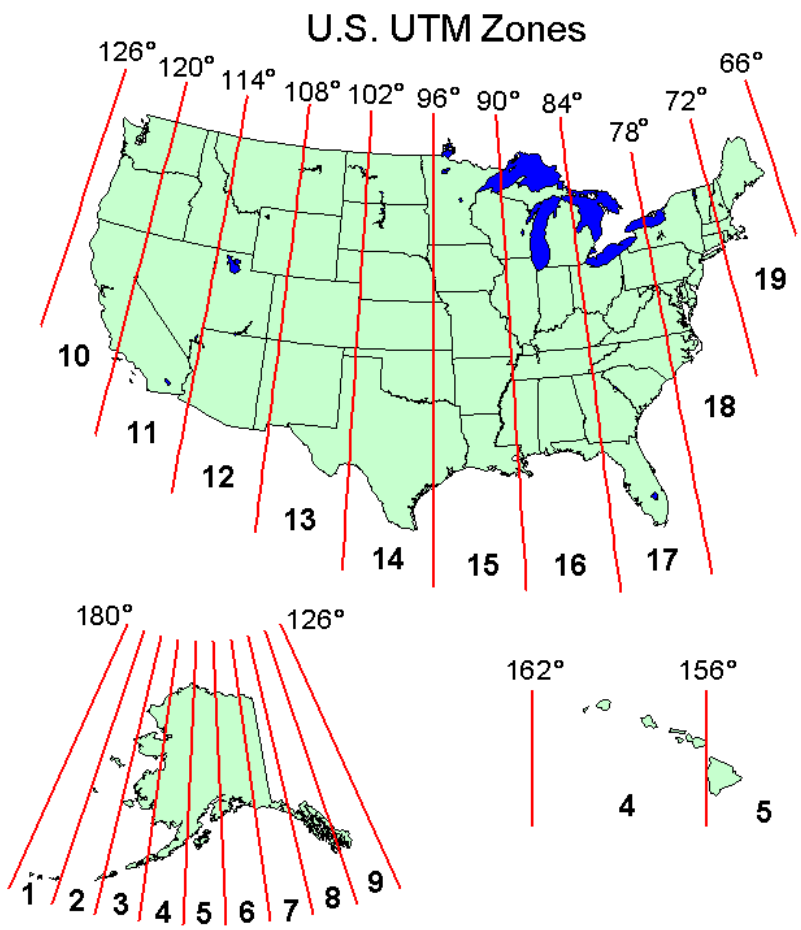
A UTM (Universal Transverse Mercator) system is a world-wide system defined in meters. The world is divided into 60 zones, 6 degrees of longitude, running from 84°N to 80°S latitude. Supported UTM systems include:

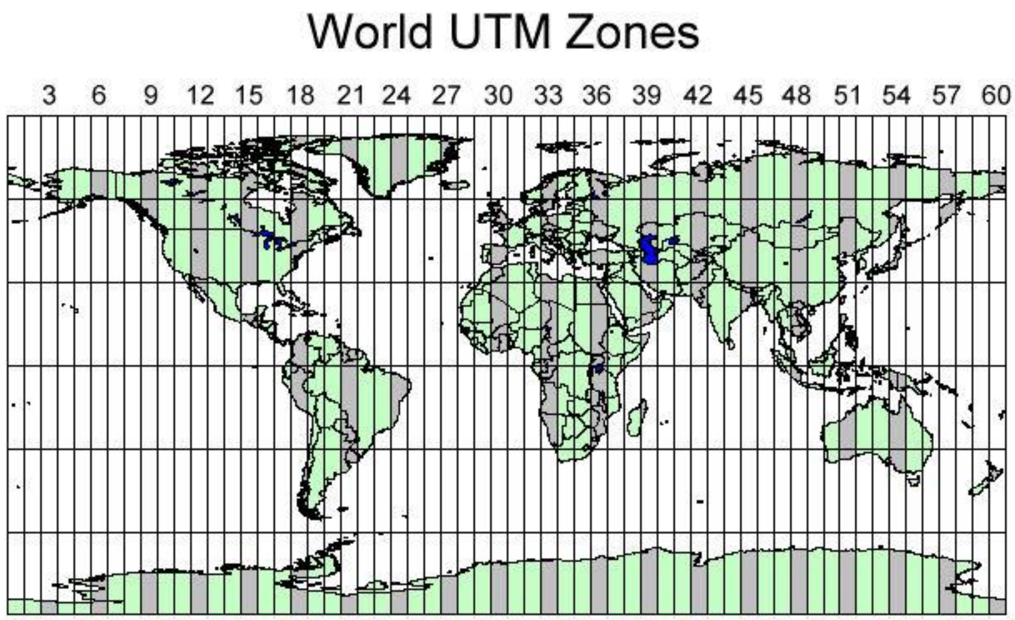
- NAD (North American Datum) 1927 and NAD 1983
- HPGN (High Precision Geodetic Network, now known as HARN - High Accuracy Precision Network)

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

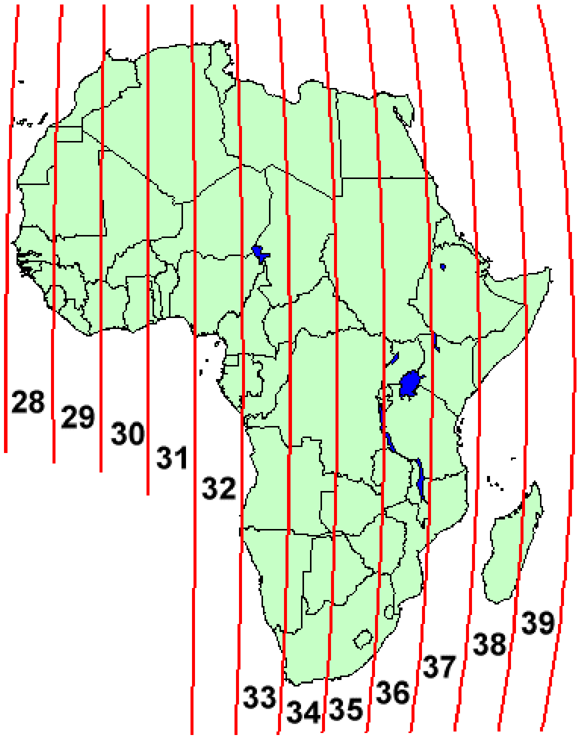
- North America
- South America
- Africa
- Asia
- Europe
- Australia

Related Topics

- Projections
- UTM Coordinates

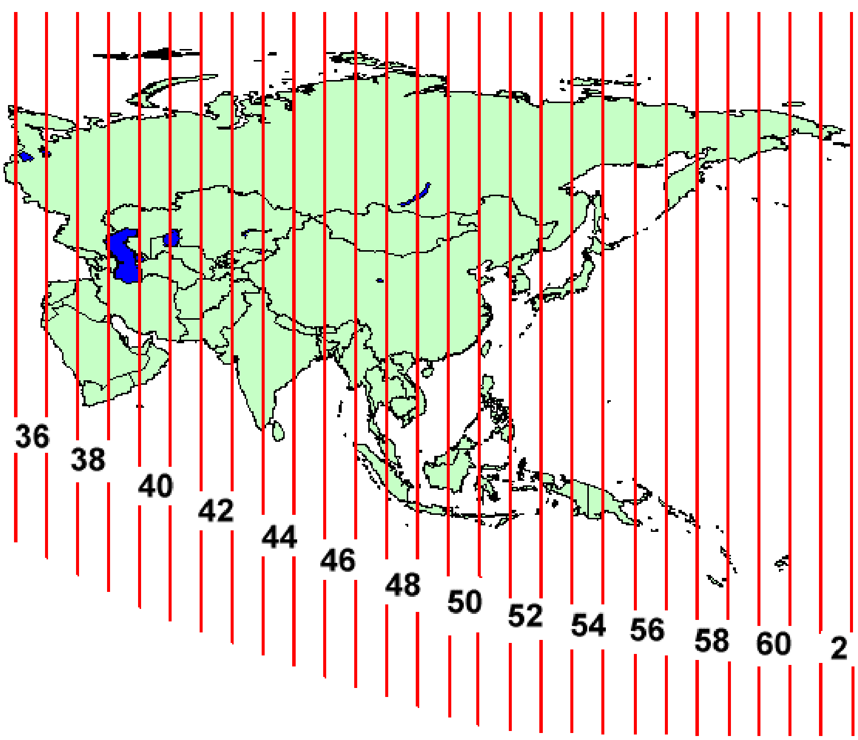
UTM Africa

The figure below shows the UTM zones for Africa.



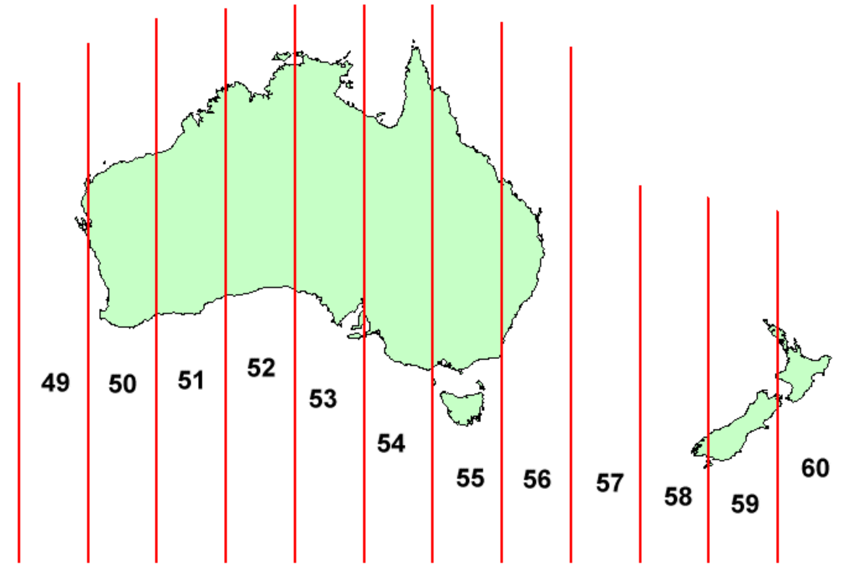
UTM Asia

The figure below shows the UTM zones for Asia.



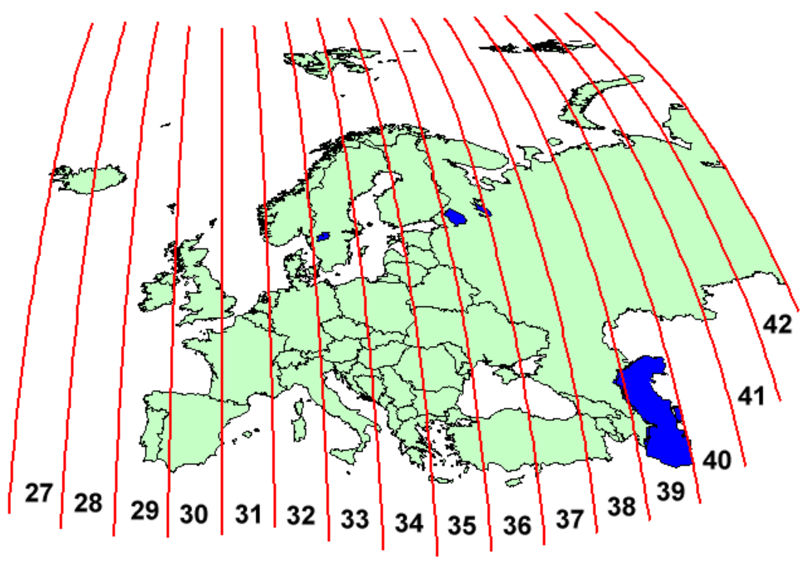
UTM Australia

The figure below shows the UTM zones for Australia.



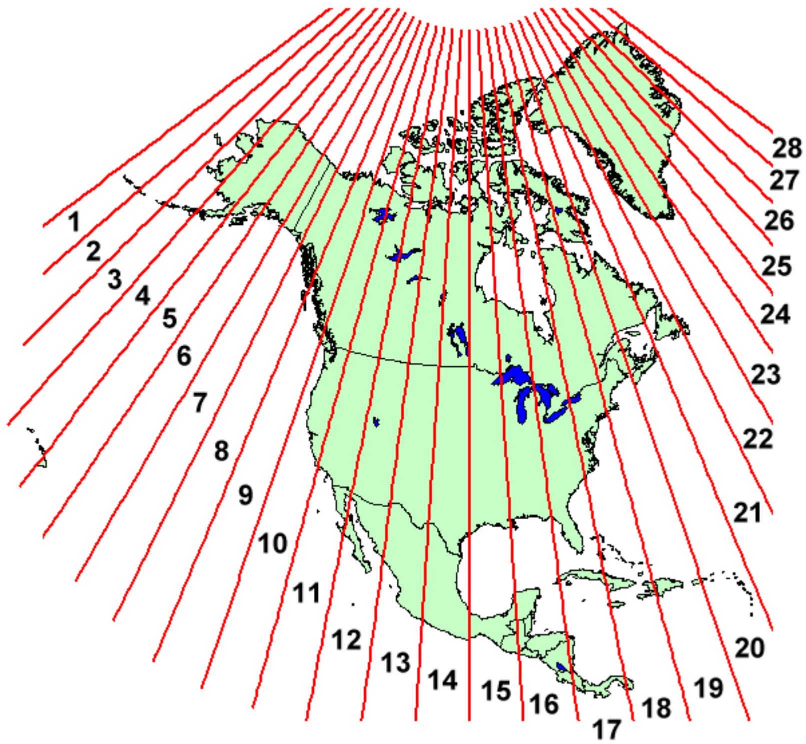
UTM Europe

The figure below shows the UTM zones for Europe.



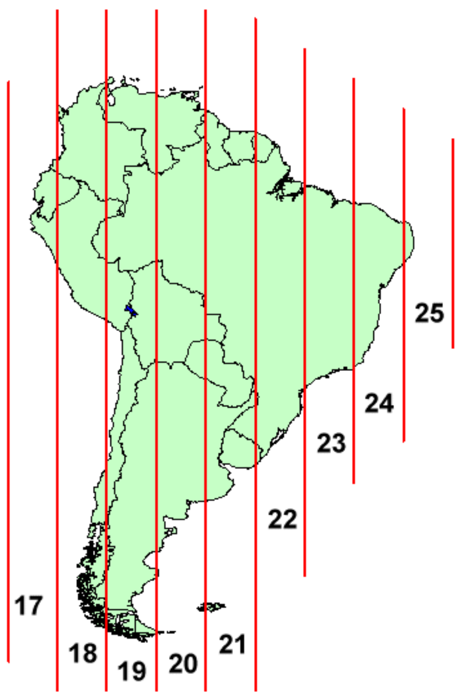
UTM North America

The figure below shows the UTM zones for North America.



UTM South American

The figure below shows the UTM zones for South America.



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

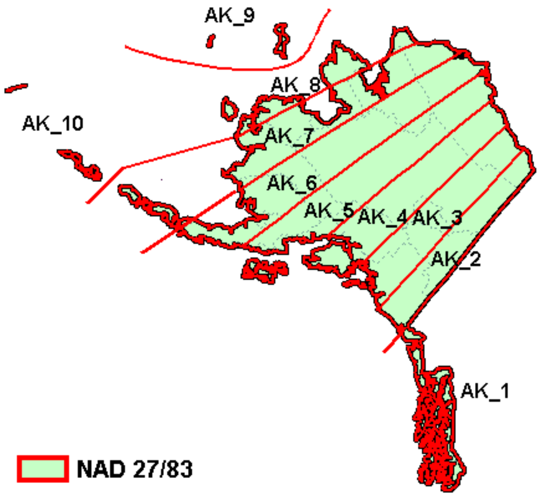
- South East
- Southwest
- Virgina Area

Related Topics

Projections

Alaska State Plane

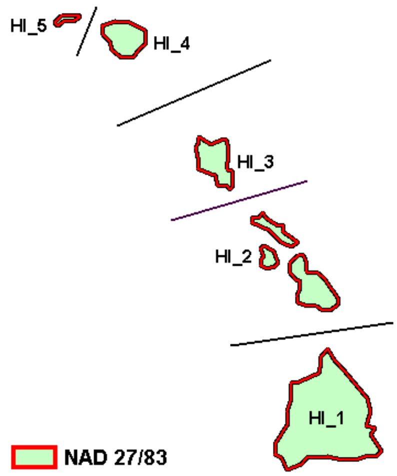
The figure below shows the state plane zones for the Alaska area.



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

The figure below shows the state plane zones for the Hawaii area.

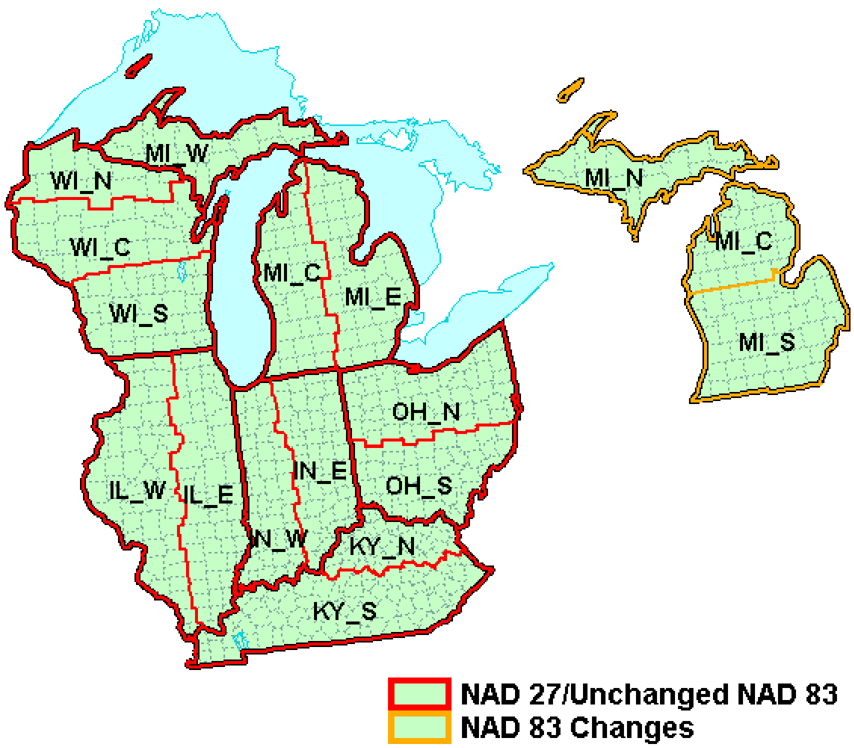


NAD 27 / 83

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

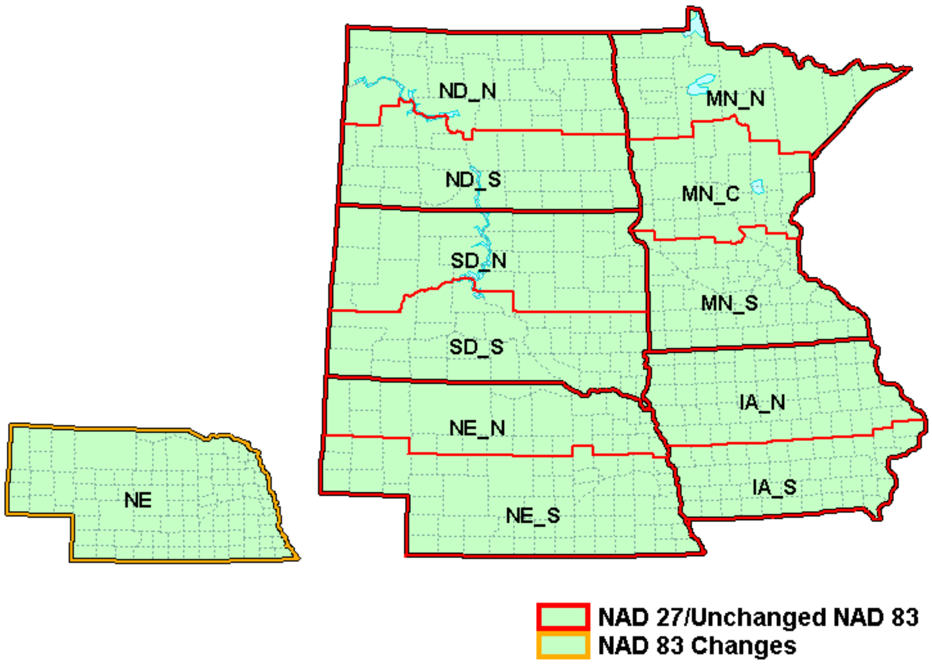
The figure below shows the state plane zones for the Mideast area.



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

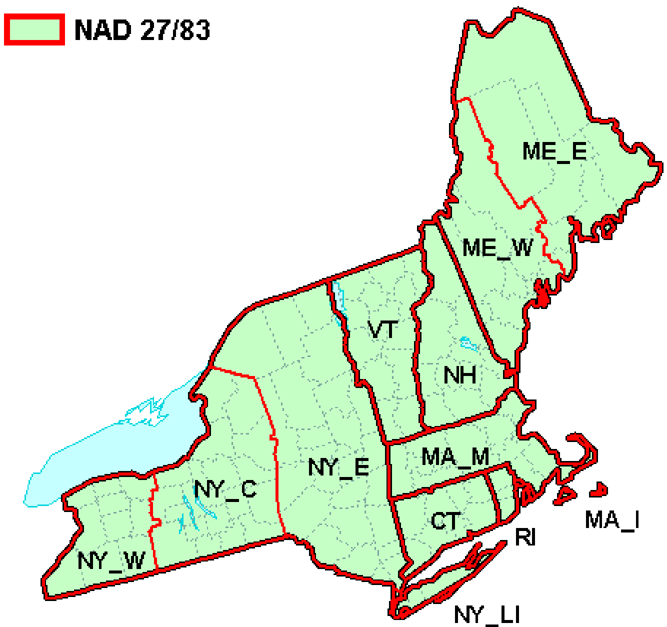
The figure below shows the state plane zones for the Midwest area.



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

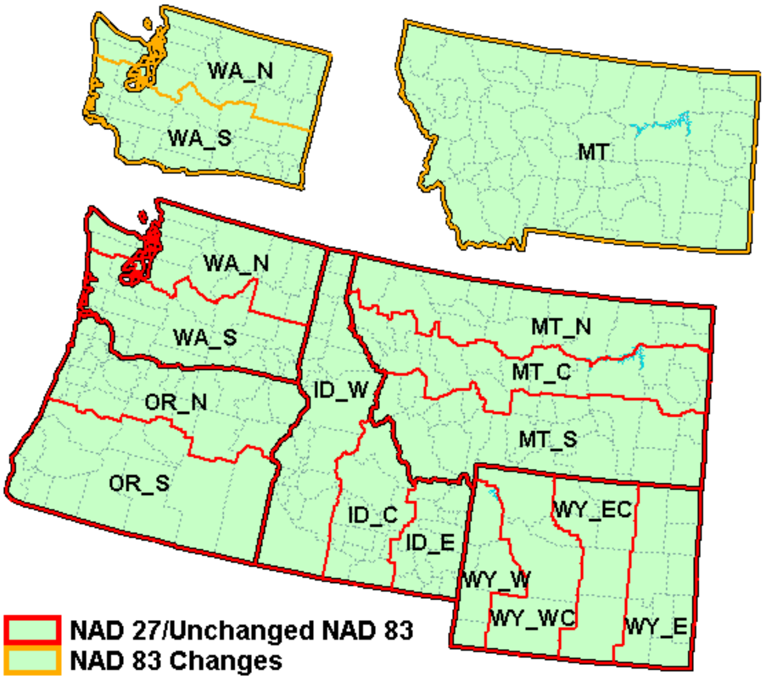
The figure below shows the state plane zones for the New England area.



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

The figure below shows the state plane zones for the Northwest area.

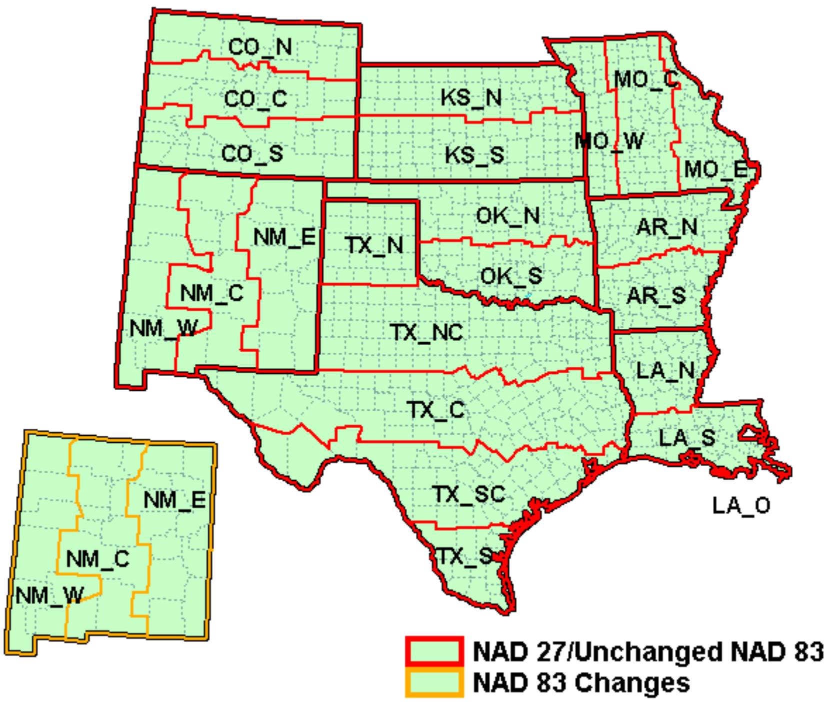


NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Idaho East	ID_E	1101
Idaho Central	ID_C	1102
Idaho West	ID_W	1103
Montana North	MT_N	2501
Montana Central	MT_C	2502
Montana South	MT_S	2503
Oregon North	OR_N	3601
Oregon South	OR_S	3602
Washington North	WA_N	4601
Washington South	WA_S	4602
Wyoming I	WY_E	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

The figure below shows the state plane zones for the South Central area.

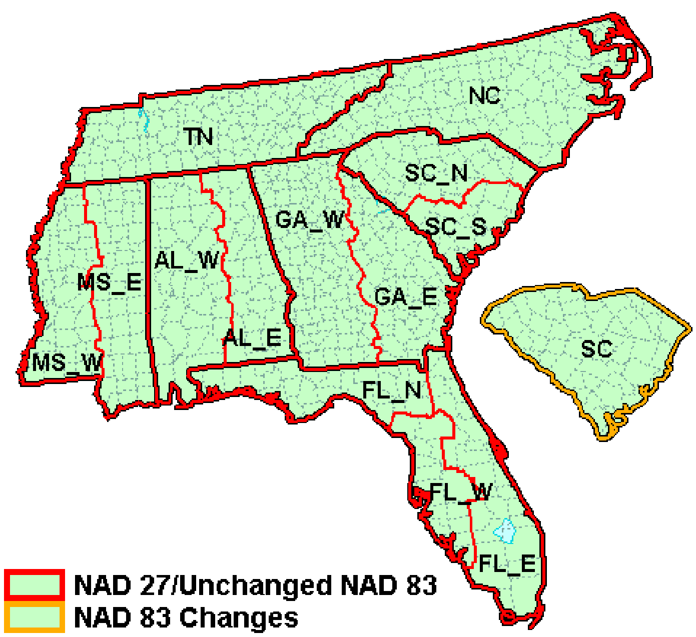


NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Arkansas North	AR_N	0301
Arkansas South	AR_S	0302
Colorado North	CO_N	0501
Colorado Central	CO_C	0502
Colorado South	CO_S	0503
Kansas North	KS_N	1501
Kansas South	KS_S	1502
Louisiana North	LA_N	1701
Louisiana South	LA_S	1702

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

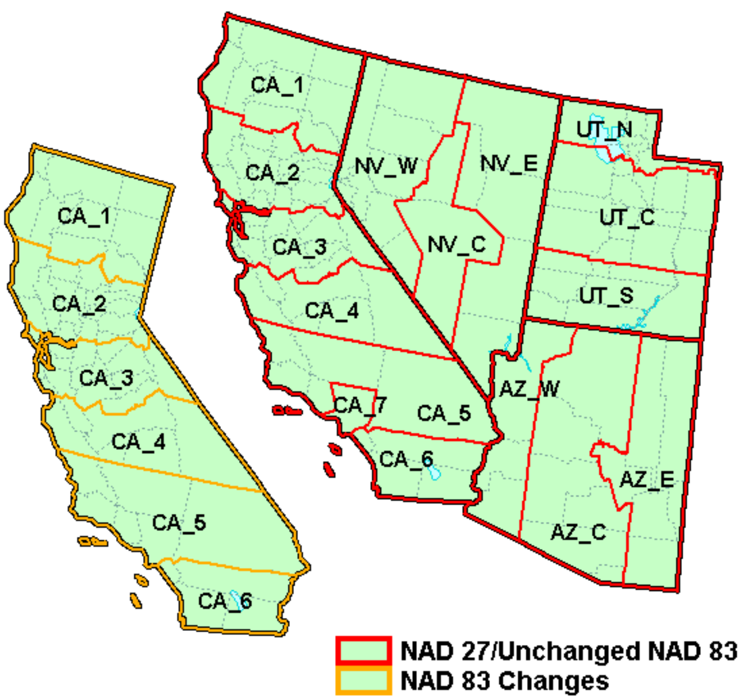
The figure below shows the state plane zones for the South East area.



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

The figure below shows the state plane zones for the Southwest area.

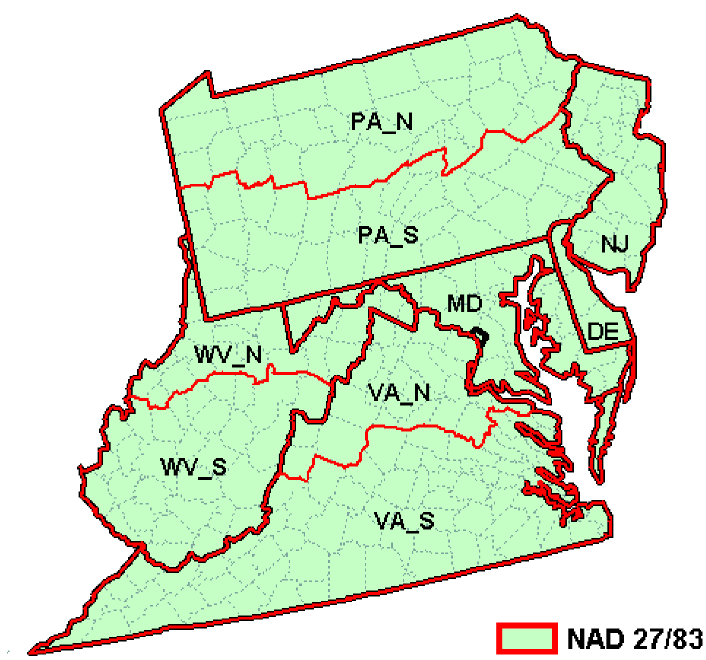


NAD 27 / 83

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

The figure below shows the state plane zones for the Virginia area.



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

Datasets 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 manually 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 dataset 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 examining 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.

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 specifying 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 dataset 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 double-click on the item.

Folder

The datasets and solutions are organized by folders. New folders can be created. It is possible to move datasets, solutions, and folders into 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

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:

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).

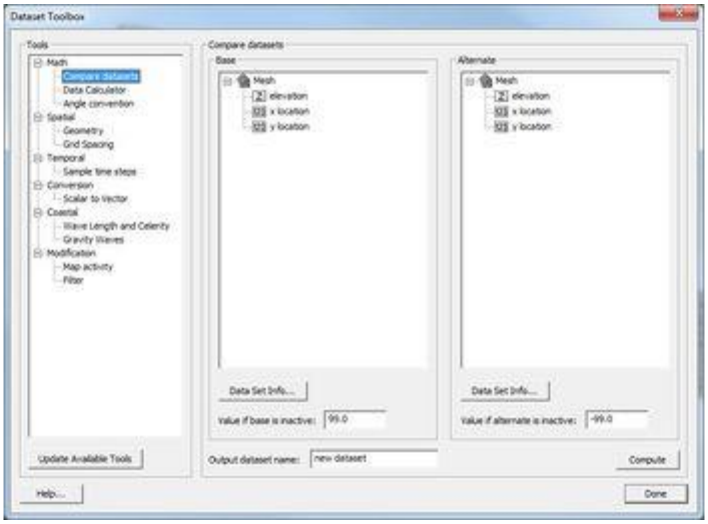
In addition, if the value passes none of the criteria, a default value can be assigned (see available options above).

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** or **Dataset Toolbox** command from the *Data* or *Edit* menu. The components of the *Data Calculator* are as follows:

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)

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 .

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 specifying a maximum slope. The process creates a new dataset which honors the maximum specified slope.

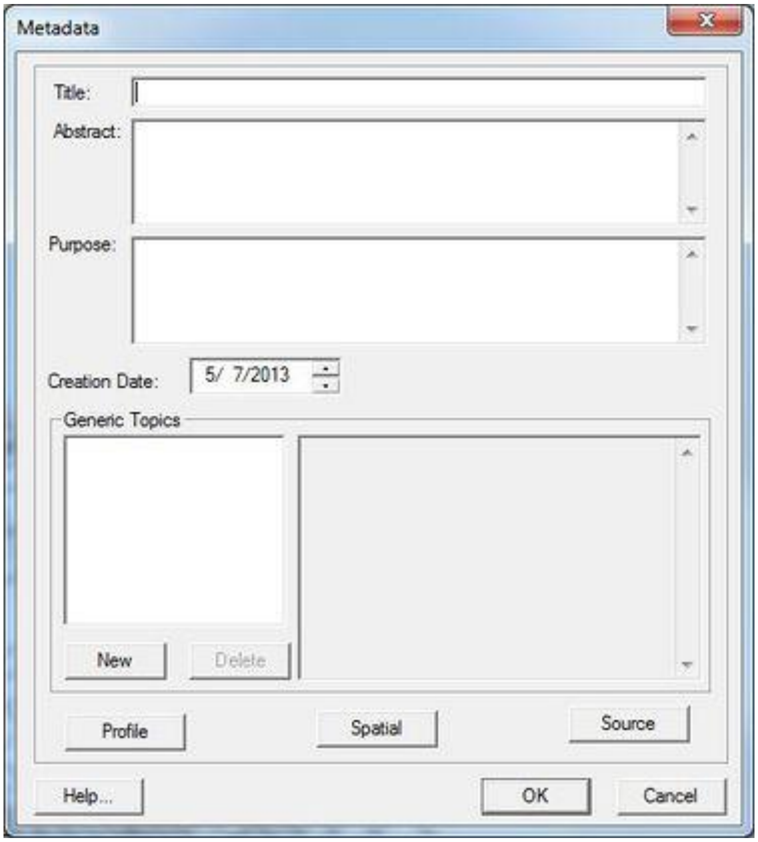
- *Minimum value anchor type* – The smoothing operation anchors the minimum dataset value (such as the lowest elevation or smallest depth) and adjusts the adjacent values to ensure the slope is less than or equal to the specified slope.
- *Maximum value anchor type* – The smoothing operation anchors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

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.

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.

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 any module 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 seen. 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 – Controls the options related to Cartesian grids and the models (such as TUFLOW, CMS-Flow and BOUSS2D) that use them.
- Curvilinear Grid – Controls the options related to the curvilinear or boundary fitted grids and the models (such as LTFATE) that use them.
- General – Controls the general display options.
- GIS – 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 – Controls the display options related to the map module (coverages) in SMS.
- Mesh – Controls the display options related to the general unstructured mesh objects.
- Particle – Controls the display options related to the PTM Lagrangian particle tracker model.
- Quadtree – Controls the display options related to the quadtree module in SMS.
- Raster – Controls the display options related to raster (or DEM) type objects.
- Scatter – 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 which 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, 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, 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, click on the button and the *Line Attributes* dialog will appear. This allows selecting 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, 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, 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.

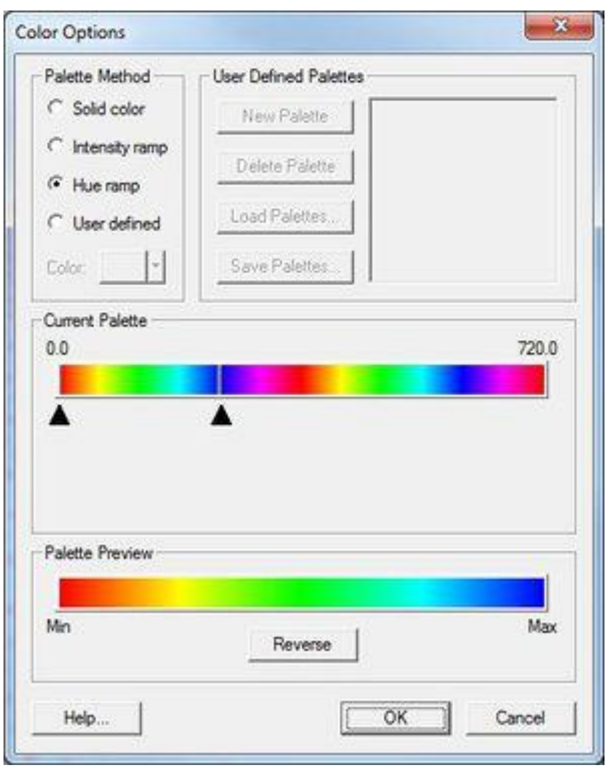
Options

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.

Related Topics

- Display Menu

Color Options



The *Color Options* dialog lets determining 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.

The default color method is the *Solid* color option. This method uses a single color for all contours. As an alternative, it's possible to 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, control 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. Modify 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

It is possible to 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. Select 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. Select, edit, and drag colors in the 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:

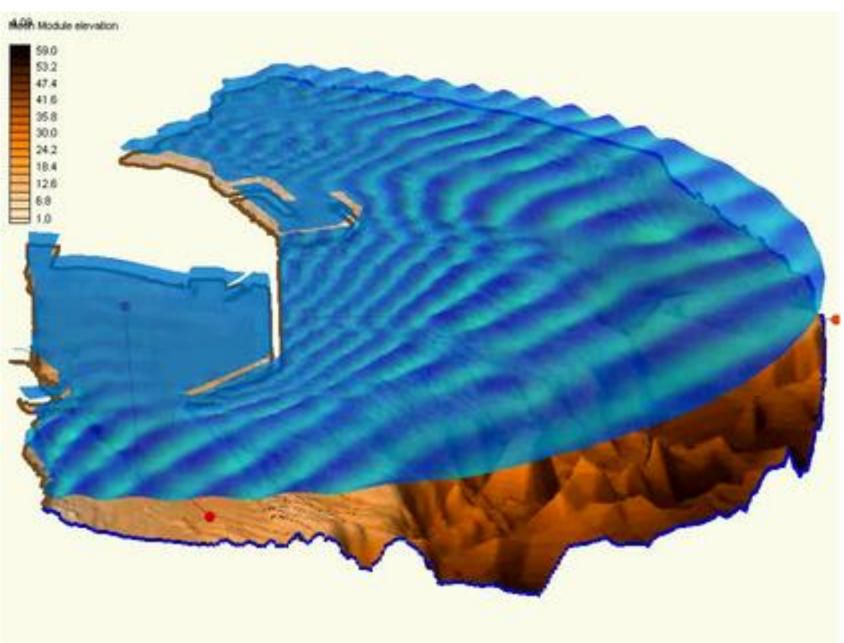
- *Initial Color Ramp Type* :
  - Solid Color
  - Intensity Ramp
  - Hue Ramp
  - Elevation
  - Ocean
  - Magnitude Difference
- *Color* – Only available for *Solid Color* and *Intensity Ramp*
- *Number of Colors* – Only available for *Solid Color* , *Intensity Ramp* , and *Hue Ramp*
- *Palette name* – Allows giving the palette a unique 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.

Related Topics

- [Contour Options]
- [Contour Labels]
- [Display Options]
- [Vector Visualization]

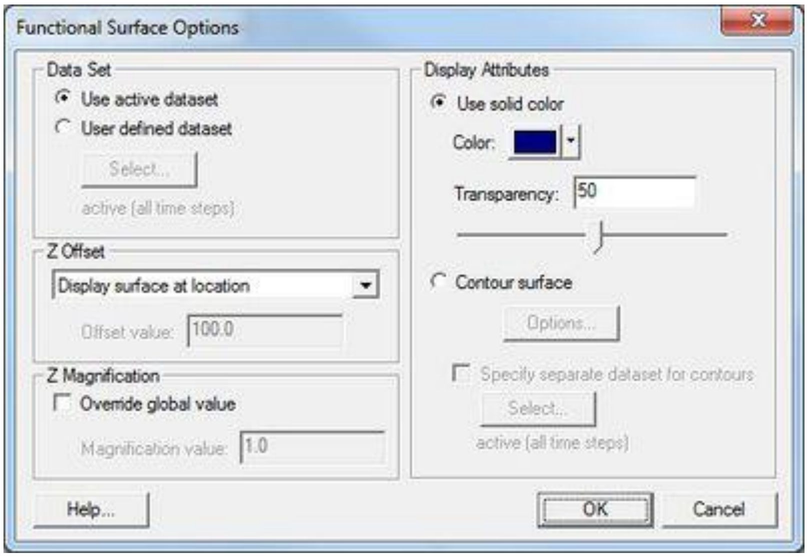
Functional Surfaces



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, enable them in the display options of the appropriate module, and specify their attributes which include:

- *Dataset* – 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* – 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 controlling 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** macro or *Lighting* 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

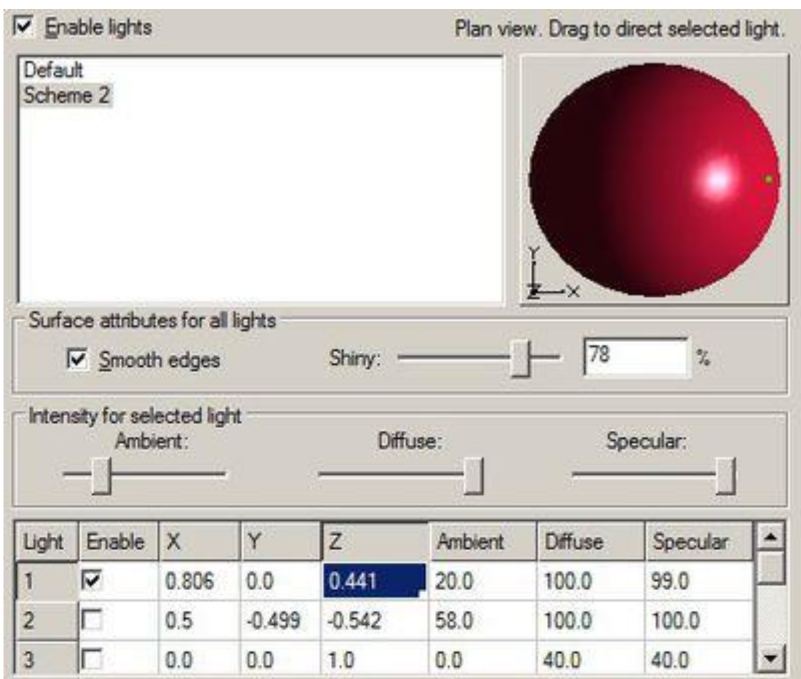
The first toggle allows turning on the use of a light source. When this toggle is selected, the second toggle becomes available. The second toggle tells SMS to smooth corners between adjacent faces. This allows the faceted surface to appear as a smooth surface.

Slider

The slide bar allows specifying 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 setting 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

XMS software provides a number of options for importing and displaying raster data.

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	http://seamless.usgs.gov||2||Supported in GMS and in SMS		
USGS NED Grid Float	*.flt	http://seamless.usgs.gov||2||Supported in GMS and in SMS		
Canadian DEM	*.dem	http://www.geobase.ca||3||Supported in GMS and supported in SMS		
DTED	*.dt0	ERDC	3	Supported in GMS and supported in SMS
Aster DEM	*.tif	http://asterweb.jpl.nasa.gov/gdem-wist.asp||4||Supported in GMS and supported in SMS as images		
SDTS	*.ddf	http://data.geocomm.com/dem/demdownload.html||5||Supported 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 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

General Display Options

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

General Tab

- *Drawing Options:*
  - *Z magnification* – Exaggerates the z scale so that the variation in the z value is more apparent.
  - *Background color* – Set the background color of the Graphics Window.
  - *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 size and color of this triad can be specified.
- *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:* Specifies 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 spaces:* Specifies how many grid lines to between displayed grid lines. The line style is also selected.
  - *Display grid points every spaces:* Enables the display of a point at selected intervals along with the symbol attributes for the points.

Lighting Tab

The *Lighting* tab accesses the Lighting Options in SMS.

View Tab

The *View* tab in the general display options allows editing 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.

The view parameters can be set in two ways: *View bounds* or *View angle* .

View bounds is used for 2D viewing. The minimum and maximum X (left/right) and Y (top/bottom) dimensions for the display in the graphics window can be set. The dimension that can be set depends on which of the following options are set:

- *Specify width with height dependent on aspect ratio*
- *Specify height with width dependent on aspect ratio*
- *Specify width and height bounds*

View angle is used for 3D viewing. It allows setting the following options:

- *Bearing*
- *Dip*
- *Looking at point* – This section allows defining a point for the center of the display. Set the *X* , *Y* , and *Z* values for this point.
- *Define view bounds size* – This section allows defining the viewing bounds of the display. Set the *Width* and *Height* of the boundary area.

Related Topics

- Display Options

Z Magnification

Occasionally an object may be very long and wide with respect to its overall depth (z dimension). In such cases, it is possible to exaggerate the z scale so that the variation in the z value is more apparent by changing the magnification factor from the default value of 1.0. 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.

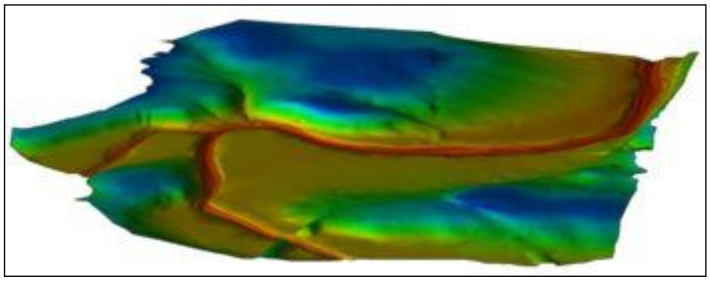
Auto Z magnification

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.

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 which allows specifying 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

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 dataset specific contour options can be defined. If wanting 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.

Data Range Options

The *Data* menu in some modules and models include commands to populate the contour range from either the visible or selected nodes/vertices. These commands are:

- **Set Contour Min/Max** – Sets the contour options based on the current options and the selected nodes/vertices or zoom level.
- **Contour Range Options –** Controls if the **Set Contour Min/Max** command applies to dataset specific contour options or the general contour options (for the mesh or scatter modules). It also sets the flags for precision and fill above and below. This command brings up the *Data Range Options* dialog.

Data Range Options Dialog

This dialog has the following options:

- *Specify precision*
- *Data range applies to dataset only*
- *Fill above*
- *Fill below*

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 entering the title "DS" for the legend title, the name of the current dataset is used. If "DS:TS" is entered, 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 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.

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 generation of automatic contour labels can be toggled on or off. If the toggle is on, specify 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, enter how many digits of accuracy are desired. The default will match the contour legend. On the right side, select a color and font for the label. For labels on contours, also specify if the contour are to 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.

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

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

The *Vector Display Placement and Filter* section controls 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, 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 displaying 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

The *Arrow Options* section 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, 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. It's also possible to 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.

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 viewing a cross section of a 3D solution.

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.

Isosurfaces

Isosurfaces can be used to display 3D solutions. The display options for isosurfaces 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 importing 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 – Delimiting Columns

The first step in the wizard delimits 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 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.

Specify the starting row the data will be imported at. If the data has a row of headings, 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 decides what kind of data to be imported (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 progressing 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.
  - *Import every____th point* – Sets the nth point filter option.

- *Area* – Only reads points falling within the specified x, y boundary. Useful for filtering data outside of the area of interest.
  - *Xmin* – Sets the minimum x boundary
  - *Xmax* – Sets the maximum x boundary
  - *Ymin* – Sets the minimum y boundary
  - *Ymax* – Sets the maximum y boundary

- *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."
  - *Delta X*
  - *Delta y*
  - *# Columns*
  - *# Rows*

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

EXAMPLE

"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

EXAMPLE

"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

EXAMPLE

"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	

EXAMPLE

"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 in the system to a tabular data format. This type of file is sometimes referred to as a CSV (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. Supply 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.

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

Export Tabular File Dialog

This dialog appears after selecting the "Tabular Data File" option in the *Save As* dialog then clicking on **Save** . The dialog has many options for formatting tabular data being saved.

These options include:

- *File Header* – this field will list a default header for the file. The field can be edited to change the header name.
- *Number of Columns* – specifies the number columns to be included in the tabular file. Changing this number will be reflected in the field below.
- *Precision* – indicate precision of digits to save values.
- *Column Headings* – toggling on this options allows entering a custom heading for each column in the field below.
- *Delimiter* – allows specifying the delimiter used in the tabular file. Options include: "Space", "Tab", and "Comma".
- *Data* &ndash: this shows which scatter set is being saved. If multiple scatter sets exist in the project, all will be listed here.
- *Save Meta Data File* – toggling on this option allows meta data to be saved with the file.
    - *Options* – brings up the *Meta File Options* dialog.
- Data specification field – gives an overview of what will be saved in the tabular file. A column will be shown for each column to be saved based on the number of columns specified above. The **Data** button in each column can be used to specify which scatter dataset to be included in the column. Only datasets in the active scatter set can be selected. The discription will show which dataset has been selected for the column. If headers have been toggled on, a field appears to get a header name for each column.

Related Topics

- Tabular Data Files – SHOALS *.pts

Exporting Profile Dialog

The *Exporting Profile* dialog allows exporting the plot data.

Export

- Image Export – this option selects the image format the plot data will be exported as. The following image formats are available:
  - EMF
  - WMF
  - BMP
  - JPG
  - PNG
- Text / Data – this option will export the data as a simple text file.

Export Destination

In this section, select where SMS will send the plot data when exporting.

- ClipBoard – sends the plot data to the clipboard memory of the computer.
- File – Creates an ASCII text file of the data
- Printer – Exports the data to an active 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 .

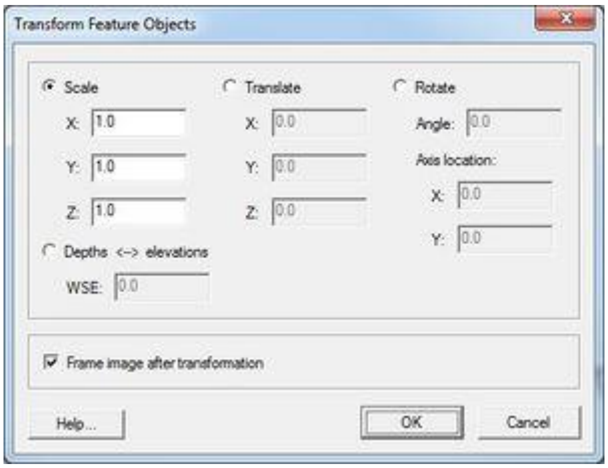
- File Type
  - Binary Dataset Files (*.dat) – Benefits include fast read/write times, small file size.
  - ASCII Dataset Files (*.dat) – Can be imported into Microsoft Excel and viewed with standard text editors.
  - XMDF Dataset Files (*.h5) – Benefits include fast read/write times, small file size, 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.

Related Topics

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

2.8. Geometric Tools

Data Transform



At a glance

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

The **Transform** command is used to move scatter points. A prompt will ask what 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, it's necessary to 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.

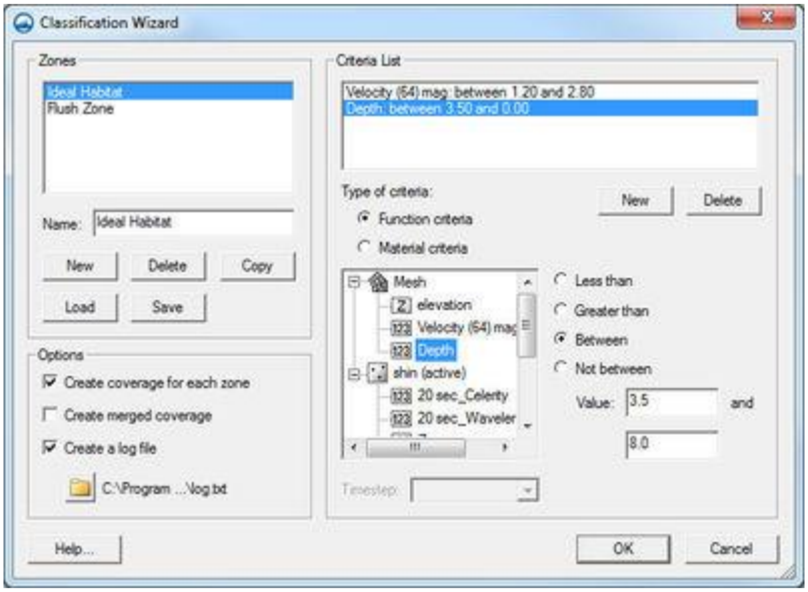
By default, the image will be framed after the transformation takes place. However, this can be turned off by using the *Frame Image After Transformation* option.

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 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.

The tool is accessed through the **Zonal Classification** command in the *Data* menu.

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 using multiple zones, it is possible to 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

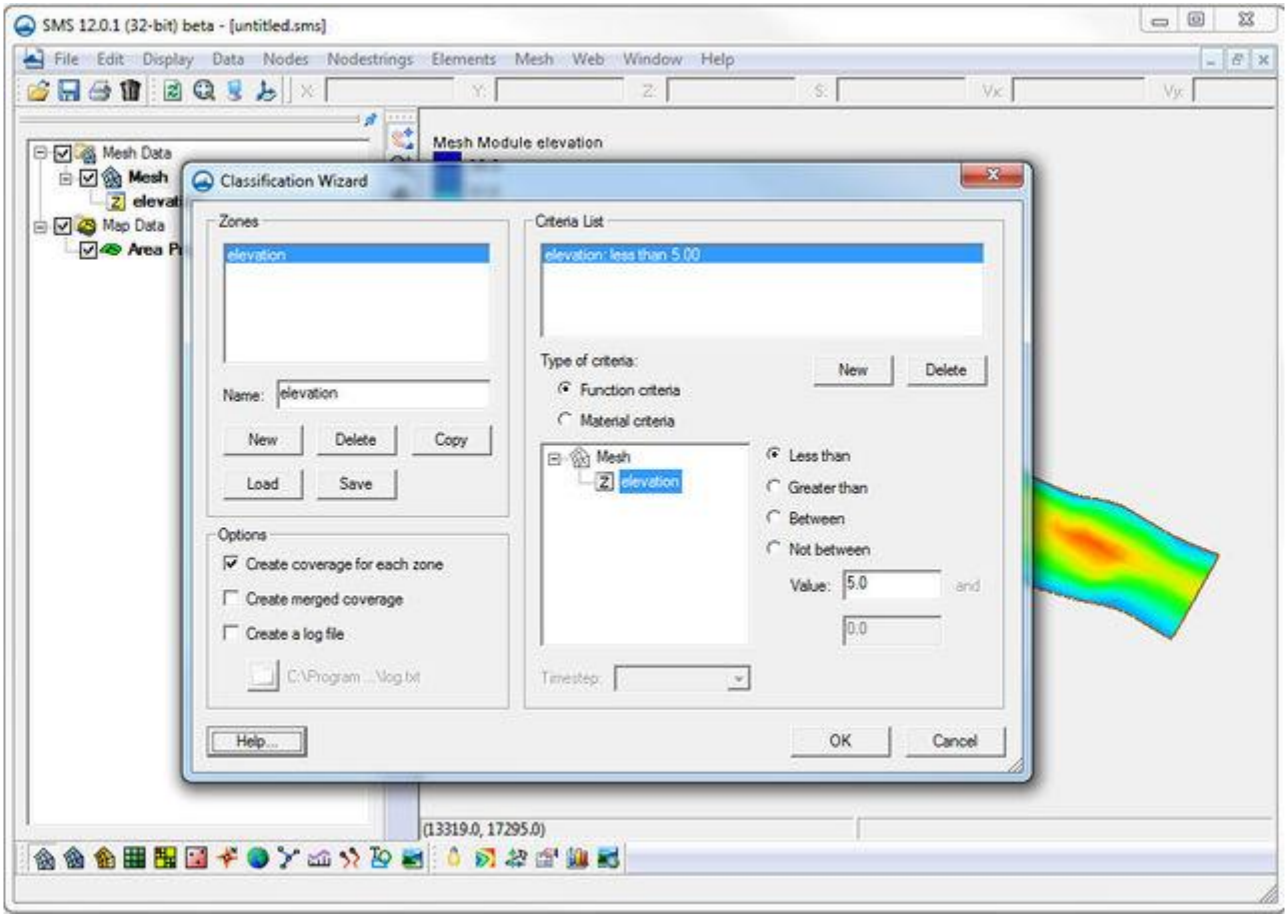
One application of zonal classification is to help quantify the amount and quality of fish habitat. Certain types of fish prefer or require different depths, velocities, and substrate. These preferences depend upon the life-cycle stage for fish.

The following demonstrates how to 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.

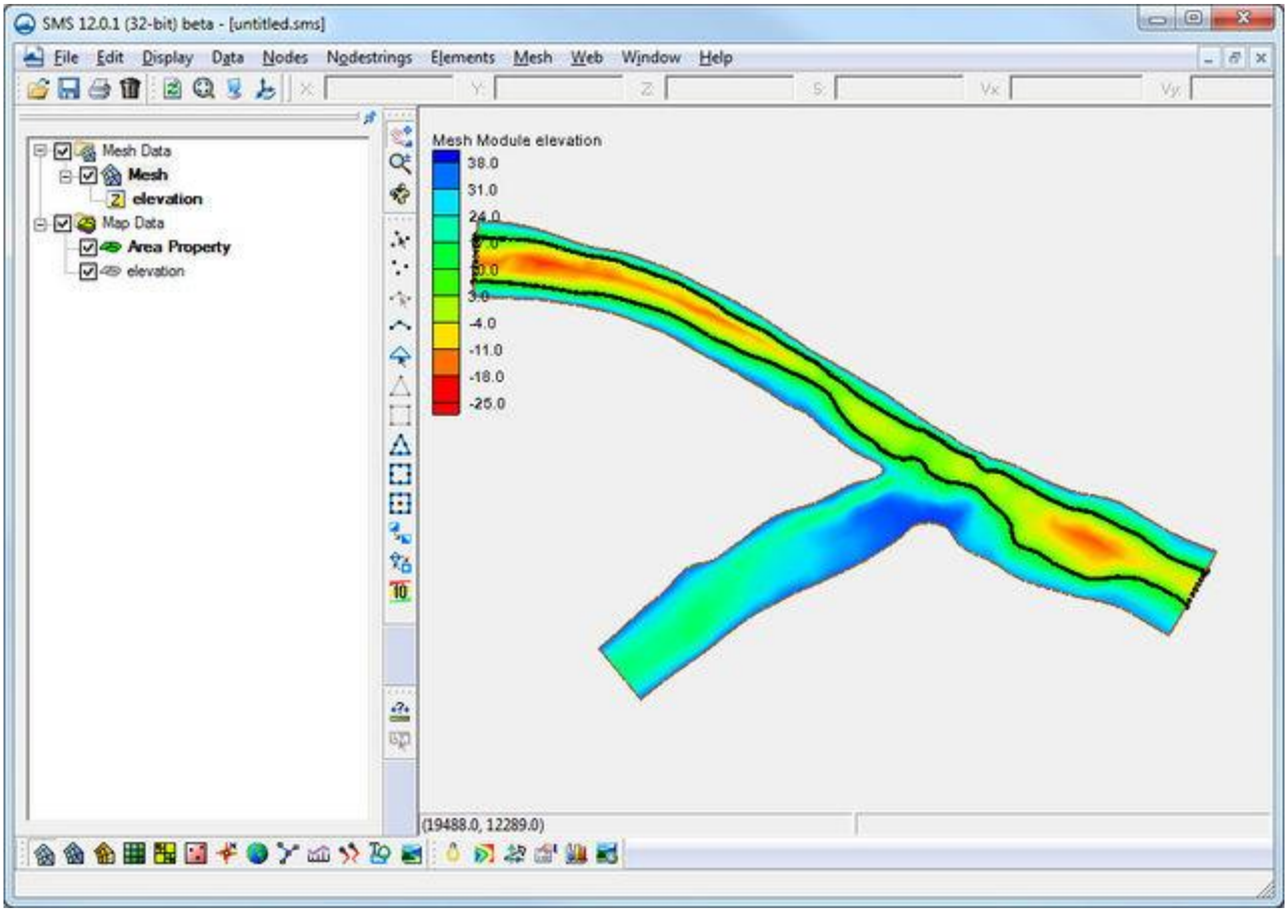
For the first sample criteria, identify areas that have an elevation of 5 ft or less. This is done by:

1) Creating a new zone based upon this criteria.
2) Creating a new criteria based upon a functional criteria.

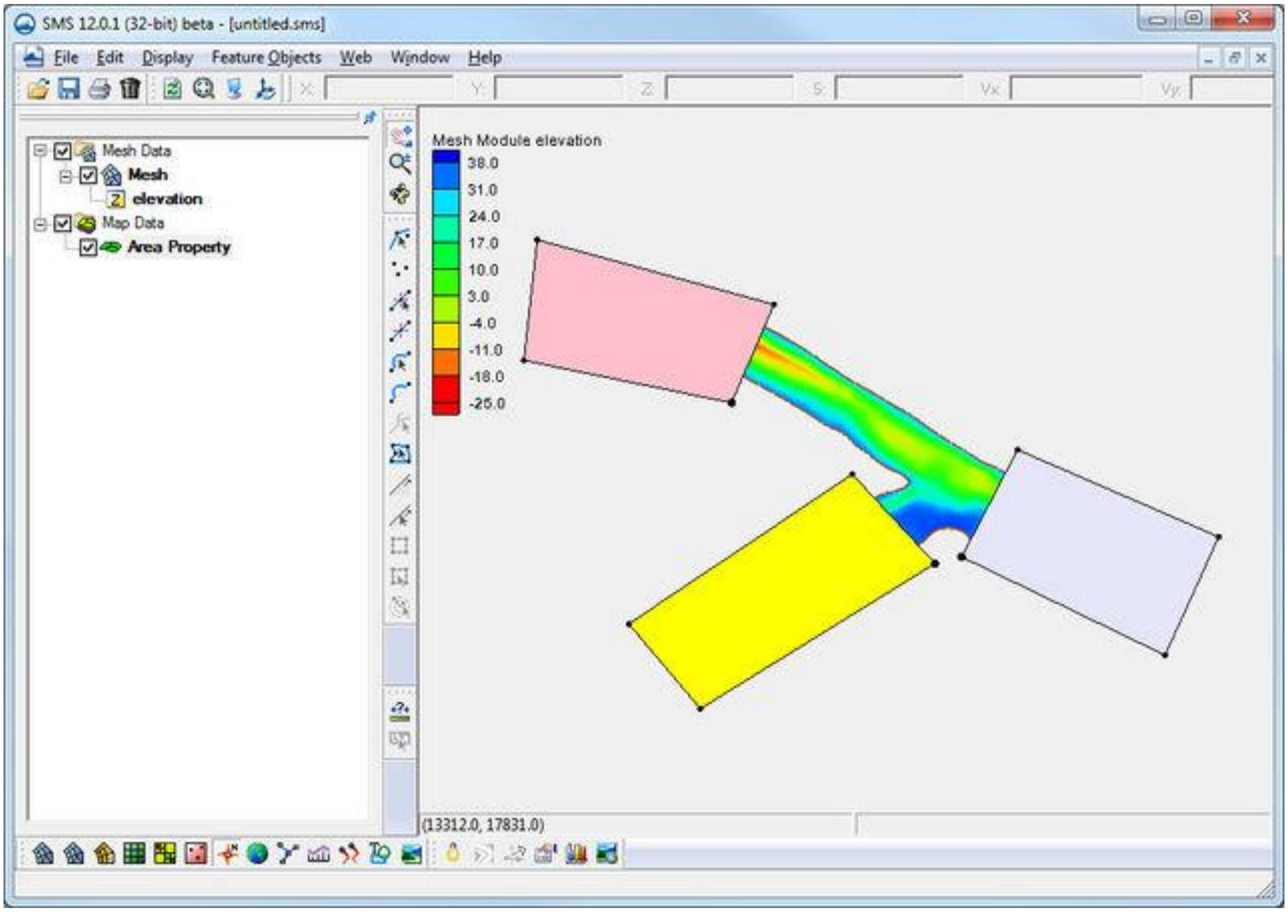
3) Specify the elevation dataset and the criteria to be less than 5 ft.



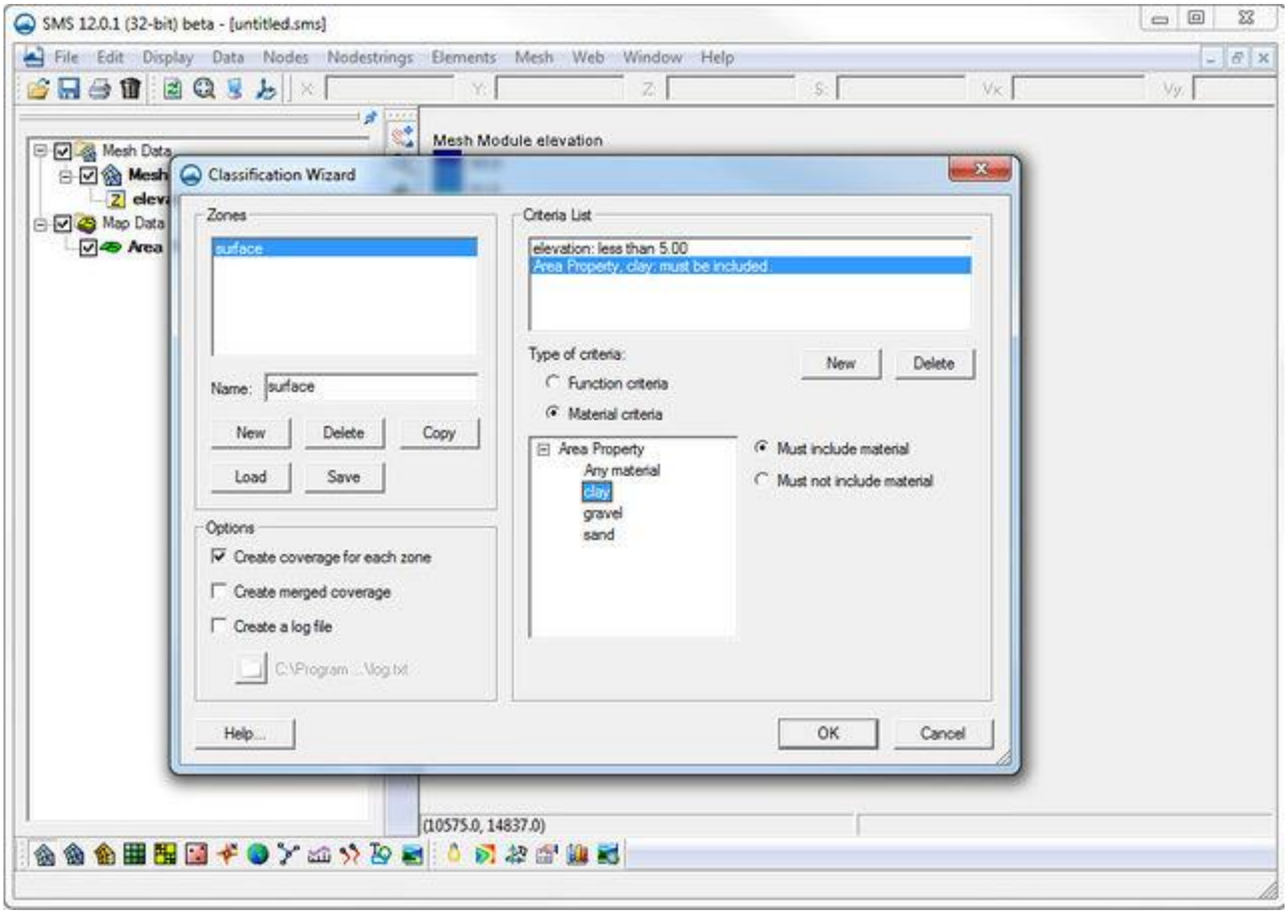
After executing the zonal classification a new coverage was created and polygons identify the areas that meet the requirements of the zone (in our case the elevation less than 5 ft). Assuming that our criteria identifies a target habitat, it is easy to see the areas that meet the criteria.



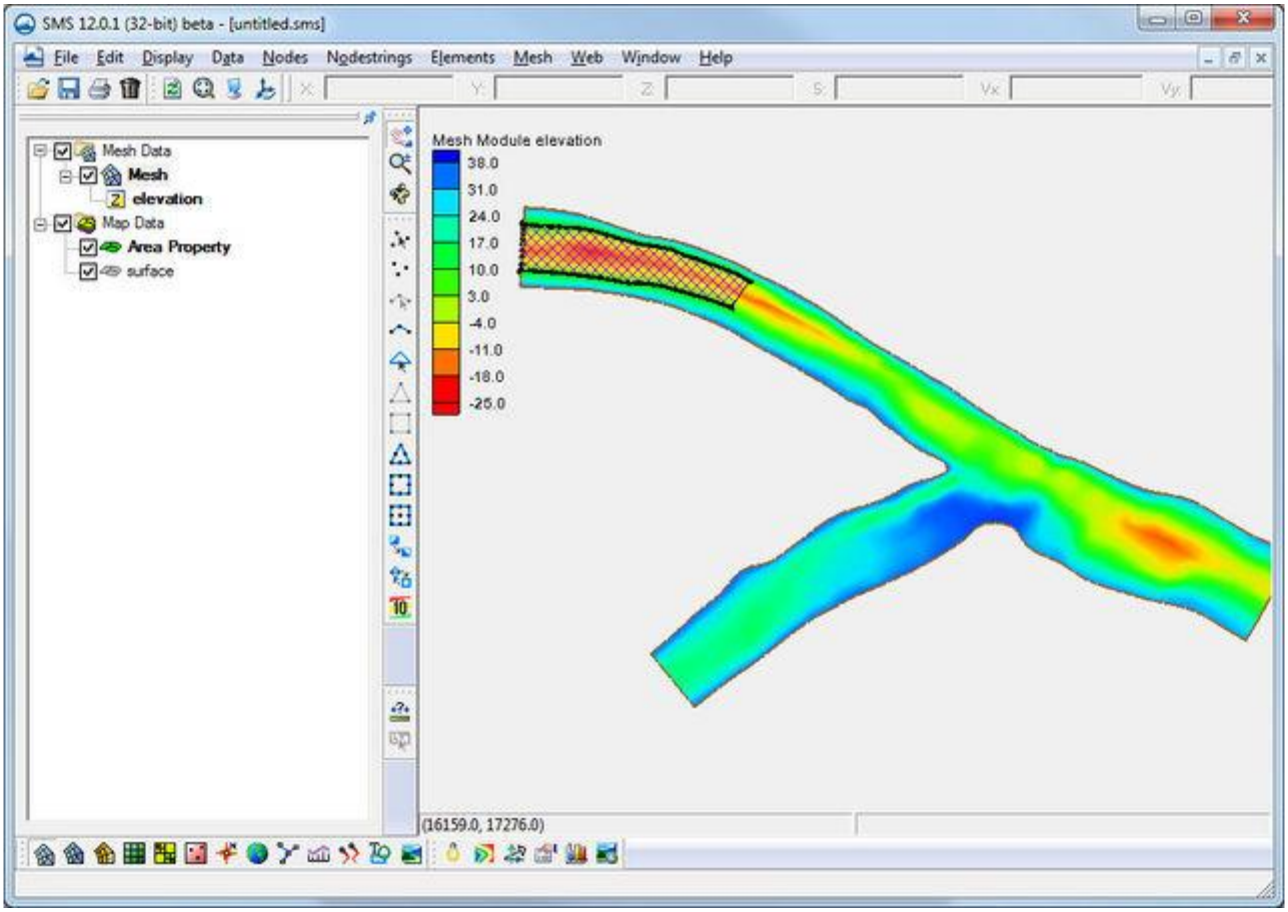
In addition to the elevation, let's assume that our target habitat also requires a specific type of substrate (bottom sediment type). For this example there are polygons created in an area property coverage and areas identified with different substrates (again this information is fictional).



Another criteria has been added to the 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.



References

- Jones, R. D. (2003). *Vector Based Classification of Zones from Distributed Datasets Or GIS Polygon Data* (Doctoral dissertation, Brigham Young University. Department of Civil and Environmental Engineering).

Related Topics

- Data Transform

2.9. Images

Images

At a glance

- Multiple images can be read/viewed at the same time (tiled or overlay)
- Independent transparency specified for each image
- Images can be loaded from web services as either static or dynamic images
- Images can be draped over mesh or scatter data
- Many image formats are supported including JPG, TIFF, PNG, MrSID, and ECW
- Local images can be geo-referenced to view images along with other data
- Image pyramids can be created for very large images

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 .

In SMS versions 11.2, the use of images and similar raster data was greatly enhanced with the addition of the Get Online Maps... and Import from Web... commands in the Web Menu . These commands greatly simplify the acquisition and use of image data. They do require an internet connection, and may require some time to update the image data during display updates. For this reason, utilize the functionality to obtain image data then convert the image to a local *static* image for use with a specific project.

Supported Image File Formats

- Enhanced Compression Wavelet (*.ecw)
- Graphics Interchange Format (*.gif)
- Joint Photographic Experts Group – (*.jpg/jpeg)
- Multiresolution Seamless Image Database – (*.MrSID)
- Portable Network Graphic – (*.png)
- Tagged Image File Format – (*.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 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. This brings up the *Layer Transparency* dialog where a slider allows adjusting the image transparency. The amount of transparency will not be shown until the **OK** button is clicked.

Image Deletion

A single image is deleted by right-clicking on the image in the Project Explorer and selecting the **Remove** command or pressing the *Delete* key.

To delete all images, right-click on the Images folder in the Project Explorer and select **Clear Images** (this feature was removed in SMS 12.0 and higher).

Dynamic Imagery

Starting with SMS version 11.2, dynamic imagery is available through SMS as long as SMS has access to the internet. Availability and quality of the images depend on the area being modeled and the web services available for that area and to the specific user. Many public domain web services are available in the United States and more are being made available all the time.

See Get Online Maps and Import from Web for more information about dynamic images.

From ArcGIS

For version SMS 11.0 (32-bit only), dynamic background images can be accessed from the web only through ArcGIS and only when there is an ArcGIS installation on the computer. In that version, 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 only applies to SMS verions 11.0 and 11.1.

Related Topics

- Get Online Maps
- Import from Web
- Registering an Image
- Image Pyramids

Image Pyramids

The XMS packages include an option to generate multiple resolution versions of an image when it is imported. This can improve the display quality of an image when the native resolution is much higher than the screen resolution. In essence, if the image resolution is larger than the screen resolution, a the display of the image skips pixels so the image appears discontinuous.

The process of generating multiple versions of the image is referred to as *Image Pyramids* . If the feature is invoked, XMS will average 2x2 blocks of pixels in the image creating an approximate version of the image at half the resolution. This process is repeated on the smaller image creating an image at one fourth the original resolution, and so on. Up to four images are generated, based on the relative native resolution and the screen resolution. The goal is to get an image in the *pyramid* that is approximately the same as screen resolution.

When displaying all or part of an image, XMS determines which of the *pyramid* images have a resolution that most closely matches the current screen pixel size and uses that version of the image.

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 the computer hardware. Building pyramids uses more memory RAM. Building pyramids may not improve the on screen display of all images.

Once an image pyramid has been built for a particular image file, SMS will not ask again to build pyramids for that image file unless the image is moved or altered.

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.

Image Preferences

For XMS versions released after summer 2015, the default option is to not generate image pyramids and will not even ask whether pyramids should be generated. This was implemented because the use of large static images has been largely replaced by dynamic images from web services, for which image pyramids do not improve quality.

If desiring to generate pyramids for a large static image, the 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

SMS and WMS make use of the *Import from Web* feature. GMS no longer uses this feature.

Overview

The *Import from Web* feature connects to the internet to download free data – images, elevation data etc. If able to connect to the internet, 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 . 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 selecting (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, 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 the screen can be downloaded by right-clicking on the map and selecting the **Export** command. This command will download the map to the 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 More Info	• NED data – USGS
• World Street Maps More Info	• ASTER and SRTM data – USGS & NASA
• World Topo Maps More Info	• NLCD and CORINE (European) Land Cover data
• MapQuest OpenStreetMap Worldwide Street Maps	• World Imagery More Info
• OpenSTreetMap.org (Global Street Maps)	• World Street Maps More Info
• Other data sources (use the advanced button)	• World Topo Maps More Info
	• USA Topo Maps More Info
	• MapQuest OpenStreetMap Worldwide Street Maps
	• USA Flood Hazard Zones
	• Land Use Shapefiles
	• STATSGO and SSURGO Soil Type Shapefiles
	• Harmonized World Soil Database v 1.1
	• Global Land Cover
	• 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 the 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 .
2) Download the .zip file Globcover2009_V2.3_Global_.zip and unzip this file on the 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 for 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 from the Harmonized World Soil Database web site .
2) Launch the HWSD Viewer on the 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 downloaded. The program may also be installed in c:\program files\<...> if 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 the 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 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 if 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 the location of interest.
  - **Zoom** in or out using the controls or the mouse wheel.
  - **Pan** using the controls or by clicking and dragging. It's also possible to 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 defines the data to be downloaded. These dialogs include:

1) **Data Service Options**

   Here select which type of data of interest.

2) **Save**

Next a dialog asks where to save the data. It is only necessary to specify one file name, even if having selected more than one type of data in the previous dialog. The files will all be given the same prefix but different suffixes.

3) **Confirm File Creation**

A dialog may ask to confirm creating the files.

4) **Initialize Connection**

The following dialog is shown while the connection is being made.

5) **Select Scale**

Smaller numbers (larger scales) will result in better resolution, but longer download times.

6) **Downloading**

This dialog reports the download progress. If **Abort** is clicked, the image will exist but will be only that portion that has been 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.

Registering an Image

If an image file is not geo-referenced then it's necessary to define the coordinate system of the image. The *Register Image* dialog allows specifying 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 Image* 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® or Arc/Info® . 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 there are (x,y) coordinates in a different coordinate system than the project, the coordinates will need to be converted.

GMS Point Conversion

The **Convert Point** button in the image registration dialog will allow converting the coordinates.

SMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if it becomes necessary to convert between any two coordinate systems. 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 it becomes necessary to convert between any two coordinate systems. Perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

Save as Image

It is possible to save information displayed in the Graphic Window in an image format. It can be useful to save the contents displayed in the graphics window so the images can be used for presentation purposes, documents, etc.

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 formats:

- Bitmap Image Files (*.bmp)
- JPEG Image Files (*.jpg or *.jpeg)
- PNG Image Files (*.png)

After clicking save, SMS may take a moment to process the image. Only the contents showing in the Graphics Window are saved. Items in the Project Explorer that have been hidden will not be seen in the final image.

SMS will not center or frame the image before saving. The final image will be exactly what is displayed in the Graphics Window prior to using the **Save As** command. This includes all active display options. The final size of the saved image will also be determined by the size of the Graphics Window when saving. If wanting a larger or smaller image, either adjust the image in an image editor after saving or change the size of the Graphics Window before saving.

JPEG compression options will not be given when saving from SMS. SMS will save JPEG images at the highest (uncompressed) quality. Likewise, compression options will not be given when saving an image in the PNG format.

Related Topics

- Images
- File Menu

Web Service for Background Imagery

GMS and SMS support the ability to obtain image data from web servers. The imagery will update when panning and zooming as well as uses an appropriate resolution for the current zoom level. Since this is obtaining the information over the internet, the performance of these images will be dependent upon the speed of the internet connection.

System Requirements

SMS 11.0 and ArcGIS 9.3 or above are needed for this feature.

Projection

To use web layers, the project must be in a non-local projection.

GIS Web Layers

SMS ships with several layers that can be used. Some of the layers are specific to the US and others are worldwide. If desired, 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 folder for the version, and click on the item labeled "supporting files." This will open an explorer window to the folder that contains supporting files used with SMS. A folder named "GIS Layer Files" should appear. This folder contains the GIS web layers that ship with SMS.

In order to load the GIS layers, it's necessary 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 zooming 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 further zooming in, the more details will appear. When zooming out, the details will become less visible.

Related Topics

- Images in SMS

2.10. Preferences

Preferences

The *Edit* | **Preferences** command brings up the *Preferences* dialog. The *Preferences* dialog contains the following tabs:

General

- *File IO*
  - *Compress XMDF Files* – Use compression when saving XMDF files.
  - *Override temp directory* – Specify the location where SMS temporary files are written.
- *Help Option*
  - *Use local help* – Option to access the CHM file include with SMS when the **Help** button in a dialog is pressed.
  - *Use online help* – Option to access the XMS Wiki when the **Help** button in a dialog is pressed.
  - *3rd party online* – Allows specifying third-party help using the Dynamic Model Interface Schema.
  - *Prompt*
- *View Data File Option*
  - *Ask for Program* – SMS will prompt to specify the program used to open a data file when the *File* | **View Data File** command is called.
- *Deletions*

  - *Confirm Deletions* – Choose to be prompted to confirm the deletion whenever a set of selected objects is about to be deleted. 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.

Defaults

The *Defaults* tab was previously referred to as the *Startup* tab. It allows defining which modules and models are active by default when SMS is launched.

- *Default Module* – In this combo box, specify which module is active module at startup.
- *Default 2D Mesh Model* – In this combo box, specify which numerical engine (model) will be assigned to newly created meshes. This is the active 2D Mesh Module Model at startup. As SMS migrates to simulation based modeling, the application of this tool will be less important because simulations will be explicitly created for a specific numerical engine.
- *Default 2D Cartesian Grid Model* – In this combo box, specify which model will be assigned to newly created Cartesian grids. This is the active 2D Cartesian Grid Module Model at startup. The need for this command will also go away as SMS migrates to simulation based modeling and simulations are created for a specific numerical engine.
- *Default Coverage Type* – In this combo box, specify the active Map Module Coverage Type at startup. Currently, SMS always requires at least one coverage to exist in a session. This coverage type controls the type of that new, blank coverage as well as the default type of newly created coverages. In most commands, there is an option to specify or change the type of the coverage as it is being created.
- *Check version on startup* – This command instructs SMS to check, via an internet query for an update to SMS from Aquaveo. If the user machine is behind a firewall that does not allow access to the internet, this command will not function properly.
- *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.

Images

The images tab includes preferences related to the display and manipulation of images in SMS. Dynamic images or images loaded from web sources are available based on location of the simulation. Specific preferences include:

- *Image Pyramids* – This function specifies whether SMS will "Always Build", "Never Build", or "Prompt for Each Image" when building image pyramids . It's recommended that this option be set to or left at "Never Build" unless a very large static image file is being used. This recommendation comes from multiple sources. First, since web sources are much more standard, and image pyramids don't apply to dynamic images, the command is superfluous. Second, the creation of image pyramids results in the several new image files saved on the user's local machine. The images are multiple resolution representations of a specified loaded image. The creation of these images has led to confusion.
- *Save* – Specifies where SMS saves temporary and/or created images. Options include a specific "Image Folder" or the "Temporary Folder" (as specified int eh General tab above). The image folder is created in the directory with the SMS project. Images are created either from the command to convert the graphics window to image format, or when converting dynamic images to static images to reduce dependency on an internet connection and speed up image refresh.

File Locations

This tab allows specifying the location of applications and folders that may be used in the course of an SMS session. Once a new target is specified, SMS remembers that location for all future uses of the application. This tab will only shows models with an active license.

- *Model Executables* – This table occupies the upper section of the tab and allows specifying the location of numerical model executables. Some models include only a single executable, others include two, three or four. By default, an SMS installation is initialized to look in the *models* directory in the folder where the SMS program is installed. The installation will include a sub-folder for each model installed with the SMS installation. Both 32 and 64-bit versions of most numerical engines are available.
- *Other Files* – This table occupies the lower section of the tab and allows specifying the location of resource files that may be utilized during an SMS session. These files include:
  - The LeProvost tidal database .
  - The location of a resource folder containing bitmaps of *North Arrow* representation to be used by the annotation tools. Aquaveo provides several default bitmaps. Individual users can create custom bitmaps for this use.
  - The path to the log file for TUFLOW simlations
  - The path to the MPIEXEC application which is utilized for MPI parallel process execution.
  - Various other resources that are currently under investigation.

Project Explorer

The project explorer tab provides preferences for interacting with the data tree in the project explorer. Supported options include:

- Force active scalar and vector datasets to be in the same folder – When this option is selected, the scalar and vector datasets selected for any geometric object (mesh, grid, scatter set, boundary fitted grid, ..) must be in the same folder. If a scalar data set is selected in a different folder in the project explorer, the first vector dataset (if one exists) will also be selected. If a vector data set in a different folder is selected, the first scalar dataset in that folder will be selected.
- Add diagnostic files when reading model solutions. If this toggle is selected, SMS will add an entry to the project explorer data tree to link to the diagnostic (text messages) file associated with a numerical simulation. Not all numerical engines support this type of a file.

Toolbars

The toolbar tab allows controlling the status and position of each toolbar in SMS at startup. These positions/status values can be changed by dragging each individual toolbar during a session of SMS. The toggle box to the left of the toolbar controls whether the toolbar will be visible when SMS start. The toolbars included in this feature include:

- File Toolbar – This toolbar includes the four file menu commands (Open, Save, Print and Delete). By default it is visible and it appears at the top of the screen between the menu bar and the project explorer.
- Module Toolbar – This toolbar includes the modules. In early versions of SMS it was the only way to switch between modules and was commonly used for navigation. With the addition of the project explorer, the module toolbar was less essential. It is still ON by default, but was moved to the bottom of the screen below the project explorer as a default location.
- Display Toolbar – This toolbar includes the principal display menu commands (Refresh, Frame, Display Options and Plan View). By default it is visible and it appears at the top of the screen between the menu bar and the project explorer.
- Optional Macro – This toolbar includes the secondary display menu commands (Lighting, Contour Options, Vector Options, Info, Plot, and Web data). By default it is visible and it appears at the bottom of the graphics window.
- Edit Window – This toolbar includes the edit fields for viewing/specifying the coordinates of a selected point/vertex/cell and the associated dataset values. It appears at the top of the graphics window.
- Data Toolbar – This toolbar includes the measure tool and get image tool.

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).

Map

This tab includes options to *Snap feature objects to displayed inactive coverage nodes and vertices* when creating new feature objects.

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.

Mesh

This tab includes an option to specify the precision that will be used to output "2dm" mesh files from SMS.

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).

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.

Time Settings Dialog

The following options are available in this dialog.

- *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.

- *Display as*

  Option to use either absolute or relative 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.

  - *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.

- *Format*

Time can be displayed in a number of formats. After selecting an option, the dialog will display an example of the format. The following format options are available:

Absolute Date Format	**Example**	**Additional Options**
mm/dd/yy	05/25/00	
dd/mm/yy	25/05/00	

mm/dd/yyyy	05/25/2000	
dd/mm/yyyy	25/05/2000	
dd-mmm-yy	25-May-00	
dd-mmm-yyyy	25-May-2000	
mmm dd, yyyy	May 25, 2000	
Absolute Time Format	**Example**	**Additional Options**
hh:mm am/pm	3:22 PM	
HH:mm	15:22	
hh:mm:ss am/pm	3:22:30 PM	
HH:mm:ss	15:22:30	
Relative Time Format	**Example**	**Additional Options**
days hh:mm:ss	10 20:30:40	
hours:mm:ss	260:30:40	
days hh:mm	10 20:30	
hours:mm	260:30	
days (decimal)	10.854296	*Specify Precision*
hours (decimal)	260.51111	*Specify Precision*
minutes (decimal)	15630.66667	*Specify Precision*
seconds	937840.0	*Specify Precision*

Related Topics

- Layout of the Graphical Interface

2.11 Cross Sections

Editing Cross Sections

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

Edit cross sections in one of three ways:

1) When double-clicking on an arc in a 1D Hydraulic Cross Section coverage, it's possible to assign a cross section from a database. After assigning the cross section, also enter the editor for that cross section.
2) Opening a cross section database for editing (or create a new database) using the **Manage Cross Sections** command.
3) Opening an existing cross section database using the *File* | **Open** command.

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

General Properties

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

Editing Geometry

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

Geo-Referencing

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

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

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

Line Properties

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

Point Properties

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

Merging

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

Filtering

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

Related Topics

- 1D Hyd Cross Section Coverage
- TUFLOW 1D Cross Section Coverage

Managing Cross Sections

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

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

Cross Section Database Definition

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

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

The *Cross Section Database Management* dialog also allows creating a new cross section; edit, copy, or delete an existing cross section; insert an entire database (merge databases together); convert a cross section database to a coverage (the georeferencing of cross sections must be provided for the cross section to be included in the coverage); create a digital terrain model from the cross section geometry; and converting the coverage to line properties.

Related Topics

- 1D Hyd Cross Section Coverage
- TUFLOW 1D Cross Section Coverage

2.12. 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 point 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.

Tools

- **Select Cell** – 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 a location can be chosen 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 a location can be chosen 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

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.

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):

Parameter Settings

- **Generation Method** – Choose the method (TMA (Shallow Water), JONSWAP, Bretschneider (ITTC), Pierson-Moskowitz, or Ochi-Hubble Double Peak) to use to generate the spectra. Each method has slightly different options. JONSWAP and Pierson-Moskowitz both require additional information about what values to use to specify the spectra.
- **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

- **Projection** – The wave direction can be specified in a Oceanographic, Meteorologic, Shore Normal, or Cartesian 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** – Number of Nearest Neighbors to be calculated, 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. The dialog consists of a *File type* drop down menu and a browser button  to select an existing file.

File types

The following types of files can be imported through this dialog.

- *.*eng* – Spectral energy file. Prompt will appear to specify if the file is in the CMS-Wave or STWAVE format.
- *.*h5* – XMDF grid and spectrum.
- *.*dws* – The BOUSS-2D spectra file format.
- *.*cdip* – Data from the Coastal Data Information Program .

Import Spectra Processing

After clicking the **Import** button in the *Import Spectra* dialog, indicate to SMS how to process the selected file.

Open Files

If spectral data is in multiple files, the option to place these files in one coverage. This is done by adding these files to the *Open File* dialog that appears after clicking the **Import** button. Click the **Add** button to open a file browser to add additional files. Each file selected will appear in the field below. The **Remove** button can be used to eliminate unwanted files.

Time Settings

When importing ENG files for STWAVE or DWS files for BOUSS-2D, the *Time Settings* dialog will appear after a warning dialog appears. ENG files may contain time stamps for each time step. If the file has an 8 or 12 digit time stamp, SMS will read it in and assign the times accordingly. If the ENG file is not using time stamps, it will just have an integer ID for each set of data. In this case, it's necessary to give SMS a reference time, and SMS will treat each ID as the number of hours past the specified reference time.

Time Increment

DWS files for BOUSS-2D require an additional step. Each dataset in the DWS file will be offset by a specified number of hours from the reference time. This is done in the *Time Increment* dialog.

Grid Angle and Specify Location

When importing ENG files for CMS-Wave, the *Grid Angle* dialog will appear. CMS-Wave ENG files do not specify the grid angle, so it's necessary to enter one here.

After indicating the grid angle, the *Specify Location* dialog will appear. CMS-Wave ENG files also do not specify the location of the spectral data. It's necessary to enter in the location manually.

Times to Import

When importing CDIP files, the *Times to Import* dialog will appear to select the times to import. The *Start time* and *End time* shown represent the times available in the file. Select which start/end times to be imported, and select a time interval. The datasets will be read in or interpolated so that there is a dataset for the starting time and then one at each time interval until the end time.

Update Reference Time

In some cases, the reference time will need to be updated in the *Spectral Energy* dialog.

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 to be imported.
- **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.

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 *CMS-WAVE Model Control* 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, specify which 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. SMS does not allow specified spectra on two adjacent boundaries. Therefore, there can be 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 setting 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.

Angle Convention

The angle convention controls allow choosing 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

2.13. 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

. See the

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. 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

External Links

- www.vtk.org

Conversions Scalar/Vector

Datasets can be converted from scalar datasets to vector dataset or converted from vector datasets to scalar datasets.

Conversions Scalar↔Vector

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 checkboxes next to the datasets to be created.
4) Give the new scalar datasets a prefix in the *Prefix* field.
5) Click **Ok** .

Dataset Calculator VTK

The dataset calculator allows creating 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. 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 selecting an expression from the list of expressions, the name and expression fields will be populated. Make the current list of expressions the default list by saving settings (*File* | **Save settings** from the main menu). Move the 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, if wanting 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, specify if the dataset created is to be associated with nodes or cells. If the use doesn't specify the output location, SMS will decide based upon the variables used in the 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 to 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). 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. It is also possible to 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. 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 correcting 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.

Related Topics

- Datasets VTK

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 option.

Interpolation Dialog

When selecting an interpolation command, the *Interpolation Option* dialog appears. Selects the appropriate options and once the **OK** button is selected, the interpolation procedure is performed. Specified options include:

- Interpolation Method – TSelect a current method that is used for all interpolation until another method is selected. 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. Select 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 – Selects the datasets to interpolate from by marking their check box. Multiple datasets may be interpolated at a time.
  - New name – Renames the dataset created as a result of the interpolation.
  - Map Z – Designates 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. 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. Select an extrapolation dataset for each dataset to be interpolated from.

Related Topics

- Scatter Interpolation

3. Modules

Modules

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

Module Selection

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

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

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

- Mesh Module
- Curvilinear Grid Module
- Cartesian Grid Module
- Quadtree Module (SMS 12.0 and later)
- Scatter Module
- Map Module
- GIS Module
- 1D Grid Module
- Particle Module
- Images (merged into GIS module in SMS 12.0 and later)
- Annotations
- CAD Data

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 switching to a different module. Activating a module simply changes the set of available tools and menu commands.

Annotations

Annotations can be added to a project to provide notes and clarification. The Annotations application is included in all paid editions of GMS and SMS .

Annotation Objects

The GMS and SMS applications contain tools to annotate the data in an application for presentations, animations and screen shots.

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

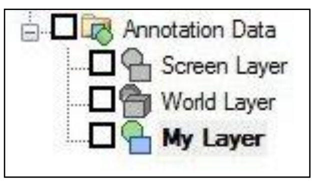
- Images
- North Arrows
- Scale Bars
- Text
- Lines
- Ovals
- Rectangles

Screen vs World Space Layers

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

When the first annotation object created, the program will ask which type of layer (screen or world space) to create and add the object to. Create additional layers by right-clicking on the *Annotation Data tree* item and selecting **Create Screen Space Layer** or **Create World Space Layer** . Layers are differentiated by including an 'S' for screen space layers or 'W' for world space layers in their icons in the project explorer.

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



Annotation Object Attributes

The extents of annotation objects defined by a frame. Initially define this frame when creating the annotation object by left-clicking at any point on the screen and dragging a rectangle with the mouse (left button still down). The display will show the frame while dragging with the mouse. (Points and lines defining degenerate frames are not allowed.)

When creating a annotation, if the frame is too big for the window, it will be resized appropriately. Annotations can't be resized or moved even partially outside of the borders of the window. If, through a quick mouse drag, resizing a annotation causes the cursor to land outside the window, the annotation will be redrawn to take up all the window space in that direction.

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

The horizontal position can be set from the left edge, the right edge or the center of the object. If positioning the left edge, the object position is defined relative to the left edge of the screen. If positioning the right edge, the object position is defined relative to the right edge of the screen. If positioning the center of the object, the object position is defined relative to the horizontal center of the screen.

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

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

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

Screen Space Images

A screen space image is simply a graphics icon mapped to the screen. A typical application would be to display a company, department, or municipality logo next to the numeric model being displayed in the graphics window. Image file formats currently supported include the following: BMP , GIF , JPG/JPEG , PNG , SID , and TIF/TIFF .

To add a screen space image, use the **Add Annotation Image** tool and click anywhere in the graphics window. Use the *Open* dialog to select and add the desired image. Dragging a box in the graphics window will fit the image to he box size.

Using the **Select Annotation Objects** tool, the image attributes can be changed by right-clicking on the image and selecting the **Properties** command. This will bring up an *Image Properties* dialog with two tabs where the following options can be used:

Image Tab

This tab gives the general image properties.

- *Fixed aspect ratio* *ndash; Assigns whether the image is being displayed as a scaled (distorted object), scaled based on its original aspect ratio, or locked at another aspect ratio.
- **Revert to Original Aspect Ratio** – Returns the image to the aspect ratio it had when it was added to the project.
- *Transparency Options* – These options allow for an image to be redrawn with a transparency.
    - *Use transparency* – When checked it will cause the image to be redrawn with the most used color in the image. Clicking the transparency checkbox to the off state causes the image to be redrawn with no transparency.

    - *Specify color* – If checked, it will activate the color button and the color button will have the latest chosen image color painted on it or the most used color in the image, if it has not been activated before. Clicking on the down arrow part of the color button causes a color popup to be displayed with swaths of the 40 most used colors in the image or all the colors in the image, if the image has less than 40 colors. Clicking on one of those colors will cause the image to be redrawn with that color made transparent in the image.
    - *Tolerance* – This edit field allows for variation in the matching of the red, green and blue components. The tolerance field ranges in allowable values from 0.0 to 1.0. 0.0 means the red, green and blue components must exactly match. Values higher than 0.0 indicate the degree of variation from the given color.

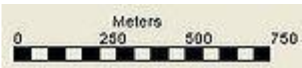
Anchor Tab

This tab handles options for the size and positioning of the image.

- *Horizontal*
    - *Anchor* – The options here determine which direct to move image when moving or resizing the image along the x-axis. Options include "Left", "Center", and "Right". "Left" will offset from the left edge of the image or resize from the left edge. "Right" will use the right edge of the image. "Center" determines the horizontal center of the image and uses that for moving along the x-axis or resizing along the x-axis.
    - *Offset* – Moves the image along the x-axis based on the selected anchor type.
    - *Size* – Increases or decreases the image size along the x-axis based on the selected anchor type.
- *Vertical*
    - *Anchor* – The options here determine which direct to move image when moving or resizing the image along the y-axis. Options include "Top", "Middle", and "Bottom". "Top" will offset from the top edge of the image or resize from the top edge. "Bottom" will use the bottom edge of the image. "Middle" determines the vertical center of the image and uses that for moving along the y-axis or resizing along the y-axis.
    - *Offset* – Moves the image along the y-axis based on the selected anchor type.
    - *Size* – Increases or decreases the image size along the y-axis based on the selected anchor type.

Scale Bars

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



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

Using the **Create Scale Bar**  tool, draw a box in the graphics window to indicated the initial size of the scale bar. The *Scale Bar Properties* dialog will appear. This dialog can be reached later by right-clicking on the scale bar and selecting the **Properties** command.

In the *Scale Bar Properties* dialog, attributes of the scale bar include:

- *Units* – Options in "Meters", "U.S. Survey Feet", and "International Feet".
- *Font* – Selecting this button bring ups the *Font* dialog where the font type, style, and size are selected. The arrow next to the button will bring up a color picker where the font color can be chosen.
- *Text spacing* – The minimum spacing between distance labels.
- *Min division width* – The minimum division width (in pixels). The XMS application determines the number of divisions based on the minimum divisiion width and the width of the frame.
  - *Background* – Opts to fill behind the scale bar with the background color or another color.
  - *Fill behind* – Toggles on or off the option to create a colored field behind the scale bar.
  - *Background color* – Sets the background color as the same background color selected in for the graphics window in the *Display Options* dialog.
  - *Other color* – Clicking this button will bring up a *Color* dialog where a larger selection of colors can be chosen. The arrow next to the button will bring up a color picker with a preset number of color options.
- *Anchor* tab – This is identical to the *Anchor* tab in the *Image Properties* dialog.

North Arrows

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

A north arrow object is added by using the **Create North Arrow** tool. Properties for a north arrow objects are set in the *North Arrow Properties* dialog. This dialog has the the same options as the *Image Properties* dialog.

Text

Text can be created in world or screen space layers.

Enter text by clicking in the graphics window with the **Create Text** tool active. This will bring up the *Text Properties* dialog. In this dialog, the attributes for the text object can be defined.

The following text attributes can be set:

- *Text* – Options for specifying the text to be displayed and font style.
  - *Text* – Field where the text to be displayed is entered.
  - **Font** – Selecting this button bring ups the *Font* dialog where the font type, style, and size are selected. The arrow next to the button will bring up a color picker where the font color can be chosen.
  - **Color** – Another method for selecting the font color. Clicking this button will bring up a *Color* dialog where a larger selection of colors can be chosen. The arrow next to the button will bring up a color picker identical to the one above.
- *Background* – Fill behind the text with the background color or another color.
  - *Background color* – Sets the background color as the same background color selected in for the graphics window in the *Display Options* dialog.
  - *Other color* – Activates color button identical to the one in the *Text* section.

- *Border* – Contains option for defining a border around the text object.
    - *Draw border* – Activates a border around the entire text object. The color button allows changing the border color.
    - *Specifed thickness* – Activates the option to change the border thickness. The default border thickness is 1 pixel.

Lines/Arrows

Create lines or arrows using the **Create Line** tool. Lines/arrows can be created in screen or world space layers. Lines can be straight or curved. Click once to start a line. Click again to curve the line. Double-click or hit enter to complete a line. Once the line has been completed, the *Line Properties* dialog will appear. This dialog can also be accessed by right-clicking on the line and selecting the **Properties** command.

In the *Line Properties* dialog, the attributes available for lines and arrows include:

- *Line* – Attributes for a line include:
    - *Dashed* – Sets the line as an evenly spaced dashed line. The length of each dash and the length of each space is a set value that cannot be changed at this time.
    - *Solid* – Specifies the line as a solid line.
    - *Width* – Specified the line width.
    - *Line Color* – Clicking this button will bring up a *Color* dialog where a larger selection of colors can be chosen. The arrow next to the button will bring up a color picker with preset color options.
- *Arrowheads* – This section has option for making the line into an arrow.
    - Location – Drop menu that defines where the arrow head will be placed on the line. "None" will leave off arrowheads. "Begin" places the arrowhead where the line was started when created. "End" placed the arrowhead where the line terminated during creation. "Both" places an arrowhead at the start and end of the line.
    - *Length* – Defines the length of the arrowhead. The point of the arrowhead will not go past the start or end point of the line. Increasing the size will move arrowhead further up or down the line.
    - *Width* – Defines the arrowhead width. The arrowhead width will be equally divided along either side of the line.

- *Preview* – Shows what the line or arrow will look like the current selected options. Options are not applied until the **OK** button is clicked.

Rectangles and Ovals

Create rectangles by dragging a box with **Create Rectangle** tool active and create ovals with the **Create Oval** tool active. Rectangles or ovals can be created in world or screen space layers.

After designating where the rectangle or oval will be drawn, the *Rectangle/Oval Properties* dialog will appear. This is the same dialog for both rectangles and ovals. It can also be accessed by right-clicking on the rectangle or oval and selecting the **Properties** command.

IN the *Rectangle/Oval Properties* dialog, the attributes for rectangles and ovals include:

- *Line* – Attributes for a line around the rectangle or oval. Option include:
  - *Dashed* – Sets the line as an evenly spaced dashed line. The length of each dash and the length of each space is a set value that cannot be changed at this time.
  - *Solid* – Specifies the line as a solid line.
  - *Width* – Specified the line width.
  - *Line Color* – Clicking this button will bring up a *Color* dialog where a larger selection of colors can be chosen. The arrow next to the button will bring up a color picker with preset color options.
- *Fill*
  - *Fill* – Designates that the area of the rectangle or oval will have a solid color. This activates a color button like the one in the *Line* section.
  - *No fill* – Designates that the area of the rectangle or oval will be empty.

Selection

The **Select Annotation Object** tool is used to select and set attributes for annotation objects. This requires that objects exist to be selected. In this case when using this tool and left-clicking in the annotation object, the object frame will be drawn around the annotation. In addition to the frame, the XMS application displays grab handles on the corners and edges of the frame. Modify the rectangular shape of the annotation by dragging one of the grab handles and changes the position of the object by dragging the annotation (click at any point in the object interior).

Right-clicking on annotation object will produce a menu with the following commands:

- **Delete** – Removes the annotation object.
- **Duplicate** – Creates a copy of the annotation object on the same annotation layer.

- **Properties** – Brings up the properties dialog for the selected object.

Viewing Annotations At Specific Time Intervals

Available in SMS v11.1 and higher, annotations can be setup to be viewed at specific time intervals. This feature is currently under development in GMS. To setup annotations so they only are displayed at specified time intervals do the following:

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

Annotations will not be displayed when the specified time range is active. This applies to data in the graphics window and film loops.

3.1. 1D Grid Module

1D Grid Module

The 1D grid module is used to display 1D coastal data.

A 1D grid can be created from a map coverage using the **Create 1D Grid Frame** tool. A 1D grid is oriented with the water on the left and the land to the right. For example, if the 1D grid was oriented from north to south, the water would be to the east (left) and the land would be to the west (right). The dimensions of the 1D grid are specified using the grid frame .

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

1D Grid Module Tools

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

- **Select Point** – Selects or drags a point on the initial shoreline.
- **Create Point** – Create a point along the coastline.
- **Select Detached Breakwater** – Edits a breakwater positioned on the grid.
- **Create Breakwater** – Creates a breakwater in a simulation.
- **Select Jetty or Groin** – Edits the length of a groin or jetty.
- **Create Jetty or Groin** – Creates a groin in a simulation.
- **Select Seawall** – Edits the shape of an existing seawall.

- **Create Seawall** – Creates seawall segments along the grid.

See 1D Grid Module Tools for more information.

1D Grid Module Menus

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

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

Related Topics

- 1D Grid Display Options
- Modules
- GenCade

1D Grid Display Options

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

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

Grid Objects

- *Y Scale Factor* – Magnification in the direction perpendicular to shore
- *Grid* – Shows or hides the 1D grid line.
- *Initial shoreline* – Shows or hides the arc used as the initial coastline.
- *Current shoreline* – Based on dataset.
    - *Shoreline points* – Show or hides points in use along the shoreline arc.
- *Minimum Shoreline*
- *Maximum Shoreline*
- *Minimum/Maximum shoreline envelope* – Shows or hides zone covered by transient coastline and extremes based on dataset.
- *Reference shoreline* – Shows or hides the shoreline arc used as a reference for the grid.
- *Regional contour* – Shows or hides the arc used as the regional contour shoreline.

Structure Objects

- Seawalls
- Groins
- Breakwaters
- Bypass Cells
- Beach Fills
- Inlets

- SBAS Polygons
- SBAS Flux

For more information see the article GenCade Structures .

Related Topics

1D Grid Module

1D Grid Tools

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

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

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

The tools include:

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

-  **Select Seawall** – This tool is used to edit the shape of an existing seawall along the grid. With this tool active, a click in the graphics window selects a point in a seawall. Dragging the mouse with the point selected modifies the seawall. As with grids, seawalls often include multiple segments making them tedious to edit using this approach.
-  **Create Seawall** – This tool can be used to create seawall segments along the grid. With this tool active, a click in the graphics window starts the creation of a segment. A second click terminates the segment. When the segment is complete, SMS determines if this segment overlaps existing segments and trims the existing segments to the new segment if an overlap exists.

Related Topics

- 1D Grid Module

3.2. Cartesian Grid Module

Cartesian Grid Module

At a glance

- Used to create, edit, and visualize rectilinear grids
- Datasets can have values at cells, corners, and midsides
- Can use cell-centered or mesh-centered grids

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

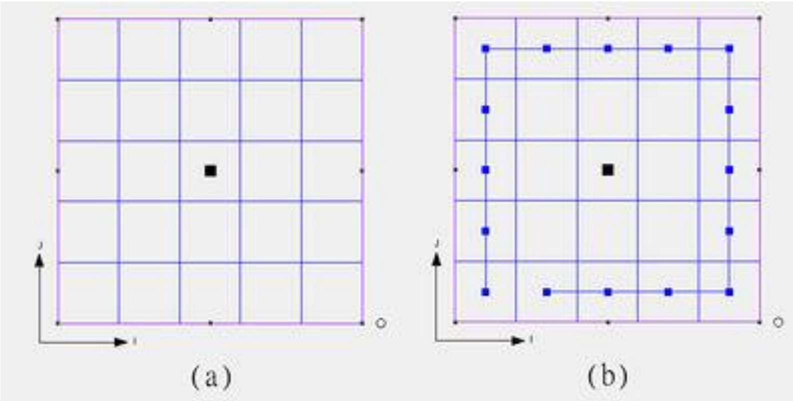
Some models limit the grid to be defined with square cells, others limit to constant sized rectangular cells, while others add the flexibility of having variable sizes to the cells (variable row height or column width.

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

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

The Grid module is included with all paid editions of SMS.

Grid Types



Multiple types of grids are supported in the 2D Grid module because each numerical engine has its own limitations and supported features. Variations include:

- Mesh centered -vs- Cell centered. This refers to where data values are associated with a grid. For mesh centered grids, values are assumed to be at the intersection of lines or cell corners. The BOUSS-2D model is an example of this type. For cell centered grids, values are associated with a cell in general and assumed to be at the cell centroid. STWAVE and CMS-Wave are example of this type. In reality, this is just a visualization difference, but in practice it can get complicated because a numerical model may have some data expected as grid centered and other data as cell centered. There is also an option for some types of data to actually be associated with a face of the cell such as the vertical (or J direction) flow being associated with the top face and the horizontal (or I direction) flow associated with the left face. This is the situation with the CMS-Flow engine.
- Rotational limitations. Some numerical engines require that the grid be a true Cartesian grid, meaning that it is not rotated. The WAM model requires this condition. Other engines operate in their own local space, so while the I/J directions of the grid do not necessarily correspond to a global orientation (such as East and North), they are still considered Cartesian space from the numerical analysis perspective.
- Cell shape limitations. Some numerical engines require that all cells be square. WAM is an example of this. Earlier versions of STWAVE had this restriction but now supports rectangular cells.
- Refinement limitations. Some numerical engines allow the cells to change aspect ratio. Obviously, in order for this to be supported, the engine must also support non-square cells. The CMS models are the only engines supported in SMS that allow this type of grid. The SMS interface allows creating refine points in a Cartesian grid coverage to specify the desired row width (and/or cell height) at a specific location in the domain. The grid generation method will create cells that match the smallest specified dimension and grow at a user specified rate to a maximum dimension.
- Telescoping or QuadTree grids . The CMS-Flow engine also supports computation on a telescoping grid or quad tree. In this geometric object, specify a constant resolution grid as a background grid (either number of rows/columns or a fixed width/height), and then specifies refinements as a maximum allowable dimension in a polygonal zone. This is described more in the Telescoping Grids definition.
- Projection limitations. Most numerical engines are designed to work in a specific type of coordinate system or projection. The WAM model only operates on a grid in geographic space. The other numerical engines currently in the SMS interface require a Cartesian space such as UTM or State Plane. Some responsibility is left to ensure that the projection being used in a simulation is compatible with the numerical engine being employed.

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

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

Creating and Editing 2D Grids

Creating 2D Grids

The two main techniques used to create 2D grids are: the *Map → 2D Grid* command and the *Create Grid* command. It is encouraged to utilize the Map Module rather than generating grids directly in the Cartesian grid module because the map module creates a record that can be edited and repeated.

Cartesian Grid Generator Coverage

Starting with SMS 12.0, a generic Cartesian Grid Generator coverage or CGrid Generator coverage can be used with the Map module. This coverage allows access to the **Create 2D Grid Frame** tool to frame feature objects in the coverage. Once the grid frame has been created, the feature objects in the coverage can be converted to the Cartesian Grid module by using the **Map → 2D Grid** command.

When creating the CGrid Generator coverage from the *New Coverage* dialog, the *Select Grid Type* dialog will appear before the coverage is created in the project explorer. This dialog gives the option to create the coverage as "Cell-centered" or "Mesh-centered". The selection should depend on which model the feature objects will be used in after conversion to the Cartesian Grid module.

Map → 2D Grid

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

Parameters specified to create the grid include:

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

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

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

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

If the grid is intended for CMS-Flow , and telescoping resolution is specified in at least one feature polyon, SMS will generate a recursively refining grid based on the input parameters.

Create Grid

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

Editing 2D Grids

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

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

Smoothing 2D Grids

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

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

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

Converting 2D Grids

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

Project Explorer

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

Cartesian Grid Module Root Folder Right-Click Menus

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

- Display Options

Cartesian Grid Item Right-Click Menus

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

- **Smooth** – Opens the *Cartesian Grid Smoothing Options* dialog.

Model Specific Right-Click Menus

- **Create Transformed Grid**

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

Related Models: CMS-Wave , STWAVE

Cartesian Grid Module Tools

See Cartesian Grid Module Tools for more information.

Cartesian Grid Module Menus

See Cartesian Grid Module Menus for more information.

How do I?

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

Related Coverages

The grid module currently includes interfaces for:

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

Related Topics

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

Cartesian Grid Coordinates

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

Editing the Grid

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

Changing the Grid Projection

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

Floating Projection

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

Related Topics

- SMS:Cartesian Grid Module

Cartesian Grid Data Menu

Most of the SMS modules have a *Data* menu, but the items in this menu are different for each module.

The *Cartesian Grid Module Data* menu commands include:

- **Steering Module** – Launches the steering tool.
- **Dataset Toolbox** – Contains tools for working with datasets. Includes the *Data Calculator* .
- **Switch Current Model** – Changes current active model.
- **Vector Options** – Opens a dialog where options to generate vectors can be edited.
- **Contour Options** – Opens a dialog where dataset specific contour options can be defined.
- **Set Contour Min/Max** – This command sets the contour options based on the current options and the selected nodes/vertices or zoom level.
- **Contour Range Options** – This controls if the **Set Contour Min/Max** command applies to dataset specific contour options or the general contour options (for the mesh or scatter modules). It also sets the flags for precision and fill above and below.
- **Film Loop** – Opens the *Film Loop Setup* wizard.
- **Grid → Scatterpoint** – Converts grid data into the scatter module.
- **Grid → Map** – Converts grid data into a map coverage.
- **Grid → Mesh** – Converts grid data into the mesh module.
- **Find Cell** – Used to locate a cell either with a specific i,j location, or near a specific location.
- **Map Elevation** – Allows use of another functional dataset as the mapped elevation function.
- **Zonal Classification** – Tool to identify areas that meet as set of criteria.

Model Specific Menus

The following models have specific commands included in the *Data* menu.

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

Data Conversion Commands

Grid to Scatterpoint

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

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

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

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

Grid to Map

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

The command includes the following options:

Land/Water Boundary → Arcs

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

Grid Boundary → Arcs

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

Observation Cells → Points

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

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

Grid to Mesh

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

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

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

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

Switch Current Model

The **Switch Current Model** command brings up a *Select Current Model* dialog. This dialog can only be used when a grid does not currently exist.

Find Cell

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

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

Obsolete Commands

- Create Datasets – Opens a dialog that can be used to create functions for the entire mesh or active scatter set. No longer available as of SMS 10.1. Replaced by the *Dataset Toolbox* .

Related Topics

- Cartesian Grid Module

Cartesian Grid Module Display Options

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

Cell display

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

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

Cell String display

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

Grid display

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

- *Contours* – The contours are drawn for the active scalar dataset. All standard contour display options are supported for cartesian grid contours.
- *Vectors* – The cartesian grid vectors are drawn for the active vector dataset. All standard vector display options are supported.
- *Grid Boundary* – A line around the perimeter of the cartesian grid can be drawn. This is useful when the cells are turned off. Line color and thickness can be specified.
- *Computational Domain* – It can be useful to delineate the active (or water cells) from the rest of the grid. This option allows for specification of a line to outline the computational cells.
- *Cell id* – Displays the ID for each cell. The text font, color, and size can be specified.
- *Cell i, j* – The i,j coordinate can be drawn in each cell.
- *Elevations* – The current scalar value can be drawn in each cell.
- *IJ triad* – Arrows can be displayed at the origin of the grid showing the i and j directions.
- *Display Inactive Grids* – SMS supports the ability to store/manipulate multiple Cartesian grids at the same time. This allows for functionality such as nested grids in STWAVE and steering between STWAVE and M2D. However, only one grid can be edited at a time. This is the "active" grid. The outline of other grids can be displayed using the *Display Inactive Grid* option.
- *Functional Surface* – Show surfaces representing one of the functional datasets associated with a mesh, grid or TIN.

Model specific options

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

Related Topics

- Cartesian Grid Module
- Display Options

Cartesian Grid Tools

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

Tool	Tool Name	Description	Right-Click Menu
	**Select Cell**	The **Select Cell** tool is used to select a grid cell. A single cell is selected by clicking on it. A second cell can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple cells can be selected at once by dragging a box around them. A selected cell can be de-selected by holding the *SHIFT* key as it is clicked. When a single cell is selected, its Z coordinate is shown in the *Edit Window* . The Z coordinates can be changed by typing in the edit field, which updates the depth function. If multiple cells are selected, the Z Coordinate field in the *Edit Window* shows the average depth of all selected cells. If this value is changed, the new value will be assigned to all selected points. With one cell selected, the *Edit Window* shows the point i,j location. With multiple cells selected, the *Edit Window* shows the number of selected cells. The number and size of the cells can be changed in the *Model Control* .	• **Interpolate Bathymetry...** – Requires one or more Cartesian grid cells to be selected, and to have a scatter set in the project, This option brings up the *Interpolation* dialog where the desired source scatter dataset can be selected. When **OK** is clicked, SMS will interpolate the selected cell(s)' elevations based on the chosen scatter dataset. • **Cells to Active Coverage** – Converts the selected cells into polygons on the current active coverage.
	**Select Row**	The **Select Row** tool is used to select cell rows. Rows are selected in the same manner as selecting individual cells.	N/A
	**Select Column**	The **Select Column** tool is used to select cell columns. Columns are selected in the same manner as selecting individual cells.	N/A
	**Split Grid Column**	Inserts a new column into an existing grid. This tool splits an existing column into two columns at the selected location selected.	N/A
	**Split Grid Row**	Inserts a new row into an existing grid. This tool splits an existing row into two rows at the selected	N/A

		location.	
	Drag Column Boundary	Edit column boundary. This tool makes one column narrower while making its neighbor wider.	N/A
	Drag Row Boundary	Edit row boundary. This tool makes one row taller while making its neighbor shorter.	N/A
	Select Cell String	Select a “Cell String”. Allows assignment of boundary conditions.	N/A
	Create Cell String	Create a “Cell String”. This tools defines a string of cells for later assignment of boundary conditions or flux observations.	N/A
	Label Contours	Used to manually create a contour label using the mouse. To add a label, click on the point where the label should be created. The label will remain on the screen until either it is manually removed or the automatic contour label options are changed. To manually remove a contour label, hold the *SHIFT* key and clicking on it. There are also available automatic contour label options.	N/A

Interactive options

- **Move Frame** – Click inside or on an edge where the frame is not highlighted and drag.
- **Resize Frame** – Click a highlighted corner or edge and drag to resize.
- **Rotate Frame** – Click inside the circle near the bottom right corner of the frame and drag to rotate the frame.
- – Redraw the screen.
- – Zoom to the extents of the data in the screen.

While graphically manipulating the grid frame, the current values of origin, orientation and/or size are displayed at the bottom of the graphics window.

Related Topics

- SMS:Cartesian Grid Module

Grid Frame Properties

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

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

- *Origin X/Y* – Set the bottom left corner location of the grid frame.
- *Orientation*
  - *Angle* – Set the angle the grid frame will be rotated counter-clockwise from the +x axis.
  - *I/J size* – Set the length and width of the grid frame. The cell dimensions are updated: Cell X = Grid X / Number of Columns, Cell Y = Grid Y / Number of Rows.
- *Cell Options* – These options vary based on the grid type being generated. A few of the options are illustrated below. Specify the size of the cells to be created in each of the applicable directions from the following options:

  - *Cell size* (or *Base cell size* if telescoping is an option) – Set the dimension (width or height) of each cell for the grid to be created.
  - *Number of cells* – Set the number of rows and/or columns for the grid to be created.
  - *Use refine points* – Tell SMS to use specified refine points in a CMS-Flow or CMS-Wave coverage. This will result in rectangular cells with variable aspect ratio.
    - *Maximum cell size* – The max size the should exists when growing
    - *Maximum bias* – The max growth ratio to be used when growing
    - *Use inner growth* – Specifies whether the cell sizes should grow between two refine points
- *Grid size* – The grid dimension in the specified direction

- *Adjust base cell size* – This button appears only for CMS-Flow coverages when there are feature polygons with target maximum size guidelines specified. Clicking on this button will update the base cell sizes so that the larger of the two is a multiple of the minimum feature size that is to be represented in the grid. For example, if having specified 20 m as the maximum cell size (to allow for 5 cells to represent a channel that is 100 m across), and the base cell size is 75 m in both directions, this button will increase the base cell size to 80 meters in both directions. Therefore the smallest cell generated will be 20 m by 20 m, whereas, a base cell size of 75 m would result in a cell 18.25 m on each side. The objective is to make the base cell size correspond to the smallest user specified target cell size.
- *Options* – Set the interpolation options for the Scatter applications. Not used for the Cartesian or 1D Grid applications.

When specifying *Define cell sizes* , there are a few options available. These options are:

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

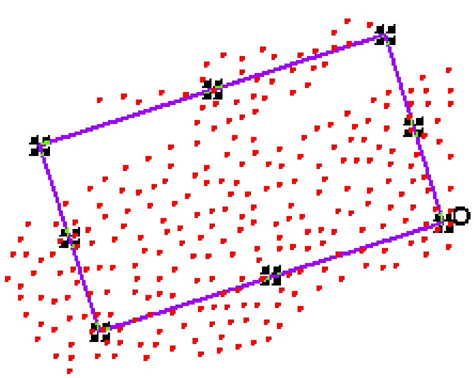
If the grid is to have square cells, the v direction cell size will always be linked to the u direction cell size.

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

They are also created temporarily in a number of functions in the SMS interface including:

- Creating a grid in the Grid Module
- Interpolating from a scatter set to a scatter grid (*Scatter* | **Interp to Scatter Grid** in the Scatter module).
- Interpolating from a scatter set to a nautical grid (*Scatter* | **Interp to Nautical Grid** in the Scatter module).

Grid frames consist of a rectangular shape, normally defined by three graphical clicks, which can be oriented and stretched using handles on the sides and corners as shown in the following image to create an arbitrary rectangle shape.



Depending on the application of the grid frame, it will have a variety of attributes or properties.

Interact with the grid frame when the **Select Grid Frame** tool is active, or define grid frame properties explicitly using the *Grid Frame Properties* dialog (this dialog is accessible by right-clicking on the grid frame or when converting to a grid.

Related Topics

- Map Module Tools
- Feature Objects Menu

Grid Smoothing

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

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

Related Topics

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

Refine Point Dialog

The *Refine Point* dialog is used to set the attributes for a refine point represented by a feature point in a Cartesian Grid model coverage. Refine points for a Cartesian Grid allow changing the cell dimensions when generating the grid. The dialog is reached by selecting a point in a Cartesian grid coverage and selecting the *Feature Objects* | **Attributes** command.

Attributes that can be specified for each refine point include:

- *Refine grid in I direction* – Has the following option when turn on:
  - *Base cell size* – Specify the cell I size in the vicinity of the refine point.
- *Refine grid in J direction* – Has the following option when turned on:

- *Base cell size* – Specify the cell J size in the vicinity of the refine point.

Only refine points located within a grid frame are used when the **Map → 2D Grid** command is executed. Refine points are not available for all models, since some Cartesian Grid models require uniform cell sizes. When the refining is performed, the base size may be changed in order to fit the other restrictions applied to the refining process. If two refine points are too close to each other to allow the cell size to transition, one will be ignored when generating the grid

1D Grid models can also make use of a refine point. For information on this, see GenCade .

Related Topics

- Feature Objects Menu

3.3. Curvilinear Grid Module

Curvilinear Grid Module

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

Nodes

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

Cells

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

Delete Cells

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

Add Cells

Cells can only be added to an existing Curvilinear grid.

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

Nodestrings

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

Delete Nodes (Vtk Mesh only)

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

Merging Two Curvilinear Grids

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

To do a merge:

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

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

Models

The curvilinear grid module currently includes interfaces for:

- LTFATE

Tools

See Curvilinear Grid Module Tools for more information.

Menus

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

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

Display Options

See Curvilinear Grid Display Options for more information.

Related Topics

- Curvilinear Grid Display Options
- SMS Modules

Curvilinear Grid Display Options

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

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

Model Specific Options

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

Related Topics

- Curvilinear Grid Module
- Display Options

Curvilinear Grid Module Tools

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

Tool	Tool Name	Description	Right Click Menu
	**Select Curvilinear Grid Node**	The **Select Curvilinear Grid Node** tool is used to manually select and edit an individual node location. Currently, the only reason to select a grid node is to adjust the shape of the adjacent (4) elements. The status bar at the bottom of the screen displays the i, j, and id of the selected node. The current position of the node is displayed in the edit fields at the top of the screen. (Dragging of curvilinear grid nodes was added for SMS 11.2.)	N/A

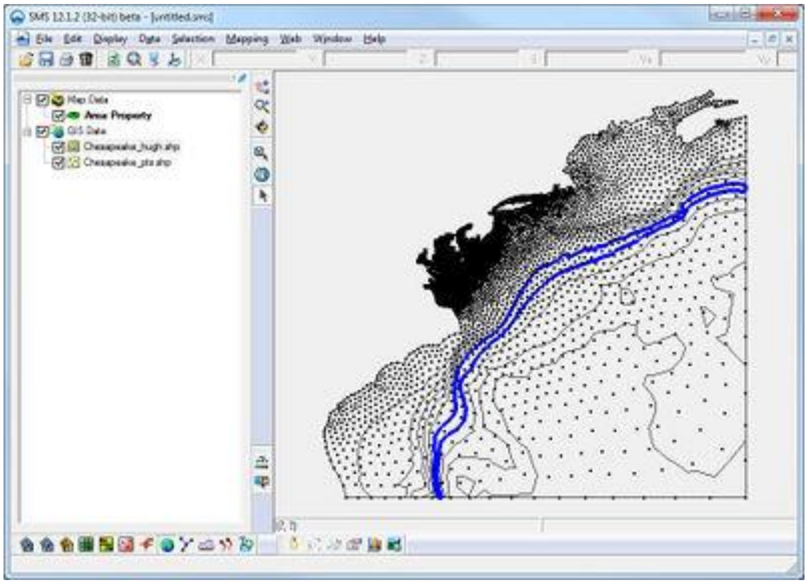
	**Select Cell**	The **Select Cell** tool is used to select a grid cell. A single cell is selected by clicking on it. A second cell can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple cells can be selected at once by dragging a box around them. A selected cell can be de-selected by holding the *SHIFT* key as it is clicked. When a single cell is selected, its Z coordinate is shown in the Edit Window . The Z coordinates can be changed by typing in the edit field, which updates the depth function. If multiple cells are selected, the Z Coordinate field in the *Edit Window* shows the average depth of all selected cells. If this value is changed, the new value will be assigned to all selected points. With one cell selected, the Edit Window shows the point i,j location. With multiple cells selected, the Edit Window shows the number of selected cells. The number and size of the cells can be changed in the *Model Control* .	When one or more curvilinear grid cells are selected, a right-click in the graphic window will bring up a menu. The menu has the following commands: • **Delete** – Delete the selected cell(s). • **Set Observation Station** – Bring up the *Observation Station* dialog.
	**Select Element**	The **Select Element** tool is used to select a cell (or element) of a curvilinear grid. Currently, the only reason to select a cell is to delete it from the grid and create a hole (or notch) in the grid. This would be done when the cell covers an island or other region that should be excluded from the computational domain. The status bar at the bottom of the screen displays the i, j, and id of the selected cell as well as the area of the cell. (Deleting of curvilinear grid cells was added for SMS 11.2.)	
	**Select Nodestrings**	The **Select Nodestrings** tool is used to select nodestrings. Nodestrings currently serve no purpose for curvilinear grids.	
	**Create Nodestrings**	The **Create Nodestrings** tool is used to create node string. Nodestrings currently serve no purpose for curvilinear grids.	N/A
	**Split Grid Column**	Inserts a new column into an existing grid. This tool splits an existing column into two columns at the selected location.	N/A
	**Split Grid Row**	Inserts a new row into an existing grid. This tool splits an existing row into two rows at the selected location.	N/A

Related Topics

- Curvilinear Grid Module

3.4. GIS Module

GIS Module



At a glance

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

The GIS module allows managing geographical information data inside of the SMS.

It has traditionally contained external data such as ArcInfo Shapefiles and MapInfo MIF/MID files. This data can be converted to Map Module Features for various purposes such as inclusion in a conceptual model to create a mesh/grid, or for inclusion in a simulation.

SMS includes two separate ways of working with or managing external GIS vector data. The first uses internal SMS functionality to select, edit and convert GIS data to other data types in SMS. The second includes a link to ArcObjects to allow direct management of the GIS data. This option is described below.

Starting with SMS version 12.0, the GIS module was modified to be consistent with the Groundwater Modeling System (GMS) and the Watershed Modeling System (WMS). With this merging of functionality, the GIS module is used to manage all external geographical data including:

- Images (*.tif, *.jpg, *.png, *.bmp, *.pdf, *.ecw, *.sid, *m.00 (ENC), ...)
- Rasters/DEM (*.dem, *.flt, *.asc, *.bil, *.tif, *.ace, ...)
- GIS (*.shp, *.mif/mid, *m.00 (ENC), ...)
- Lidar Surveys (*.las)

When opening a file recognized as a GIS data type, SMS will create an entry in the project explorer for that object. The object may contain any combination of the following:

- image data (colors on a grid)
- raster data (values on a grid)

- vector data (vector objects)
- GIS objects (data recognized as being native to ArcInfo, MapInfo, QGIS)

In addition to data type, the data source for a GIS object may be static (a single file) or dynamic (a link to a web site).

The icon, displayed to the left of the entry of the tree indicates what type(s) of data reside in that object and whether the object is static or dynamic.

By right-clicking on a GIS object in the project explorer, the popup menu shows the applications for the specific data type contained in the selected object.

See GIS Module Right-Click Commands for more information.

Sample applications of the GIS module include:

- Display Extract a subset of data from a large file for inclusion in a simulation
- Display of background information (such as an aerial photo or map)
- Provide topographic/bathymetric data sources (digital elevation map) which can be used in place of a TIN or scatter set when creating a mesh/grid.

Some of the key functionality available in the GIS module includes:

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

The GIS module is included with all paid editions of SMS.

Interpolation

To interpolate from a DEM or image with elevation data, right-click on the object and select *Interpolate* | <*target*> (where <*target*> defines the object the values are being interpolated to such as a scatter set, grid or mesh). The principal application of this is to assign geometric elevations or depths to the geometry.

Another application is to assign spatial attributes based on land use attributes, land cover attributes, or another set of values on the DEM. This is currently available in the *ADCIRC* menu under the **Nodal Attributes** command. Selecting the land use coverage to populate a specific nodal attribute will average the values over the area of interest.

Conversion

These commands create new geometric entities in the SMS project such as a scatter set or feature objects. To convet from a DEM (image with elevation data), right-click on the object and select *Convert* | <*target*> (where <*target*> defines the object being created such as a scatter set or grid.

See GIS Conversion and Editing for more information.

Editing

These commands allow manipulating the DEM objects themselves and are accessed by right-clicking on the GIS object.

See GIS Conversion and Editing for more information.

Edit Values

This command uses the selected arcs or nodes in the active coverage to edit the elevation of cells in the DEM. To use the command:

- Create arcs/nodes at the locations where an edit to the DEM is desired.
- Specify the desired elevation(s) as the elevations of the feature objects.

- Select feature objects to force into the DEM
- Right-click on the DEM and select the **Edit** command.

Importing Shapefiles

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

For additional information, see Importing Shapefiles .

GIS Module Tools

See GIS Module Tools for more information.

GIS Module Menus

See GIS Module Menus for more information.

Using the GIS Module with a License of ArcGIS®

SMS includes the option to use ArcObjects to incorporate much of the ArcGIS/ArcMap/ArcView® functionality directly. Any ArcGIS® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) can be opened, then used with the *ArcGIS® Display Symbology* properties to render the GIS data, and then display it in SMS.

To use SMS (32-bit only) with ArcGIS®, do the following:

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

Most of the same functionality that exists with licenses of ArcGIS® is also available without a license and a license is **NOT** required for most applications.

Related Topics

- Map Module
- ArcObjects
- Shapefiles
- Modules
- GIS Conversion and Editing

Importing Shapefiles

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

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

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

Direct Conversion of Shapefile Data to Coverages

The first is the traditional method which allows loading a shapefile layer directly into a coverage.

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

Using the GIS Module to Convert Shapefile Data to Coverages

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

Cleaning Imported Shapefile Data

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

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

Related Topics

- Shapefiles

GIS Module Tools

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

Select ArcObject

The **Select ArcObject** tool is used to select ArcObjects in the Graphics Window. Selected objects can be mapped to other data types. This tool is only available if ArcView is installed locally and enabled in SMS.

Get Attributes

The **Get Attributes** tool is used to select GIS objects and display information relating to that object in an *Info* dialog.

Select

The **Select** tool is used to select shape objects in the Graphics Window. Selected objects can be mapped to other data types.

Unlike other modules in SMS, the GIS module tools do not have any right-click menu commands.

Get Attributes Tool Info Dialog

Clicking on an object when using the **Get Attributes** tool will bring up the GIS attributes *Info* dialog. The attributes shown in this dialog are based on the GIS file data. Attributes cannot be edited in this dialog. The dialog is closed by selecting another tool.

Related Topics

- GIS Module

GIS Module Menus

The following menus are available in the the GIS Module :

Standard Menus

See SMS Menus for more information.

Module Specific Menus

Data

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

Enable ArcObjects

ArcObjects® is a development platform provided by ESRI that allows developers of other applications (such as SMS 32-bit) to incorporate ArcView/ArcGIS® capability directly within their application. SMS can use ArcObjects® to access some of the same functionality in SMS that is available in ArcGIS®, providing SMS is running on a computer that has a current license of ArcGIS®.

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

Add Data

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

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

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

Add Shapefile Data

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

Add MIF/MID File Data

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

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

Layer Properties

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

Map Properties

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

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

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

The WMS article WMS:Map Properties has more information about this dialog.

Selection

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

Command	Description	ArcObjects® Required
Select by Attributes	Opens the *ArcObjects® Query Wizard* Dialog. See the ESRI ArcGIS documentation for further explanation of the *Query Wizard* dialog.	Yes
Select by Location	Opens the *ArcObjects® Select By Location* Dialog. See the ESRI ArcGIS documentation for further explanation of the *Select By Location* dialog.	Yes
Clear Selected Features	Clears the current selection	Yes

Interactive Selection Method	Change the ArcObjects® selection options in use. Options include: • Create New Selection • Add to Current Selection • Remove from Current Selection • Select from Current Selection	Yes
Selectable Layers	Opens the *ArcObjects® Check Selectable Layers* dialog. The layers to be selected can be specified.	Yes

Mapping

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

Command	Description	ArcObjects® Required
Arc Objects → Feature Objects	Opens the *GIS to Feature Objects Wizard*	Yes
Shapes → Feature Objects	Opens the *GIS to Feature Objects Wizard*	No
Feature Objects → Geodatabase	Saves the Feature Objects as a Personal Geodatabase file (*.mdb)	Yes
Polygons → TIN	Converts polygon data to TIN data that will appear in the scatter module.	No

GIS Module Right-Click Commands

The following right-click commands are available for GIS objects when ArcObjects are not enabled:

- **Remove** – Deletes the GIS object from the project.
- **Transparency** – Brings up the *Layer Transparency* dialog.
- **Zoom to Extents** – Frames and centers in the graphics window the extents of the GIS layer.
- **Open Containing Folder**
- **Projection** – Brings up the *Object Projection* dialog.
- **Register Image** – Brings up the *Register Image* dialog.
- **Export** – Resamples and saves the image.
- **Export World File** – Exports data as a world file that includes image registration data.
- *Interpolate To* – A sub-menu for commands to interpolate the GIS object into one of the following"
  - **2D Mesh**
  - **2D Scatter** (both as regular scatter set and as land use data for ADCIRC nodal attributes)
  - **2D Grid**
  - **UGrid**
  - **Active coverage**
- *Convert To* – A sub-menu with commands to convert GIS objects.
  - **2D Scatter**
  - **2D Grid**
  - **Feature Contours** (contour(s) of the data in the DEM)
  - **Feature Contours at Given Elevation**

  - **Resampled Raster** – Creates a new DEM covering the bounds of the selected DEM with a specified spacing.
  - **Smoothed Raster** – Smooths the values on the DEM based on use specified weighting functions.
  - **Trimmed Raster** – Trims to the bounding box of a coverage or selected polygons.
  - **Merged Raster** – Appears when multiple rasters are selected. It creates a new raster that covers all selected rasters with a resolution set to the smallest cell size of any of the selected rasters.

  See the article GIS Conversion and Editing for more information.

- *Editing* – A sub-menu with the following commands:
  - **Arc Elevation Profile** – Brings up the *Profile Elevations* plot for a selected arc.
- **Convert to TUFLOW Rainfall Boundary Conditions**

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

- **Open Attribute Table**

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

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

- **Join Table to Layer**

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

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

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

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

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

Related Topics

- GIS Module
- GIS Conversion and Editing

GIS Conversion and Editing

GMS, SMS, and WMS can load GIS data such as digital elevation models (DEMs) and images. This GIS data will appear in the GIS Data section of the project explorer.

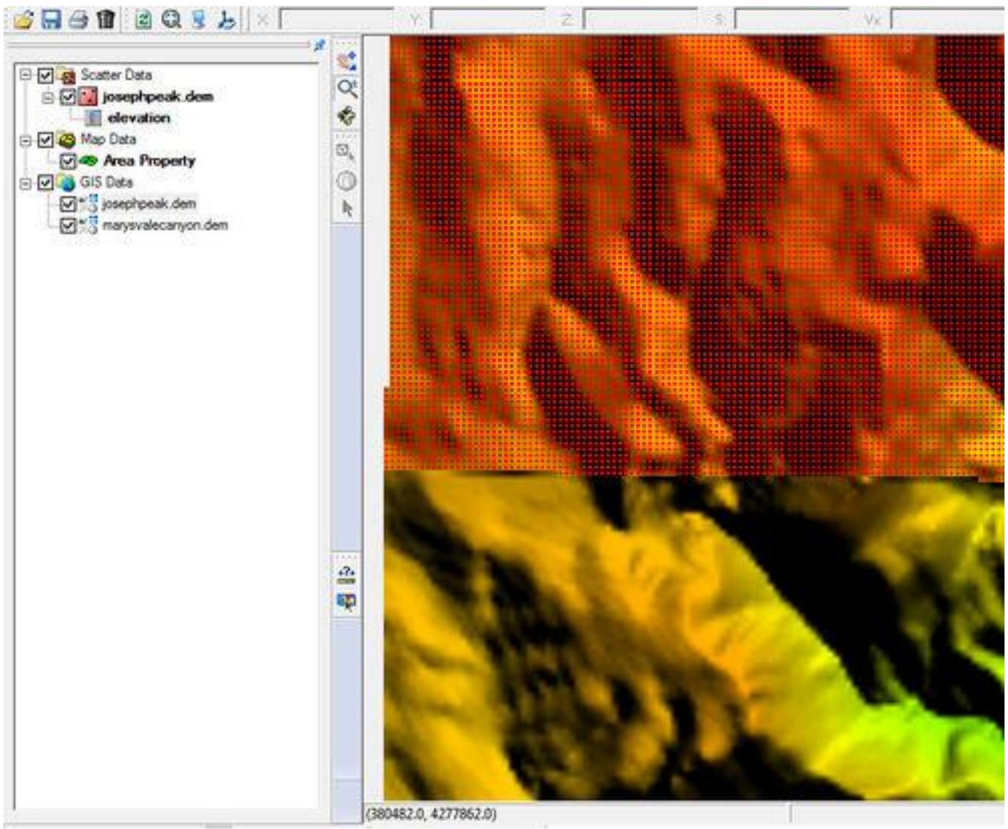
GIS Data Conversion

Both GMS and SMS offer methods to convert GIS data. Right-clicking on a GIS data item in the project explorer brings up a menu with a *Convert To* sub-menu. The commands in the *Convert To* sub-menu are:

Raster to 2D Scatter

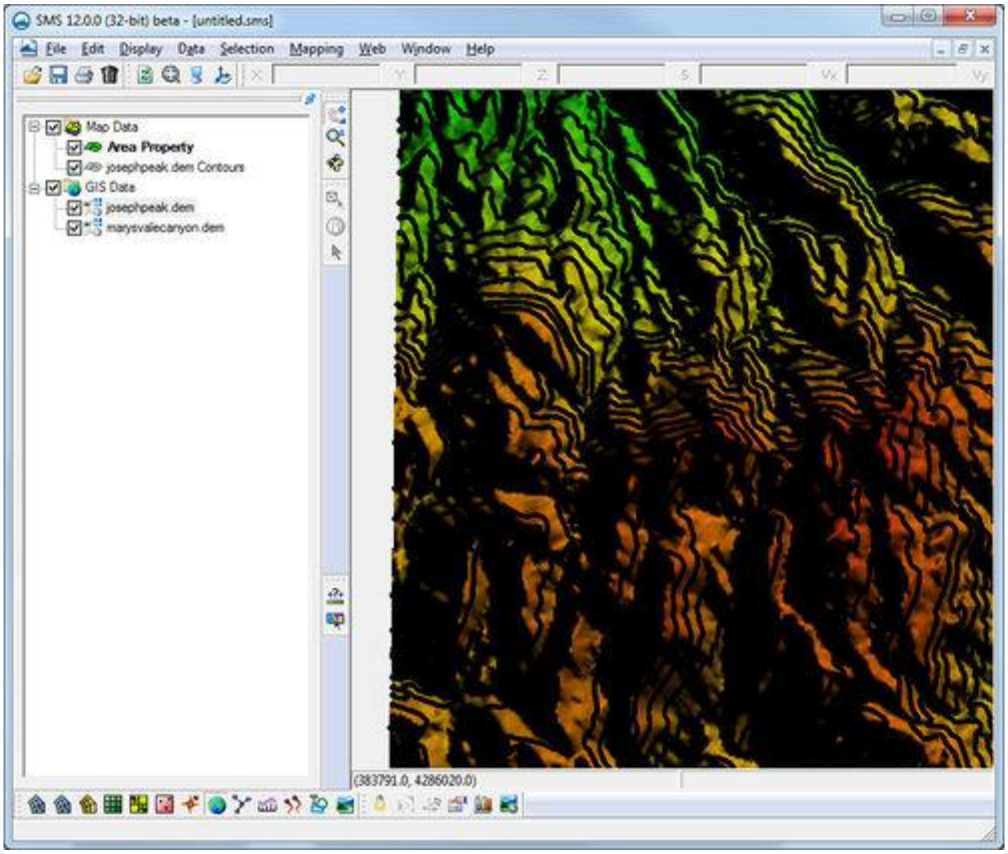
Selecting the command *Convert To* | **2D Scatter** will bring up the *Raster → Scatter* dialog. The dialog shows the number of scatter point that will be generated in the new set and allows naming the new scatter set. By default, the name of new scatter set will be the same as the raster set unless changed.

By default scatter points are shown in red unless changed in the *Display Options* dialog. It may necessary to zoom in to see each point. Scatter points are visible in other views while the DEM is visible only in plan view.



Feature Contours

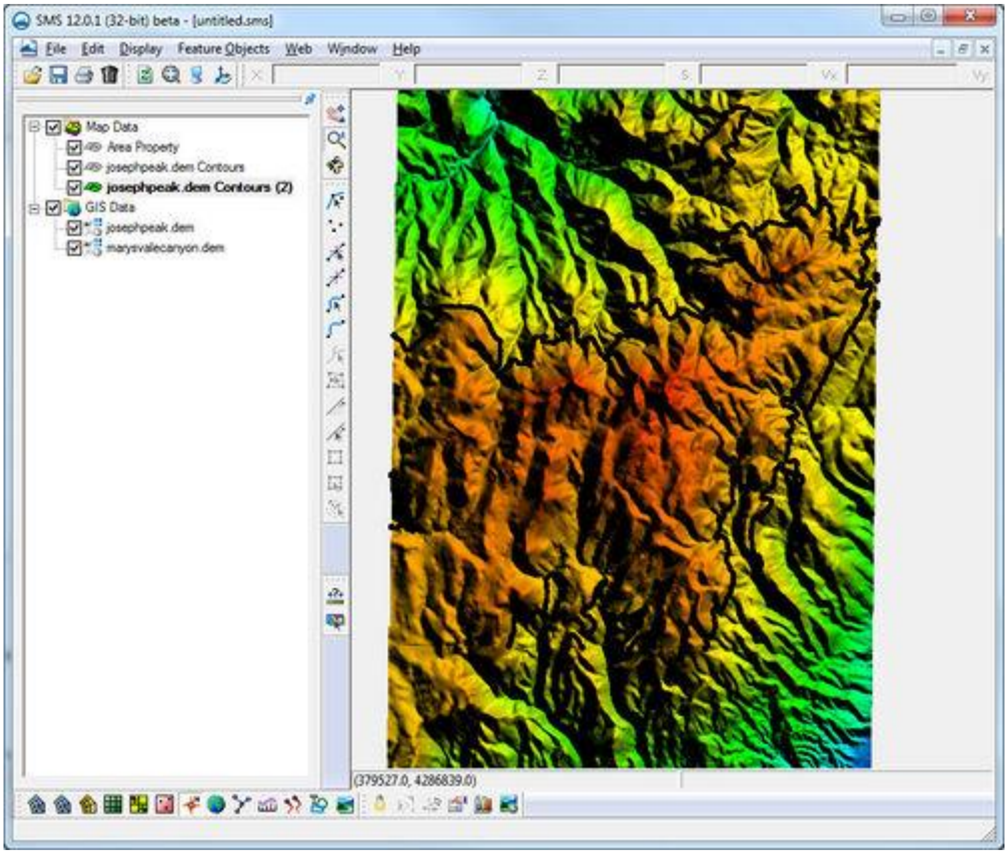
Activating this command will automatically generate contours which will appear as a new coverage under Map Data in the project explorer. By default, contour lines are shown in black unless the contour color is changed in the *Display Options* dialog. The contour interval is found by creating ten contours evenly spaced between the min and max dataset values.



Feature Contours at Given Elevation

This command will bring up the *Create Contour Arc* dialog. In this dialog, specify an elevation to be used in generating the contours.

After entering an elevation and clicking **OK** , contours will be generated similar to those created with the **Feature Contours** command but using the specified elevation. Only contours at the specified elevation will be generated.



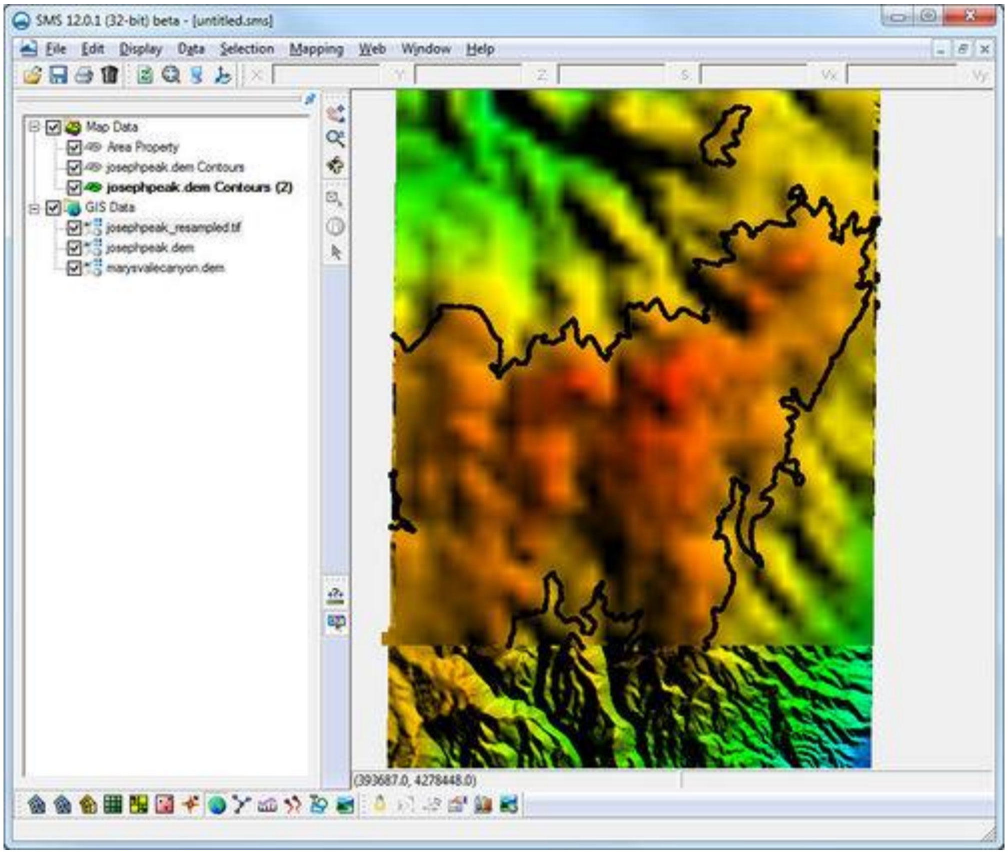
GIS Data Editing

Resampled Raster

This command will bring up the *Resample and Export Raster* dialog. Resampling creates a new DEM covering the bounds of the selected DEM with a specified spacing. The dialog has the following options:

- *Cell size* – Specify the size of each cell based on the *Resampling ratio* .
- *Resampling ratio* – Specifies the resampling ratio. This is required for processing. Changing this field will change the other fields in the dialog automatically.
- *Num pixels* X – The number of pixels on the x axis that will be generated. The original number will be displayed to the right.
- *Num pixels* Y – The number of pixels on the y axis. The original number will be displayed to the right.
- *Add to project after saving* – Toggling this option on will load the resampled file into GMS/SMS upon completion.

After completing the *Resample and Export Raster* dialog, and clicking **OK** , the *Save As* dialog will appear to save a file with the resampled data.



Smoothed Raster

This command will bring up the *Raster Smoothing Options* dialog. The dialog is used to smooth the values on the DEM based on use specified weighting functions. The dialog has the following options:

- *Filter Size* – To smooth the raster, an N x N filter matrix is placed over each elevation point and a new elevation is computed by taking an inverse-distance weighted average of all elevations within the filter. The dimension of N can be specified as either *3x3* or *5x5* , meaning that new elevations are computed from either the nearest 8 or 24 neighboring points.
- *Number of iterations* – Specify the number of smoothing iterations. By default only one iteration is done, but sometimes several smoothing iterations are required to propagate a change in elevations across a large flat area.
- *Maximum change in elevation* – Can be used to ensure that the integrity of the original elevations are maintained.
- *Filter ratio* – Should be between 0-1, and is used to specify the weight of the central cell of the filtering matrix.

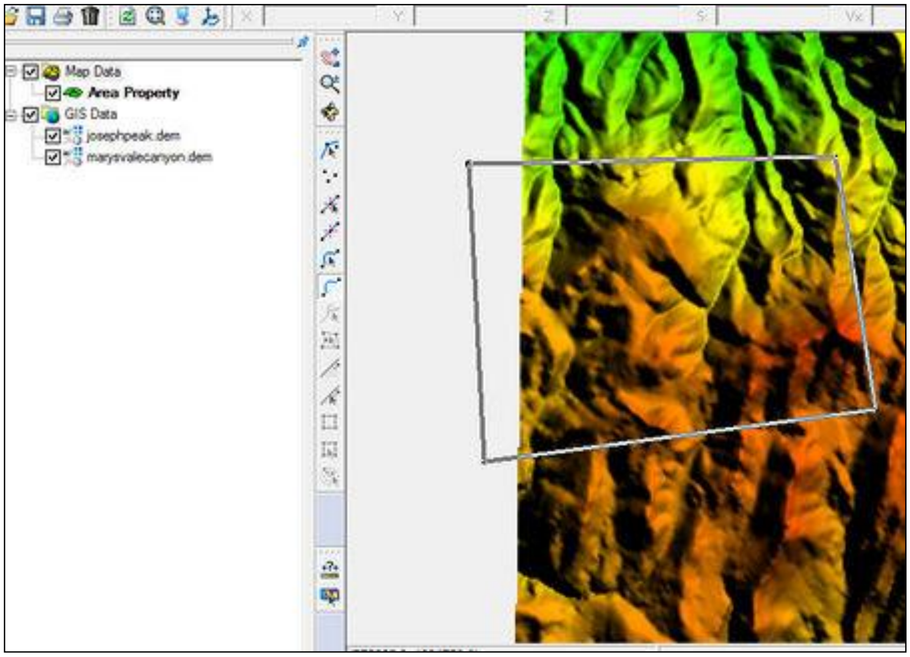
After completing th *Raster Smoothing Options* dialog, and clicking **OK** , the *Save As* dialog will appear to save a file with the smoothed data.

Trimmed Raster

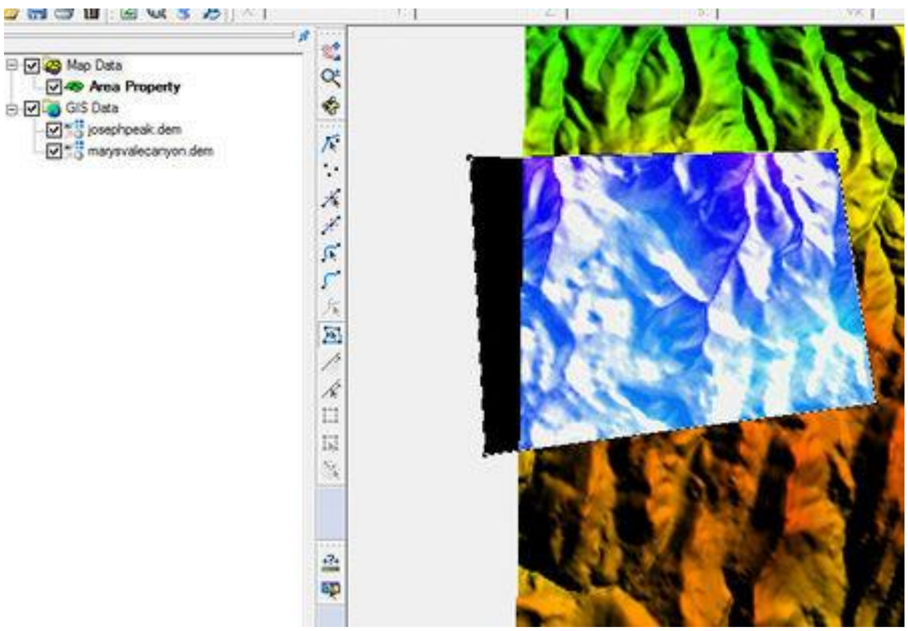
This command creates a new DEM from the selected DEM trimmed to the bounds of the active coverage or the selected polygons of the active coverage. Coverage polygons must be selected to define the trimming rectangle. If this is not done, then a warning dialog will appear as a reminder.

To create a trimmed raster:

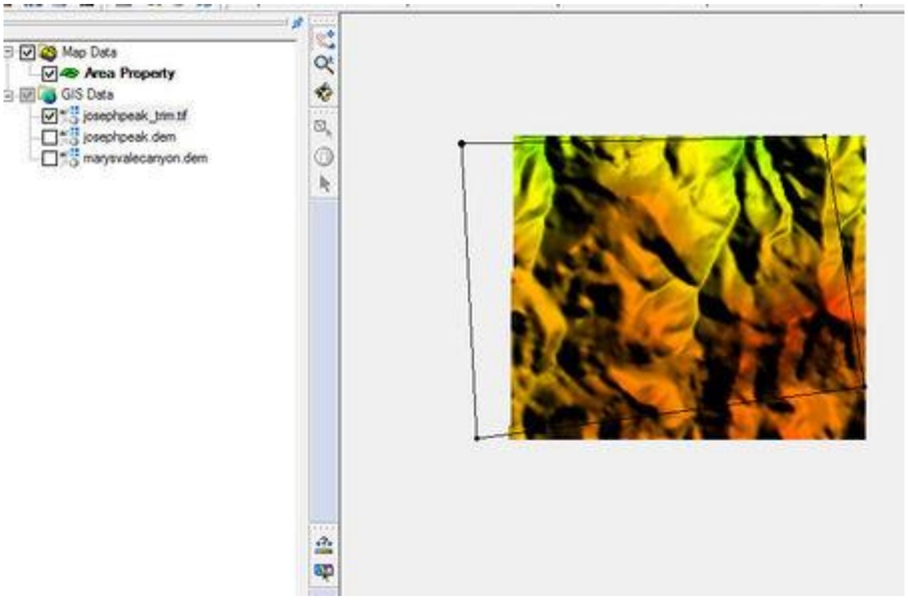
1) Select a coverage (or create a new coverage) in the Map module.
2) Select the **Create Feature Arc** tool and draw a closed set of arcs.

1. 

3) While the arcs are selected, go to *Feature Objects* | **Build Polygons** .
4) Use the **Select Polygon** tool to select the polygon just created.

2. 

5) Right-click a DEM in the GIS data and select *Convert To* | **Trimmed Raster** .
6) The *Save As* dialog will appear to save a file with the trimmed data.

3. 

The final raster is trimmed along the rectangle enclosing the polygon. Trimmed rasters will also trim to the rectangle enclosing multiple selected polygons.

Merged Raster

This command appears when multiple rasters are selected. It creates a new raster that covers all selected rasters with a resolution set to the smallest cell size of any of the selected rasters. Two or more DEM datasets can be merged together by doing the following:

1) Select two or more DEM items in the project explorer by holding down the *Shift* key while selecting each item.
2) Right-click on one of the selected DEMs and select *Convert To* | **Merged Raster** . This command is only available when multiple raster items have been selected.
3) The *Save As* dialog will appear to save a file with the merged data.

The new merged DEM will be visible in the project explorer. When merging, the smallest cell size among the merging rasters is used.

Edit Raster

A DEM may have bad cell values or based on the resolution may not represent continuous values along a line. These value can be edited to remove the bad value or ensure continuous representation of a feature such as an embankment or channel. The new values are specified using feature objects (arcs or nodes) in the Map module. To create an edited raster:

1) Select (or create) one or more feature arcs/feature nodes with the desired raster values specified on the feature object. (i.e. the z-value specified for the nodes/vertices/points represent the values desired under the feature objects in the raster)
2) Right-click on one of the selected DEMs and select *Editing* | *Arc Elevation Profile* .
3) The *Save As* dialog will appear to save a file with the edited data.

The new edited DEM will be visible in the project explorer.

Saving Raster Data

Raster data being converted can be saved in any of the following formats:

- GeoTiff Files (*.tif)
- BIL Files (*.bil)
- Arc Info ASCII Grid Files (*.asc)
- DXF 3D Point Files (*.dxf)

- Erdas Imagine IMG Files (*.img)
- Surfer ASCII Grid Files (*.grd)
- Surfer Binary grid v6 Files (*.grd)
- Surfer Binary Grid v7 Files (*.grd)
- USGS ASCII DEM Files (*.dem)
- XYZ ASCII Grid Files (*.xyz)
- Float/Grid Files (*.flt)
- DTED Files (*.dtf)
- Vertical Mapper (MapInfo) Grid Files (*.grd)
- Global Mapper Grid Files (*.gmp)
- Windsim GWS Files (*.gws)

Related Topics

- Raster Options
- Get Online Maps
- Smoothing DEMs
- GMS GIS Module
- SMS GIS Module

GIS Module Display Options

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

- *Text font*

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

- *Rasters*
  - *Display as 2D image* – makes it so the raster is only visible in plan view (like other images). This option is fast and memory efficient and can support large rasters (or several rasters).
    - *Enable hill shading* – toggle on to show shadows and thus makes the image appear 3D.
  - *Display as 3D image* – makes it so the raster is visible in any view, not just plan view. 3D points are also very memory efficient but may be a little slower than the 2D image option. Also, hill shading is not available in this mode.
    - *Point size* – specifies the size of points displayed.
    - *Maximum points* – sets a limit to the number of points to generate. Specifying a maximum number of points can be useful if the size of the raster is such that rendering becomes slow.
  - *Shader* – with either 2D or 3D, four different shaders, which control the color ramp, are available. The shaders available are:
    - Atlas Shader
    - Color Ramp Shader
    - Global Shader
    - HSV Shader

Related Topics

- GIS Module
- Display Options

ArcObjects

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

ArcObjects® are enabled by using the *GIS* | **Enable ArcObjects®** command. A few of the typical processes used in XMS must be done differently when ArcObjects® are enabled, such as importing shapefiles using the **Add Data** command instead of using the **Open** or **Add Shapefile Data** commands.

Error Enabling ArcObjects

If the DLL "EMRL_LicCheckMod.dll" fails to register automatically, selecting the **Enable ArcObjects** command will bring up the error:

> "ArcObjects could not be enabled: please check installation and licensing for ArcGIS"

To fix this, register the DLL manually by following the steps below.

1) Bring up a command prompt window.
2) Type in 'regsvr32 "<directory where XMS was installed>\EMRL_LicCheckMod.dll"

   For example, the default location for GMS 7.0 is "C:\Program Files\GMS 7.0". If the program was installed in the default location, for example, this line in the command prompt window will be: regsvr32 "C:\Program Files\GMS 7.0\EMRL_LicCheckMod.dll"

3) Hit enter to run.
4) Restart XMS

Related Topics

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

GIS to Feature Objects Wizard

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

Withing the GIS Module active selecting the *Mapping* | **ArcObjects → Feature Objects** command (with an ArcObjects license in a 32-bit version of SMS), or the *Mapping* | **Shapes → Feature Objects** (without an ArcObjects license) launches a mapping wizard which guides through the process of converting the GIS data layer features to feature objects in a map coverage. Before beginning the mapping process, first go to the map module and make sure that the currently active coverage is the coverage to map GIS data layer features to. Also, SMS will associate the GIS attributes with coverage attributes, so make sure the coverage attributes are defined before doing the conversion.

After making sure SMS will be mapping to the correct coverage, select the GIS features which overlays the study area and where wanting to map (this is done with the selection tool in the GIS module). If wanting to map all the features, choose the *Edit* | **Select All** command, or just execute the **Mapping** command, and a prompt will ask if wanting to convert all features since none are selected.

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

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

Generic Model arc or node mapping

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

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

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

Related Topics

- GIS Module

3.5. Map Module

Map Module

Map Module

Map

Feature Objects

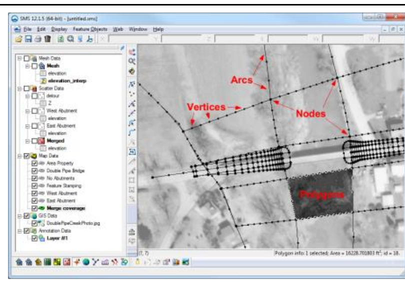
Coverages

More

Map Display Options

Map Module Tools

Map Module Menus



At a glance

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

- Conceptual model makes it easier to create, edit, and alter models

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

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

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

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

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

The Map module is included with all paid editions of SMS.

Conceptual Model

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

- Bathymetry source
- Materials information
- Boundary conditions

Creating Feature Objects

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

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

Elevations of Feature Objects

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

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

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

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

Functionalities

See the article Map Module Display Options .

Project Explorer

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

Map Module Root Folder Right-Click Menus

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

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

Coverage Item Right-Click Menus

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

- **Type** – Change the coverage type.

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

Menus

The following types of menus are available in the Map module:

- Standard Menus – see SMS Menus for more information.
- Module Specific Menus – see Feature Objects Menu for more information.

Tools

The Map module has a number of tools that are specific for creating and manipulating feature objects. See the article Map Module Tools for more information.

Related Coverages

The attributes of entities on a coverages belong to a list of attributes associated with a type of coverage. For more information see the article Coverages .

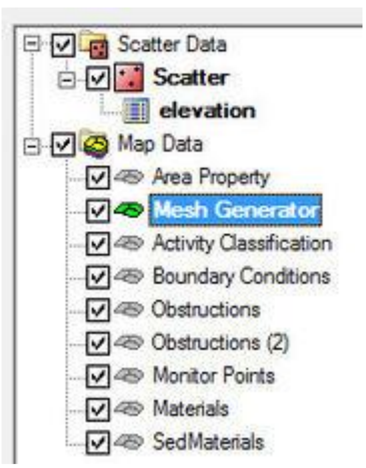
3.5.a. Coverage Types

Coverages

Feature objects in the Map module are grouped into coverages. Each coverage has a specific type, which determines the attributes that can be associated with geometric objects in that coverage. The coverages are grouped into conceptual models. The type of the active coverage is displayed in the information box displayed from the **Get Info** command.

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

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



Creating a New Coverage

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

Coverage Right-click Menu

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

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

Merging Existing Coverages

Occasionally, its desirable to hve independent features of two separate coverages combined into one converage. SMS allows merging these two coverages together. Select one of the coverages listed in the data tree then multi-select the other coverages to be merged. This can be done one at a time by holding the *CTRL* key, or several adjacent coverages can be selected by holding the *SHIFT* key and then clicking the last adjacent dataset.

Once all datasets to be merged have been selected, access the right-click menu and chooses **Merge Coverages** . A dialog may appear asking if wanting to delete the coverages used to create the merged coverage.

Coverage Types

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

Coverages are grouped into two categories:

- Generic – coverages for generic input prepossessing, generic output postprocessing and generic model interfaces.
- Models – coverages for specific models in SMS.

Related Topics

- Map Module

3.5.a.1. Generic Coverages

Generic Coverages

Generic coverages can be used for generic input prepossessing, generic output postprocessing and generic model interfaces. Generic coverages can be selected by right-clicking on the map module data and selecting *Type* | *Generic* ; or by right-clicking on the Map Data item and selecting **New Coverage** command followed by using the *New Coverage* dialog.. These coverages can be used with most models and simulations.

Generic Coverages

Currently, the following coverages are considered generic coverages:

- 1D Hyd Centerline Coverage – Used to identify the centerline in the hydraulic model.
- 1D Hyd Cross Section Coverage – Used to identify the cross section stations.
- Activity Classification – Used to define active and inactive areas of a mapped dataset.
- Area Property – Maps properties such as Manning's roughness values to the mesh, grid, or cross-sections.
- CGrid Generator – Used for building feature objects for conversion to a cartesian grid.
- Location – Allows creating points that can be used to gather measurements at specific locations during the model run similar to a model monitor points coverage. Generally designed to be used by developers using the dynamic model interface .
- Mapping – Used for building feature objects for conversion to a scatter set.
- Mesh Generator – Used for building feature objects for conversion to a mesh.
- Observation – Used for model verification and calibration processes.
- Particle/Drogue – For visualization post-processing.
- Plot Data
- Quadtree Generator – Used for building feature objects for conversion to a quadtree.
- Spatial Data – Designed to store, visualize, and analyze various types of data at node locations.
- Spectral – Used to store all spectral data by location and time.
- Stamping – Used for insertion of man-made structures into a natural topography or bathymetry set.

Related Topics

- Map Module
- Model Specific Coverages

Activity Classification Coverage

Some projects require defining which areas of the model are active or inactive. Active areas represent an elevation below the water-surface or the ocean. Inactive areas have elevations above the water-surface or are land areas. Some models automatically assign areas as active or inactive. The Activity Classification coverage in the Map module can be used for models that don't automatically classify activity based on elevation, or used to speed up the model run time in models that do automatically classify activity. The coverage can use all of the standard Map module interface components .

In the coverage, polygons can be created by creating enclosed arcs and using the *Feature Objects* | **Build Poygons** command. Each polygon can then be classified as active or inactive.

Once the active and inactive areas have been defined, the Activity Classification coverage can be linked to a simulation for use in a model run. SMS allows a project to use multiple Activity Classification coverages.

Actvity Classification Coverage Dialog

This dialog is reached by right-clicking on a polygon with the **Select Feature Poylgon** tool and selecting the **Attributes** command. The dialog that appears has the following options:

- *Active* – Classifies the polygon as below the surface-water elevation or as ocean.
- *Inactive* – Classifies the polygon as above the surface-water elevation or as land.

Related Topics

- Generic Coverages

Area Property Coverage

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

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

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

Many of the data entities constructed and edited in SMS (i.e., elements, cells) have a material ID associated with them. This material ID is an index into a list of material types. Materials contain model specific parameters such as manning's roughness, or bed material grain size. A global list of material attributes is maintained and can be edited using the menu command *Edit* | **Materials Data** . 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.

Area Property Polygon Attributes Dialog

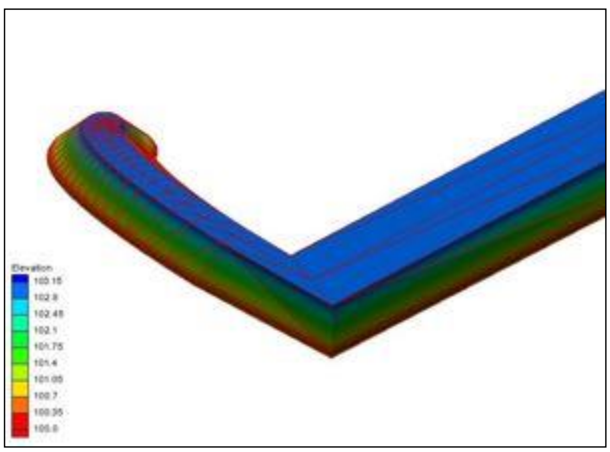
The *Land Polygon Attributes* dialog is used to set the attributes for feature polygons . It is reached by double-clicking on a polygon in an Area Properties coverage. Attributes that can be specified for each polygon include:

- Polygon Type
    - *None* – No attributes are assigned to the polygon.
    - *Material* – With this option selected, a material can be assigned to the polygon from the drop-down list below. Only materials defined in the *Materials Data* dialog before they can be assigned to a polygon.

Related Topics

- Coverages
- Materials Data

Feature Stamping



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

The Process

The basic steps to define a linear stamped feature include:

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

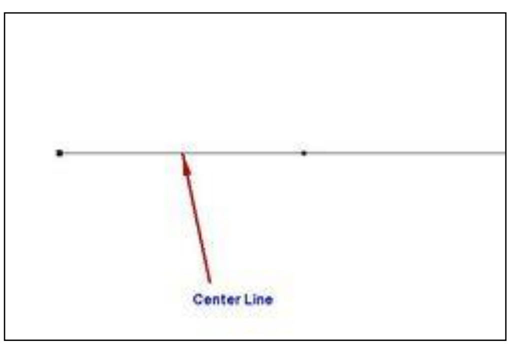
2) A scatter set defining the elevation for the feature.

Sample problems in the section below illustrate the procedure.

Define the Coverage and Centerline

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

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

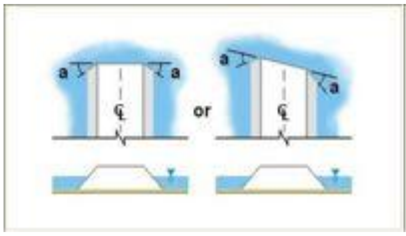


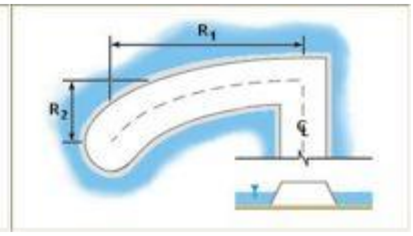
Define Feature Attributes

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

- *Feature Name* – This will be used when SMS creates stamped feature coverages and scatter sets from the stamping coverage.

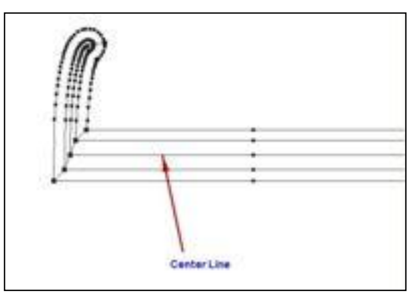
- *Stamping Type* (cut = channel or fill = embankment) – If there is a stamped feature that contains both cut and fill sections, create one coverage as a cut, then duplicate the coverage and change the copies type to fill.
- A base elevation (the top elevation of the embankment or the bottom of the channel)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





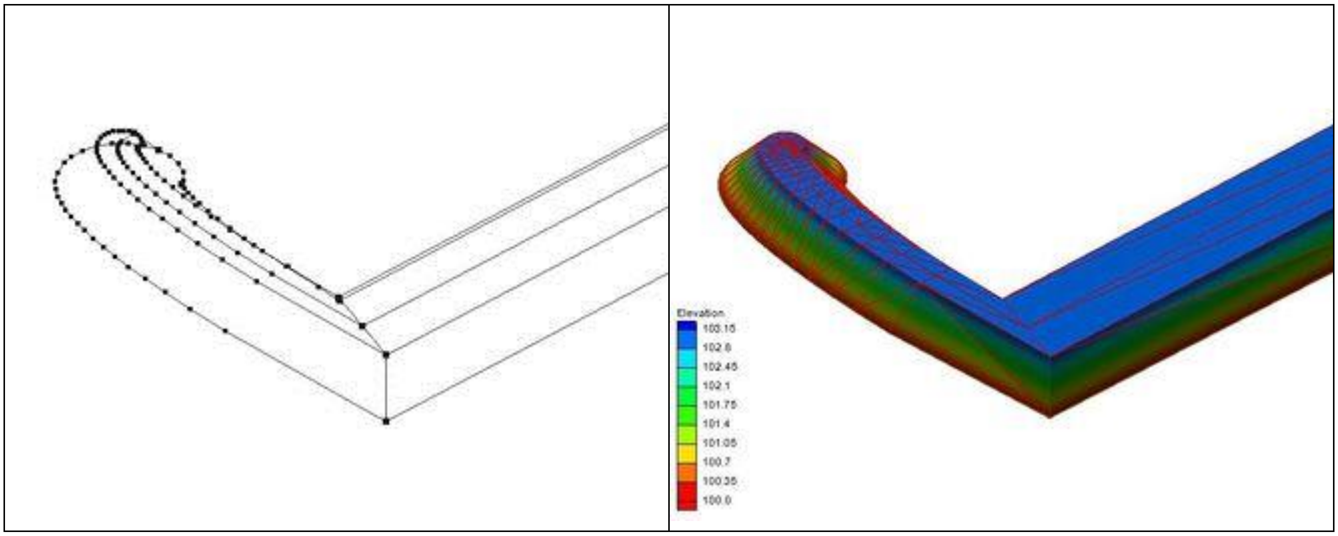
1)		

Stamping the Feature



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

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



Case Studies / Sample Problems

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

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

Related Topics

- Generic Coverages

External Links

- Emery, R. N. (2005). Refining and Expanding the Feature Stamping Process. Thesis, Brigham Young University. [69]
- Christensen, J. R. (2001). Stamped Features: Automatic Generation of Flow Modifying Structures in Conceptual Models. Thesis, Brigham Young University.
- Zundel, Alan K. and J. Ryan Christensen, “Stamped Features: Creation of Engineered Structures in Conceptual Models”, International Journal of Hyrdroinformatics, Vol. 4, No. 1, February 2002, pp. 63-72.

Mapping Coverage

The Mapping coverage is a generic coverage used for building feature objects for conversion to a scatter set.

General Mapping Options

When creating a new Mapping coverage from the *New Coverages* dialog, the *General Mapping Options* dialog will appear.

Mapping Arc Attributes Dialog

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

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

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

Mapping Polygon Attributes Dialog

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

- Extrapolation Type

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

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

Related Topics

- Feature Objects Menu
- Steering

- Generic Coverages

Observations

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

Creating an Observation Coverage

To create a new observation coverage:

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

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

Creating an Observation Point

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

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

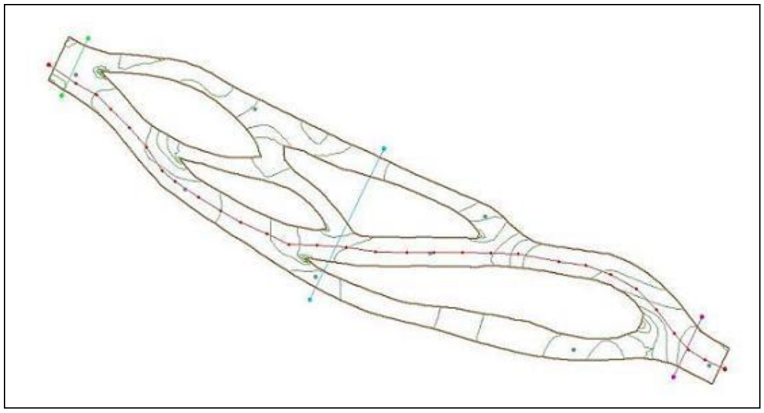
Creating an Observation Arc

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

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

Setting Observation Object Attributes

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



Viewing Results

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

Calibration

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

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

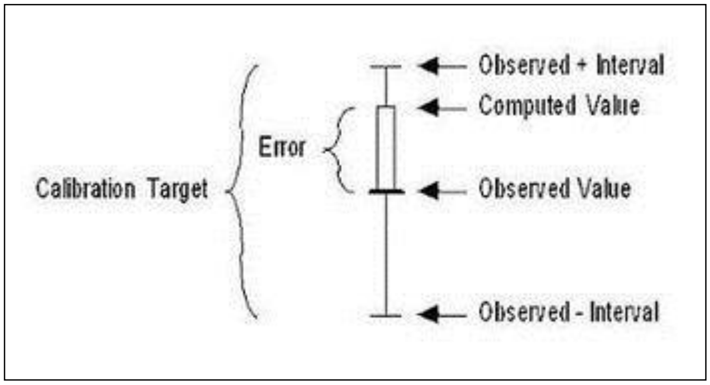
Once a set of observed point and flow values has been entered, each time a model solution is imported, SMS automatically interpolates the computed solution to the observation points. A calibration target representing the magnitude of the residual error is displayed next to each observation point and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

Calibration Target

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

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

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



Calibration Display Options

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

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

Observation Coverage Dialog

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

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

Dialog Layout

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

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

Observation Point Attributes

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

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

- **Angle** – When a measurement for observation points is tied to a vector dataset (as is the case with a Velocity measurement) an angle needs to be specified. This angle is an azimuth angle (measured clockwise from north) with the top of the screen representing north when in plan view.

Observation Arc Attributes

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

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

Each observation arc is assigned the following attributes:

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

Related Topics

- Coverages
- Plot Window

Particle/Drogue

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

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

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

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

Application

- Residence time calculations. One of the most useful applications of drogue plots is to approximate residence time of a basin or other subregion of a hydrodynamic domain. In order to approximate this:
  - Distribute a fairly large number of drogue seed points inside the basin (or sub domain) of interest).
  - Generate a drogue plot film loop of the time range of interest. (Currently the hydrodynamic solution must include enough time steps to span the residence time in question. If this is not the case, additional time steps should be added to the data set either by rerunning the simulation for a longer time period or duplicating time steps).

  - Review the number of particles still in the basin (or subdomain) at various time intervals. The percentage of particles, and their position give an indication of overall concentration and local concentration in the basin.
- General visualization of flow field.

External Links:

- "Gas Flow Visualization for Combustion Analysis", Energy and Fuels, Vol. 7, No. 6, 1993, pp. 891-896. Zundel, A.K., Saito, T., Owen, S.J., Sederberg, T.W., Christiansen, H.N. [70]

Related Topics

- Coverages
- Animations

Spatial Data

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

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

- **Time Series** – The *time series editor* allows viewing, editing, and importing/exporting transient datasets for the location. Each time series consists of either a scalar value at each time, or a vector value at each time. The vector quantities may be defined as (X,Y)components or ((Magnitude,Direction) pairs. The time values may be displayed as dates or offset values. The time series are stored in a database inside SMS. When the project is saved, the series are saved in the XMDF project file. The series may also be imported/exported using a TSD file format. .
  - *Edit Data* – This menu option invokes the *time series editor* . The list of time series curves available for this node are listed in a list box. Select the desired time series curve and the values for that curve appear in the spread sheet. The selected time series data may be modified in this spread sheet.
  - *View Data* – This menu option operates like the *Edit Data* option, but the spread sheet is set to read only. This prevents accidental modification of the values in the time series.
  - *Delete* – This option removes the association between the selected time series and the spatial data node. The time series curve remains accessible in the Time Series database.
- **Compass Plot**
  - *Show Connection Lines* – This option allows the lines connecting the compass plot to its associated spatial data node to be enabled or disabled.
  - *Properties...* – This option brings up the *Compass Plot Properties* dialog for the selected compass plot.
  - *Delete* – This option deletes the selected compass plot.

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

Compass Plot

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

Direction Convention

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

Layout

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

Creating a Compass Plot

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

1) Create or Select a *Spatial Data* Coverage.
2) Create or Select a *Feature Point* in the *Spatial Data Coverage* .
3) Make sure at least one vector time series curve is stored for the selected feature point.
4) Right-click on the point and select the **Add → Compass Plot** command. This creates a compass plot and brings ups the *Compass Plot Properties* dialog. Each time series to be included in the compass plot must be selected in the *Spatial Data* section of the dialog. Clicking **OK** will cause the dialog to disappear and the compass plot to appear. Properties of the compass plot include:
   - The name of the plot which can optionally be displayed at the top of the compass plot.
   - A flag to show/not show a vector for each vector time series stored at the *Spatial Data* point.
   - Options for a compass plot legend, including:
     - A flag to show/not show the legend.
     - Set the position of the legend. Options include any side of the plot.
     - The number of vectors to show in the legend. This can be the min/max, or one for each compass ring.
     - The number of digits of precision for the legend.
   - Control of the number of rings to show in the plot, and the percent of maximum value for each ring. By default SMS creates the compass plot with four concentric circles, representing 1/4, 1/2, 3/4 and the maximum vector magnitude.
   - Display options including:
     - The pixel size of the compass plot.
     - A flag to show only the vector direction (ignore magnitude).
     - A flag to show connection lines. Since the plot can be selected and drug to any position on the screen, these lines can be useful to show a location the vectors apply to.
     - A flag to set the background color (if desired) for the compass plot. By default, the plot is filled with a gray background.
     - An option to specify the magnitude range to display. Any vector with magnitude above the maximum will appear as a 100% magnitude vector. Below the minimum, the vector will not be displayed.
       - The vector style. (This is a future enhancement. Currently, only "Normal" is supported.

Editing a Compass Plot

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

1) Select the **Select Compass Plot** tool.
2) Left-click on the selection box of a compass plot and drag to position and resize the plot.

3) Right-click on the plot itself. I menu appears including the following options:
   - **Delete** – Allows the compass plot to be deleted
   - **Show Connection Lines** – Sets the display option of the plot
   - **Show Legend** – Turns on/off the legend for the compass plot
   - **Legend Location** – A pull right menu that positions the legend
   - **Properties** – Invokes the *Compass Plot Properties* dialog to edit any of the attributes.

Related Topics

- Time Series
- Coverages

Spectral Coverage

Spectral coverages are 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.

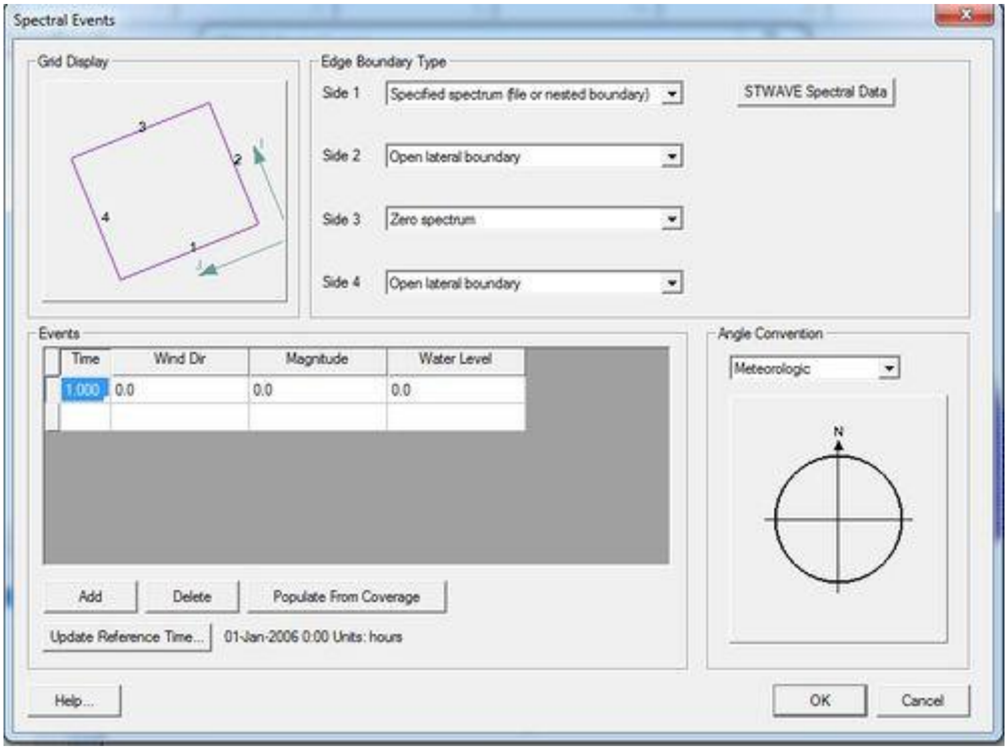
Spatial Varied Boundary Conditions

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

Creating Spectral Data

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

Creating a Spectral Event



Related Topics

- [Spectral Energy]

3.5.a.2. Model Specific Coverages

Model Specific Coverages

Specific model coverage can be selected by right-clicking on the map module data in the Project Explorer and selecting *Type* | *Models* followed by selecting the model coverage; or by right-clicking on the Map Data  item and selecting **New Coverage** command followed by using the *New Coverage* dialog.

The following model specific coverages are available:

- ADCIRC – Used to build a conceptual model of an ADCIRC project.
- ADH – Has two coverage options.
  - ADH – Used to build a conceptual model of an ADH project.
  - Vessel – Used to add vessels to the simulation and give them paths to follow.
- BOUSS2D – Used to build feature objects and parameters for a BOUSS-2D model simulation.
  - Damping – Used to create arcs with damping attributes.
  - Porosity – Used to create arcs with porosity attributes.
  - Roughness – Uses polygons to define the varying roughness.
  - Wavemaker – Used to create arcs that define wave parameters.

- BOUSS Runup/Overtopping – Uses multiple coverages to create the Runup/Overtopping model simulation. The coverages are:
    - Probes – Used to create feature objects for probes.
    - Damping – Used to create arcs with damping attributes.
    - Porosity – Used to create arcs with porosity attributes.
    - Roughness – Uses polygons to define the varying roughness.
    - Transects – Used to create arcs to represent the 1-d grid used for a run-up simulation.
    - Wavemaker – Used to create arcs that define wave parameters.
- CGWAVE – Used to build a conceptual model of a CGWAVE project.
- CMS-Flow – Utilize two different coverage types.
    - Boundary Conditions – Used to create the computational domain (or grid).
    - Save Points – Used to define special output locations from the computation.
- CMS-Wave – Used to build a conceptual model of a CMS-Wave project.
- CSHORE – Used to build a conceptual model of a CSHORE project.
- EFDC – Used to create curvilinear grids for use with the EFDC model (EFDC grid format).
- ESMF – Used to couple ADCIRC and STWAVE models.
- FESWMS – Used to build a conceptual model of a FESWMS project.
- GenCade – Used to build a conceptual model of a GenCade project.
- Generic Model Coverage
- PTM – Allows simulating particle transport processes.
- SED-ZLJ – Sediment model used with the EFDC coverage.
- SRH-2D – Uses multiple coverages to create the SRH-2D model simulation. The coverages are:
    - Boundary Conditions – Used to create boundary conditions for hydraulic computation.
    - Obstructions – Used to create feature objects that represent obstructions, such as bank protrusions and boulder clusters.
    - Monitor Points – Used to gather specific information for that location at all time steps.
    - Materials – Allows creating material zones specific to the SRH-2D model.
- STWAVE – Used to build a conceptual model of a STWAVE project.
- TABS (RMA2/RMA4) – Used to build a conceptual model for a RMA2 or RMA4 project.
- TUFLOW Coverages – Used to create feature objects for a TUFLOW simulation. Includes several coverages.
    - 1D Cross Sections – Used to define open channel cross section data for 1D networks.
    - 1D Networks – Used to create channels and nodes for a 1D domain.
    - 1D Water Level Lines – Define the locations where 1D solutions will be written as 2D output.
    - 1D Water Level Points – Used in conjunction with water level lines to guide TUFLOW on creating 2D output for 1D networks.
    - 1D/2D BCs and Links – Used to specify cell code (active/inactive) areas of the 2D model domain.
    - 1D/2D Connections – Used with the 2D BC coverage to link 2D and 1D domains.
    - 2D Flow Constriction Shapes – Used to define flow constrictions in TUFLOW.
    - 2D Flow Constriction (cell based)
    - 2D Grid Extents – Used to create TUFLOW grids.
    - 2D Materials – Used to assign material data for use in the TUFLOW simulation.

    - 2D Miscellaneous (FLC, WRF, IWL, SRF, and AD)
    - 2D Z Lines (advanced) – Modifies geometry through time to simulate levee failures or other changes to elevation data within the model run.
    - 2D Z Lines/Polygons (simple) – Used as geometry modifications and force grid elevation values using arcs or polygons.
    - 2D/2D Linkages – Used to setup TUFLOW to use multiple 2D domains.
- WAM – Used to build a conceptual model of a WAM project.
- Wind – Represents a storm as an arc or series of arcs that follow the storm track and define the storm attributes at the nodes along the arc.
    - Holland/PBL – Used to create a storm track for a PBL project for use in ADCIRC.
    - Synthetic Storm – Provides a mechanism for creating a PBL coverage based upon user specified parameters.

Related Topics

- Map Module
- Generic Coverages

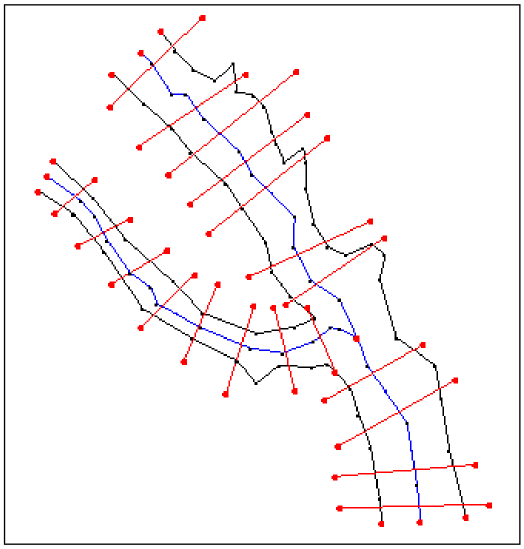
1D Hyd Centerline Coverage

The 1D-Hydraulic Centerline coverage has three possible attribute types: general, centerline, and bank. If the arc is a general arc type then it does not participate in the building of a hydraulic centerline and only provides additional visual detail to the model. A bank arc is used to mark left and right bank points for any cross sections that are automatically extracted from at digital terrain model.

A centerline arc provides the backbone of the hydraulic model definition. It's direction should be from upstream to downstream as this is the way HEC-RAS commonly views the river. This automatically defines which is the left bank and which is the right bank (think of standing up river and looking downstream when determining left and right). A centerline has as attributes the river reach properties as defined in the *River Reach Attributes* dialog.

The river reach properties include:

- *Arc Type* – sets if the attributes are for a centerline, left bank, or right bank arc.
- *River Name* – only editable for a centerline arc.
- *Reach Id* – internally assigned and not editable.
- *Reach Name*
- *Computational Length* – enerally equal to the length but this could be different in order to account for additional sinuosity.
    - **Reset** – will clear whatever has been entered for the *Computational Length* .
- *Feature Length*
- *Start Station*
- *End Station*

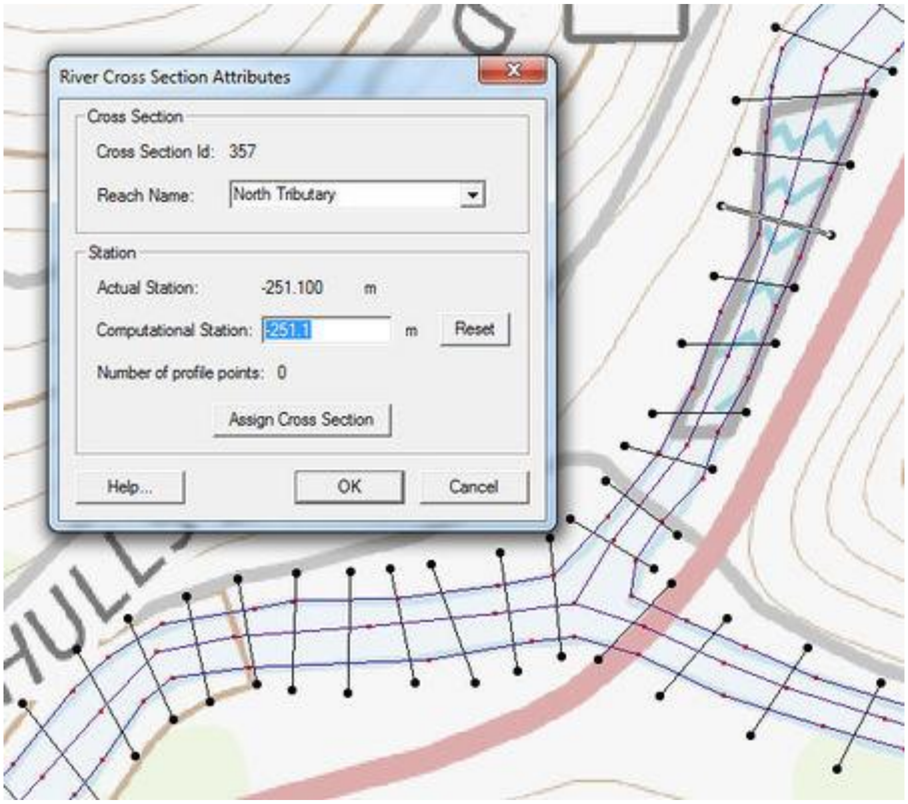


Related Topics

- 1D Hyd Cross Section Coverage
- WMS 1D-HYD Centerline Coverage Type

1D Hyd Cross Section Coverage

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



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

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

Coverage Specific Right-Click Commands

The 1D Hyd Cross Section Coverage contains all the standard commands in its right-click menu. The coverage also contains a couple specific commands unique to its right-click menu. These commands include:

Add Arcs to Mesh

Adds all cross section arcs to an existing mesh.

Extract cross sections

Brings up the *Extract Cross Sections* dialog where a cross section database can be created from the cross sections in the coverage.

Summary table

Brings up the *Summary Table Options* dialog which allows viewing calculations along cross section arcs.

Extracting Cross Sections

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

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

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

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

Cross Section Database

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

Summary Table

The summary table is a tool that allows viewing calculations along cross section arcs in a project. To use the summary table, there must exist a cross section coverage , centerline coverage , and geometry (i.e. grid, mesh, scatter). To access the summary table, right-click on the 1D-Hydraulic Cross Section coverage and select **Summary Table...** .

Summary Table Options

The *Summary Table Options* dialog is where which desired calculations are specified, as well as which geometry to use, and which portions of the cross section should be used.

- *Data Source* – The *Data source* **Select** button brings up a *Select Tree Item* dialog. This dialog is used to select the geometry that has the datasets for the calculations.
- *Cross section options* – There are three options for the cross sections: "Full cross section", "Main channel only", and "Overbanks and main channel".
    - "Full cross section" – This option is used when there is no cross-section database, overbanks have not been specified, or if the calculations should be done over the entire cross-section.
    - "Main channel only" – This option can only be used if there is a cross-section database. The calculations will be performed only on the main channel portion of the cross section , defined as the space between the right and left overbanks.
    - "Overbanks and main channel" – This option can only be used if there is a cross-section database. The calculations will be performed on three separate portions of the cross section: the left overbank, main channel, and right overbank.

- *Datasets* – After selecting a data source, the spreadsheet will be populated with all of the scalar datasets that belong to the data source. For each dataset, toggle on minimum, average, and maximum to be calculated, as well as select the dataset time step to be used. To calculated the minimum, maximum, and average values for each cross section, SMS will interpolate the values from the dataset to the points along the cross section. These interpolated values will then be used to determine the minimum, maximum, and average value.
- *Defaults – The Select defaults button will search for common dataset keywords in the list and automatically turn on the average calculation. The keywords are: "water_elev", "wse", "vel_mag", "vmag", and "froude".*
- *Advanced Calculations* – Some other helpful calculations are available in the *Advanced Calculations* section.
    - *Flow* – To calculate the flow over the cross sections, it is necessary to specify a velocity dataset (vector) and a depth dataset (scalar). SMS will then calculate the flow over the cross section at the specified time step.
    - *Width* – To calculate the width of the cross sections, it is necessary to specify an elevation dataset and a water surface elevation dataset. SMS then compares the water surface elevation with the cross section elevation to determine the width.

Summary Table

Once all of the options are set, click on **Generate table...** to have SMS compute the desired values and display them in a table.

Related Topics

- 1D Hyd Centerline Coverage

ADCIRC

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

ADCIRC Conceptual Model Development

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

Define Coastline

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

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

Define Ocean Boundary

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

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

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

Build Polygons

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

Choose Mesh Generation Method

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

LTEA Toolbox

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

Manual Mesh Generation

To manually generate a mesh:

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

Related Topics

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

ADCIRC Wind Coverage

The ADCIRC wind coverage represents a storm, such as a tropical depression or hurricane, as an arc or series of arcs that follow the storm track and define the storm attributes at the nodes along the arc. This information is often available in a "ATCF best track" similar file. It is called a "similar" file, because the ADCIRC development group have modified the format slightly for each type of application.

Data in the ATCF best track format for historical storms are available from various locations including tropicalatlatic.com/modelsOLD .

SMS can import data in this format or extract data from the HURDAT database and convert it to this format to associate it with an ADCIRC simulation. The data can be utilized for three of the Node Wind Stress (NWS) types supported by ADCIRC including: NWS = 8 (symmetric vortex model), NWS = 12 (OWI PBL wind format), and NWS = 19 (asymmetric vortex model). Depending on the selected option for NWS, SMS will export the data as a file named fort.22 for the ADCIRC simulation in formats compatible with either NWS = 8 or NWS = 19, or as a ".trop" file for use by PBL.

Various model checks have been implemented to help catch mistakes and avoid crossing ADCIRC's limitations.

To create a wind coverage, simply open one of these file into SMS. Alternatively, create a new coverage and select the coverage type *Models* | **Wind** or convert an existing coverage by right-clicking it and selecting *Type* | *Models* | **Wind** .

Storm Attributes

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

- *Wind Model* : Choose between symmetric and asymmetric definition of the storm. The wind model will determine which fields are displayed and which are hidden (by default) in the node attributes dialog.
  - *Holland Symmetrical* : The basic and default choice, and assumes a simple storm definition will suffice.
  - *Holland Asymmetrical* : Gives more options for defining the storm's shape and orientation.
  - *Planetary Boundary Layer (PBL)* : Activates an **Options** button that will bring up the *PBL Model Control* dialog. Model isn't currently available.
- *Wind Attributes*
  - *Basin* : Defines a world region where the storm is taking place.
  - *Subregion* : Defines a world region where the storm is taking place.
  - *Annual cyclone number* : Does not affect calculations, but is valuable for book keeping.

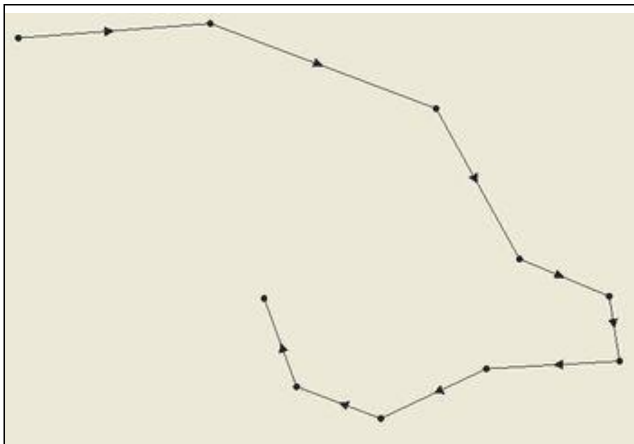
Building a Storm Path

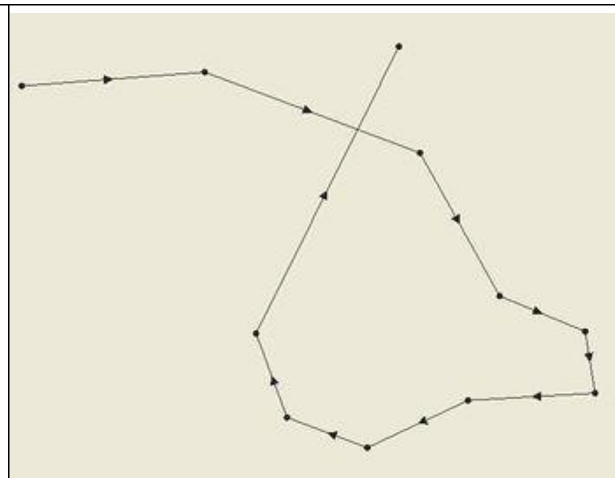
The storm path should be a single continuous line with no breaks or branches. Simply clicking out an arc is sufficient—vertices will be converted to nodes once the *Node Attributes* dialog is entered. Do:

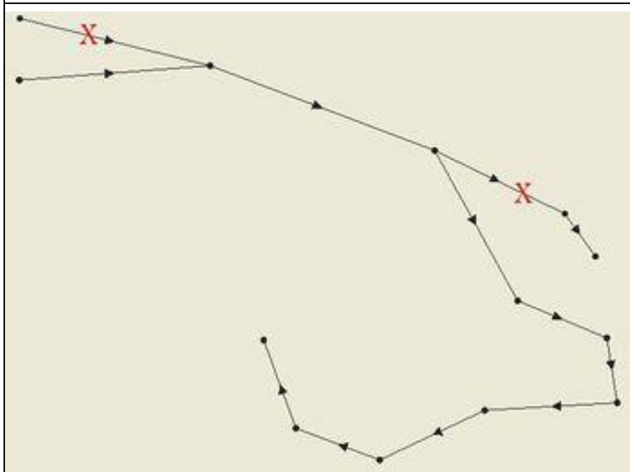
- Operate in Geographic Coordinates
- Make a single path with no breaks
- Create the number of vertices/nodes that there is data for (adjust their positions manually with the nodes/vertices selected or can set their positions within the *Node Attributes* dialog)

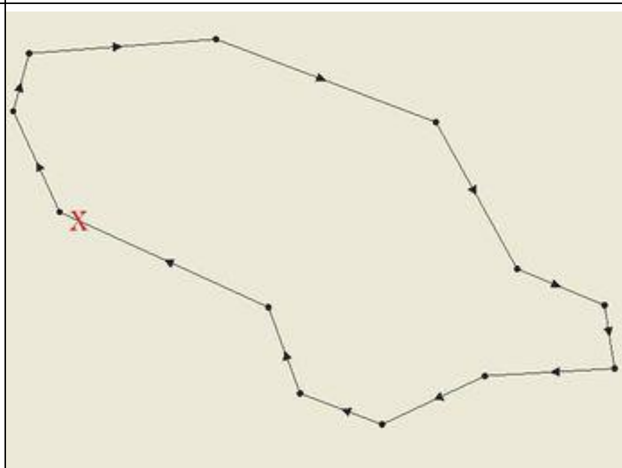
Don't:

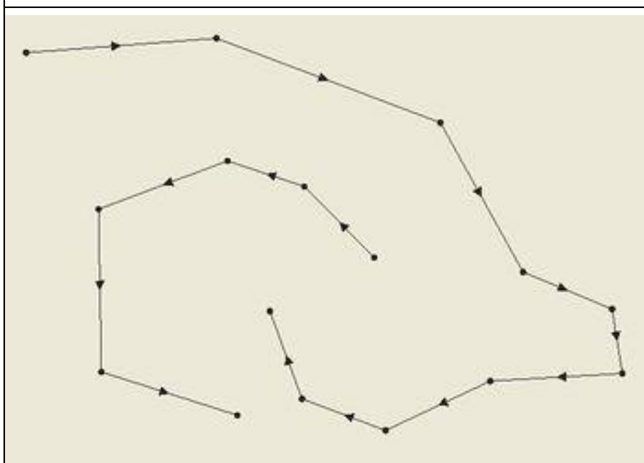
- Create multiple paths in the same coverage
- Split the path in multiple directions
- Create loops with the path

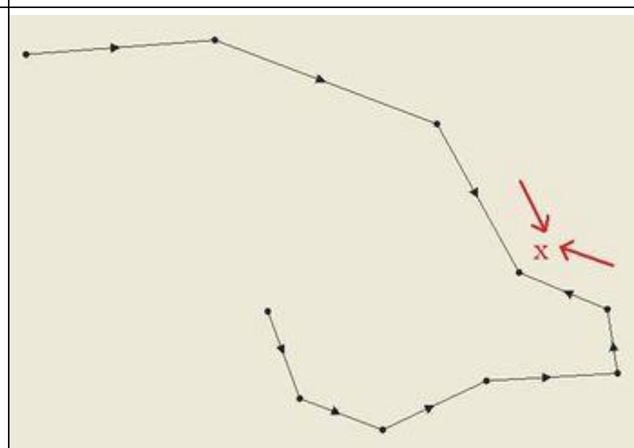












Notes

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

Node Attributes

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

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

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

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

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

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

- *Storm speed (SPEED)*
- *Storm name (STORMNAME)*
- *System depth (DEPTH)*
- *Wave height for radii (SEAS)*
- *Seas radius code (SEASCODE)*
- *Wave height radius 1-4 (SEAS1-4)*

Linking the Coverage to the ADCIRC Project

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

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

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

Related Topics

- ADCIRC

ADH Vessel

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

Boat Path

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

In general, there will always be exactly one path per vessel coverage. The path can be made up of multiple arcs, but the arcs should not split into multiple paths. The one exception to the rule of "one path per coverage" is when there are multiple vessels that are identical except for their speeds and paths. In this case SMS can have multiple separate paths in the same coverage, and each will create its own separate copy of the boat defined in the coverage properties. These paths can cross each other as long as they do not connect to each other (at a node). Another option is to simply duplicate the coverage after the boat properties are defined. This allows adding and removing the boats from the simulation separately, and the boats can have vertices and nodes in the same place (same x and y coord) along their path without conflicting. The boat path writes the FDEF and SDEF cards to the boat file .

Dialogs

Node Attributes

Nodes on arcs change the speed of the boat or, in the case of the first node of the path, define the boat's starting speed. To change the speed of the boat at a node, right-click on it then select **Node Attributes...** . This will bring up the ADH *Vessel Node Properties* dialog.

Alternately, double-click the node to reach the ADH *Vessel Node Properties* dialog.

Boat Parameters

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

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

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

Propeller (PROP card):

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

Related Topics

- ADH

CMS-Flow Coverages

The CMS-Flow model makes use of the simulation based modeling approach. This requires defining coverages in the Map module to build the components for use in the CMS-Flow simulation .

Boundary Conditions

All numeric models require boundary condition data. In CMS-Flow, boundary conditions are defined on feature arcs in a boundary conditions coverage.

The **Create Feature Arc** can be used to click out boundary conditions for the model or arcs can be converted from other coverages or modules. Arcs can also be imported.

One the boundary condition arcs have been created, right-click on the arc with the **Select Feature Arc** tool and select the **Assign Boundary Conditions** command to bring up the *Arc Boundary Condition* dialog. This command is unique to the CMS-Flow Boundary Condition coverage and is only accessible by right-clicking on a selected arc.

Arc Boundary Conditions Dialog

This dialog has the following options for boundary condition parameters.

- *Name* – Assign a name to the boundary arc.
- *Type* – Has the following options:
  - "Unassigned" – The default option.

- "Cross-shore"
- "Flow rate forcing" – Specifies an inflow rate (flow in cubic meters/second at each cell). This can be used to represent a river flowing into the domain.
  - *Flow Source*
  - *Inflow direction*
  - *Conveyance coefficient*
- "WSE forcing" – Specifies the water surface elevation as a function of time for the cellstring. Options include specifying a single curve (water level -vs- time) and all the cells will have the same water level at the specified time and extracting individual curves for each cell either from a regional tidal database (ADCIRC database) or from a regional (larger) circulation model.
  - *WSE Source*
  - *WSE offset*

Save Points

CMS-Flow includes save points which can be used to output calculations at specific locations.

Save points are created in the Save Points coverage using the **Create Feature Point** tool. When the coverage is linked to the CMS-Flow simulation data will be collected during the simulation model run.

The coverage has two unique commands. The coverage right-click menu in the Project Explorer has a **Properties** command that will bring up the *Save Points Properties* dialog. Right-clicking on a point in the graphics window and selecting the **Assign Save Points...** command bring up the *Assign Save Points* dialog.

Save Points Properties Dialog

In the *Save Points Properties* dialog the output interval can be specified for data collected at each save point. The interval options can be specified for any of the following data types:

- Hydro
- Sediment
- Salinity
- Waves

All interval options can be specified in seconds, minutes, or hours.

Assign Save Points

Each save point created in the coverage needs to be given parameters as to what type of data to collect during the simulation model run. Using the **Select Feature Point** tool, right-click on each save point and use the **Assign Save Points...** command. This will bring up the *Assign Save Points* dialog where the type of data to be gathered can be specified.

The dialog has the following options:

- *Name* – Each save point can be given a unique name. The given name will appear next to the point in the graphics window after being assigned.
- *Hydro* – Sets the save point to collect hydrologic data.
- *Sediment* – Sets the save point to collect sediment data.
- *Salinity* – Sets the save point to collect salinity data.
- *Waves* – Sets the save point to collect wave data.

Related Topics

- CMS-Flow
- CMS-Flow Simulation

ESMF – Earth System Modeling Framework

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

- ADCIRC – STWAVE

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

- Right-click in the Project Explorer .
- Choose the menu *New Simulation* | **Hurricane** .
- Change the simulation name if desired .
- Create the elements to be include in the simulation and drag tree pointers representing them into the simulation. The following items can be linked to an Hurricane simulation:
- ADCIRC – STWAVE
  - ADCIRC Mesh
  - STWAVE Grid
  - Hurricane Coverage
  - PBL Wind Coverage
  - WAM Simulation
- Right-click on the Hurricane simulation in the Project Explorer .

- Choose the menu **Properties** . This action will create a new Hurricane coverage containing polygons that identify overlapping grid sections and areas of interest defined. It will then bring up a *Hurricane Mode Project Summary* property sheet containing four tabbed dialogs.
  - Overview
  - ESMF
  - Spatial View
  - Timeline
- Right-click on the ESMF simulation and choose **Export ESMF Files** . SMS will create a folder named ESMF in the same directory as the .sms file of the current project. Inside of the ESMF folder will be a folder and input files for the ESMF Simulation. The ESMF Simulation folder name will be the same as the name given to the ESMF Simulations in the Project Explorer.

ESMF Hurricane Overview

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

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

Hurricane ESMF

- Set the ESMF simulation options in the *Data Exchange* and *Area Mapping* sections of the dialog
  - Data Exchange
    - Model A – The meshed based model to be use (hard-coded to ADCIRC).
    - Data Exchange – Controls how data is exchanged between the two models (↔ bi-directional, → uni-direction A to B, ← uni-directional B to A).
    - Model B – The grid based model to be use (hard-coded to STWAVE).
    - Model A → B – How frequently the results of Model A are passed to Model B.

        - Units – Frequency units for A to B exchange (days/hours/minutes/seconds).
        - Model A ← B – How frequently the results of Model B are passed to Model A.
        - Units – Frequency units for B to A exchange (days/hours/minutes/seconds).
- Area Mapping
    - ID – Polygon identifier correlated to the ids visible in the main graphics window.
    - Mapping
        - Single – Model A exchanges data with a single instance of Model B.
        - Combined – Model A exchanges data with multiple instances of Model B. Model A can receive either the average or maximum values from the instances of Model B.
    - Option – If the "Mapping" field is set to "Single" and multiple grids overlap the identified polygon, this field allows selecting which grid will be used. If the "Mapping" field is set to "Combined", this field select how the data from the grids will be combined (Average or Maximum).

Hurricane Spatial View

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

Hurricane Timeline

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

Related Topics

- Model Specific Coverages
- CSTORM-MS
- Steering

Generic Model Coverage

Generic model coverage is for a Generic model . The generic model interface spans both the map and the mesh modules. Create a Generic Model coverage and assign arc boundary conditions based on the types defined in the generic model template. Also attributes can be assigned to feature points in the coverage.

More information about a meshing coverage can be found under the article Coverages .

Convert Map feature arc and point attributes to mesh nodestring and node attributes

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

Convert a Map to 2D Mesh

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

Related Topics

- Generic Model

SED-ZLJ

The SED-ZLJ sediment model is a sub-model of the EFDC model .

SED-ZLJ Options

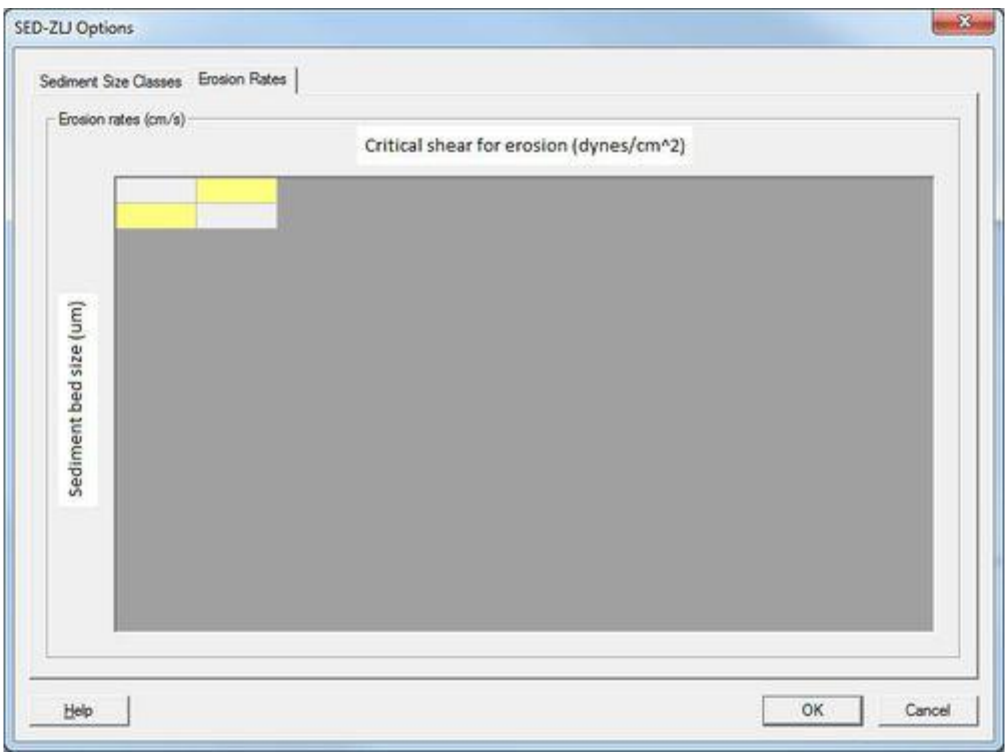
The
dialog is reached by selecting the SED-ZJL coverage in the Map Module and then double-clicking on a active node.

Sediment Size Classes

The *Sediment Size Classes* tab has the following options for entering data:

- Use measured cohesive setline velocities
- Cohesive
- Settling velocity
- D50
- Specific gravity
- Crit. shear for susp.
- Crit. shear for erosion
- Initial concentration

Erosion Rates



Related Topics

- Model Specific Coverages

Synthetic Storm Coverage

Background

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

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

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

Inputs

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

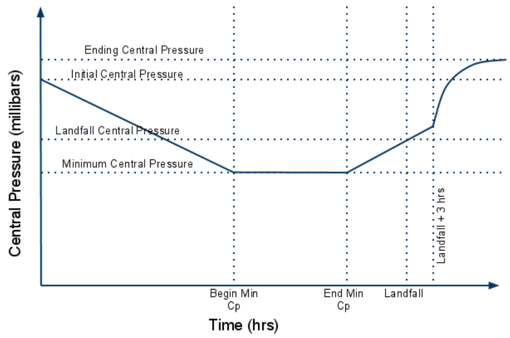
- Storm name
- Storm number
- Starting date/time (nearest hour)
- Forward speed in knots
- Far field atmospheric pressure in millibars
- Initial central pressure in millibars
- Minimum central pressure in millibars
- Landfall central pressure in millibars
- Initial storm radius in miles
- Landfall storm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

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

Methodology

ERDC has developed curves to represent general behavior of storms. These curves are a simplistic approach and should be used with caution. When in doubt if providing appropriate wind fields, consult a meteorologist.

Sample curves for central pressure, Holland B, and storm radius can be seen below. Some of the values shown are user supplied and others are determined by the synthetic storm generator (like ending radius).



User Interface

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

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

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

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

Related Topics

- Coverages
- PBL

3.5.b. Interface Components

Interface Components

The Map module interface consists of the display options, menus, right-click menus, and tools associated with the Map module. The Map module interface is the default interface when SMS is first started.

Display Options

The display of map module features in the Graphics Window can be altered using the *Display Options* dialog. Standard viewing options (pan, frame, rotate) are also available. For more information, see: Map Module Display Options .

Menus

The map module makes use of the standard menus: *File* , *Edit* , *Display* , *Web, Window,* and *Help.*

In addition to the standard menus, the Map Module has the *Feature Objects* menu as well as right-click menus. See the following articles for more information:

- Map Feature Objects Menu
- Map Module Right-Click Menus

Tools

In addition to the standard tools , the Map module has a number of tools used for creating and modifying feature objects in the Graphics Window. Some tools contain right-click menus when clicking in the Graphics Window. For more information, see: Map Module Tools .

Project Explorer Items

In the Project Explorer, the Map module displays the available coverages and allows the coverages to be organized in folders . Right-click menus are available for both coverage items and folders. See Project Explorer Items for more information.

Related Topics

- SMS Menus
- Dynamic Tools

Map Module Display Options

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

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

The visibility of feature objects can be controlled in this section. The following options are available:

- *Node* or *Point* – Display the vertices along the nodes or points. The button next to this option allows opening a *Symbol Attributes* dialog to define the size and shape of the nodes or points.
- *Nodal BC* or *Point BC*
  - *Refine Points*
- Node or Point *ID* – Display the ID of the feature node or point. The button next to this option allows opening a *Font* dialog.
- *Arcs* – Display of feature arcs in the Graphics Window. The button next to this option allows opening a *Line Attributes* dialog where the style, width, and color of the arc can be selected.
- *Arc BC* – Activates an **Options** button that opens the *Arc Display Options* dialog with display options for the boundary condition specific to the active coverage.
- *Vertices* – Display the vertices along the arcs. The button next to this option allows opening a *Symbol Attributes* dialog to define the size and shape of the vertices.
- Arc *ID* – Display the ID of the feature arcs. The button next to this option allows opening a *Font* dialog.
- *Arc Types* – Arc attributes may be displayed depending on the coverage.
- *CL Flow Direction Arrow* – Display an arrowhead showing the direction of center line arcs.
- *Snap Preview* – Turns on or off the display of the where a feature object will align along a grid or mesh. Requires that the coverage be linked to a simulation . The rules for snapping vary depending on the coverage type, therefore, the snap preview may change when switching between coverages linked to the same simulation. *Shift + Q* can be used to turn on or off the preview mode without entering the *Display Options* dialog.
- *Polygon Fill* – Display the fill color of feature polygons. Generally the fill color matches the material data assigned to the polygon.

  - Show Materials / Mesh Type – Fill the polygons to display the materials assigned to the polygons or the mesh generation type assigned to each polygon.
  - Material Opts – Set the display options of the materials.
  - Land / Ocean – Set the color fill of land and ocean polygons. (Cartesian grid coverages only)
- Polygon *ID* – Display the ID of the feature polygons. The button next to this option allows opening a *Font* dialog.
- Grid *Frame* – Set the display of the grid frame. (Cartesian grid coverages only) The button next to this option allows opening a *Line Attributes* dialog where the style, width, and color of the grid frame can be selected.
- *Legend*
- *Inactive Coverages* – Set the color for the display of all inactive coverages.

Legend

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

- *Legend title* – Allows entering a name for the legend.
- *Location Preference*
  - *Legend location* – Allows defining the location for the legend in the graphics window. Options include: "Top Left Corner", "Bottom Left Corner", "Top Right Corner", "Bottom Right Corner", "Screen Location", or "World Location". Using the "Screen Location" or "World Location" option will activate the *X location* and *Y location* options where a more precise location can be specified. The "World Location" option will also activate the *Z location* option.
- *Font* – Clicking the button in this section brings up a *Font* dialog to set the attributes of the legend text.
- *Size*
  - *Width*
  - *Height*

Observation Coverage Only

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

Related Topics

- Map Module
- Display Options

Map Feature Objects Menu

The *Feature Objects* Menu includes the following commands:

General Commands

Delete

Deletes all the feature objects in an SMS session and creates a new blank coverage (since SMS requires an active coverage at all times). SMS will ask for confirmation of this action.

Attributes

Brings up an *Attributes* dialog. The specifics of the dialog are unique to each coverage.

Create Arc Group

Creates a new entity from a group of selected contiguous (end to end) arcs. If the selected arcs are not connected end to end, SMS will give an error message. These arc groups can be used in some models to assign boundary conditions. This command is used to create an arc group from a continuous string of selected arcs. Once the arc group is created, it can be selected using the **Select Arc Group** tool. Properties can be assigned to the arc group as a whole, and the arc group can be selected to display the computed flow through the arc group. An arc group is deleted by selecting the arc group and selecting the *DELETE* key. Deleting an arc group does not delete the underlying arcs.

Build Polygons

While most feature objects can be constructed with tools in the *Tool Palette* , polygons are constructed with the **Build Polygons** command. Since polygons are defined by arcs, the first step in constructing a polygon is to create the arcs forming the boundary of the polygon. After forming loops with arcs, choose *Feature Objects* | **Build Polygons** from the menu. The build polygons command will form polygons from all closed loops in the coverage.

Clean

Opens the *Clean* dialog which can fix certain feature object errors.

Vertices ↔ Nodes

In some cases, it is necessary to split an arc into two arcs. This can be accomplished using the **Vertex ↔ Node** command. Before selecting this command, a vertex on the arc at the location where the arc is to be split should be selected. The selected vertex is converted to a node and the arc is split in two. The **Vertex ↔ Node** command can also be used to combine two adjacent arcs into a single arc. This is accomplished by converting the node joining the two arcs into a vertex. Two arcs can only be merged if no other arcs are connected to the node separating the arcs. Otherwise, the node must be preserved to define the junction between the branching arcs.

Reverse Arc Direction

Reverses the direction of all selected arcs.

Redistribute Vertices

Automatically creates a new set of vertices along a selected set of arcs at either a higher or lower density.

Transform Feature Objects

Brings up the *Transform Feature Objects* dialog where data can be scaled, translated, or rotated.

Select/Delete Data...

Requires that one or more polygons be selected. Brings up the *Select/Delete Data* dialog.

Find

Allows finding a feature object node, arc or polygon by its specified ID number.

Map → 2D Mesh

Used to generate a 2D finite element mesh from feature objects.

Map → 2D Grid

Used to generate a 2D grid from feature objects.

Map → Scatter

Allows scatter sets to be interpolated from map data. Scatter points can be created from a specified source. Scatter points data can be extrapolated from feature points and vertices on arc or on feature points only on arc, feature points not on arcs or from feature polygon meshing options. The elevation source can be obtained from arc elevation, arc node and vertex elevations or from arc spacing.

Coverage Type Specific Menus

Optional menu items appear according to the active coverage type.

Generic Coverage Types

Stamping

- Stamp Features

Model Coverage Types

ADCIRC

- **Model Control** – Brings up the *ADCIRC Model Control* dialog.
- **Create Coastline** – Opens the *Create Contour Arcs* dialog. For more information, see Arcs: Create Contour Arcs . This command is available if the current coverage type is SHOALS, ADCIRC , or CGWAVE .
- **Define Domain** – Brings up the *Domain Options* dialog.

BOUSS-2D

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

CGWAVE

- **Model Control** – Brings up the *CGWAVE Model Control* dialog.
- **Create Coastline** – Brings up the Create Contour Arcs dialog.
- **Define Domain** – Brings up the *Domain Options* dialog.

CMS-Flow

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

CMS-Wave

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

GenCade

- **Grid Frame Properties** – Brings up the *Grid Frame Properties* dialog for GenCade.

Related Topics

- Map Module

Map Module Tools

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

Tool	Tool Name	Description

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

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

General Tool Right-Click Menus

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

- **Clear Selection** – Undoes the selection of the object that was clicked on.

- **Invert Selection** – Selects every object of the same type selected, and undoes the selection of the object that was originally selected.
- **Zoom to Selection** – Causes the object selected to fill and be centered in the Graphics Window.

Tool Specific Right-Click Menus

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

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

	Arc Group	
	**Select Feature Polygon**	• **Delete** – Deletes Feature Polygon (not the arcs that make up the polygon). • **Attributes** – Model specific.
	**Select 1D Grid Frame**	• **Properties** ' – Invokes the **Grid Frame Properties** dialog that allows specifying the location, orientation, size and spacing of the grid frame using text fields; Specific to 1D grid coverages.
	**Select 2D Grid Frame**	Type Specific • **Properties** – Invokes the **Grid Frame Properties** dialog where the 2D grid frame properties can be edited. Model specific.

Related Topics

- Map Module

Project Explorer Items

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

Map Module Right-Click Menus

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

Map Module Root Folder Right-Click Menus

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

- **New coverage** – Opens the *New Coverage* Dialog.
- **New Folder** – Creates a new folder under the Map module root folder.
- **Clear Coverages** – Deletes all coverages.
- **Display Options** – Opens the *Display Options* dialog.
- **Collapse all** – Collapses the project explorer tree under the map module root folder.
- **Expand all** – Expands all entries in the project explorer tree under the map module root folder.
- **Check all** – Selects all entries in the project explorer tree under the map module root folder as visible.
- **Uncheck all** – Deselects all entries in the project explorer tree under the map module root folder making them invisible.

Coverage Item Right-Click Menus

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

- **Duplicate** – Adds another coverage exactly identical to the existing coverage clicked on.
- **Rename** – Specifies a new name for the coverage.

- **Convert** – Converts coverage to a 2D grid, mesh or scatter object. In order to convert to a 2D Grid the coverage must be of a type that is associated with 2D grids and include a grid frame. In order to convert to a mesh, the coverage must be compatible with mesh generation and include at least one polygon. All coverages can be converted to a scatter set. Selecting this command invokes the Map → Scatter dialog.
- **Add Arcs to Mesh** – This command was added in SMS 12.0. When this command is invoked, any selected arcs in the coverage are forced directly into the active mesh and a nodestring is created following each arc. If no arcs are selected when this command is invoked, all the arcs in the selected coverage are forced into the mesh. This operation requires that all nodes and vertices on the arc lie inside the mesh it is being forced into. If this is not the case, the arc is skipped. The arc can leave the mesh and reenter, but all vertices must be inside the mesh.
- **Projection –** Sets the projection of the coverage.
- **Reprojection** – Reprojects the projection of the coverage.
- **Metadata** – Annotates the coverage.
- **Zoom to Coverage** – Zooms to area where coverage is within the graphic window.
- **Type** – Change the coverage type.

New Folder Right-Click Menus

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

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

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

3.5.c. Functionalities

Feature Objects Types

Feature objects in SMS have been patterned after Geographic Information Systems (GIS) objects and include points, nodes, arcs, and polygons. Feature objects can be grouped together into coverages . Each coverage defines a particular set of attributes that store information for the objects. Since feature objects are patterned after GIS objects, it is possible to import data from GIS applications such as ESRI Shapefiles (Arc/Info or ArcView) and MIF/MID file pairs (Map Info).

The primary use of feature objects is generate high level conceptual representations of a site. The area included by the polygons defines the domain of the mesh, grid, or limit the extents of cross sections. Each polygon represents a material zone or element type. Special points can be identified in the interior of the domain as areas of particular interest. Boundary parameters such as flow and head values can also be assigned to points or arcs. Depending on the numerical model to be used, SMS either passes this conceptual representation to the model, or constructs finite element meshes , finite difference grids, or one-dimensional cross sections that a numerical model will use. Thus, it's possible to focus on a simplified, high level representation of the model and little or not tedious cell-by-cell editing is required. The conceptual model approach can be used to build models for any of the numeric models supported by the SMS interface.

Feature Object Types

The definition of feature objects in SMS follows that used by typical GIS software that supports vector data. The basic object types are points, nodes, vertices, arcs, and polygons. The relationship between these objects is illustrated in the figure below.

Points

Points are XY locations that are not attached to an arc. Points have unique ids and can be assigned attributes such as a source or sink. Points are often used to refine a mesh in an area of interest. Points are also used when importing a set of XY locations for the purpose of creating arcs or polygons.

Arcs

Arcs are sequences of line segments or edges, which are grouped together as a single "polyline" entity. Arcs have unique ids and can be assigned attributes such as specified head. Arcs are grouped together to form polygons or are used independently to represent geometrical features such as ridges or channels. The two end points of an arc are called "nodes" and the intermediate points are called "vertices".

The vertices in an arc define the shape. As more vertices are added, the shape can be more complex. An arc is split into to arcs by selecting a vertex in the arc and converting that vertex to a node. Two adjacent arcs are merged into a single arc by selecting the node that joins them, and converting it to a vertex. Several tools or utilities are provided for working with arcs. These can be accessed while the select arc tool is active by right clicking in the graphics window. Some of the tools also require that at least one arc be selected. The tools include:

- Delete the selected (or all) arcs.
- Filter arc(s)
- Split arc(s)
- Offset arc(s)
- Redistribute vertices
- Reverse arc direction
- Smooth arc(s)
- Transform
- The standard selection utilities (clear, invert, zoom to selection)
- Select connected arcs

Nodes

Nodes define the beginning and ending XY locations of an arc. Nodes have unique ids and can be assigned attributes.

Vertices

Vertices are XY locations along arcs in between the beginning and ending nodes. They are used solely to define the geometry of the arcs. Vertices do not have ids or attributes.

Polygons

Polygons are a group of connected arcs that form a closed loop. A polygon consists of one or more arcs. If two polygons are adjacent, the arc(s) forming the boundary between the polygons is shared (not duplicated). Polygons may not overlap. However, a polygon can have a hole(s) defined by having a set of closed arcs defining interior polygon(s). An example of a hole is shown in the figure below. In this case, four arcs define two polygons. Polygon A is made up of arcs 1, 2, 3 and 4, whereas polygon B is defined by a single arc (arc 2). For polygon A, arcs 1, 3, and 4 define the exterior boundary whereas arc 2 defines a hole.

Polygons have unique ids and can be assigned attributes. Polygons are used to represent material zones such as main channel, overbank flood plain, lakes, etc.

Coverages

Feature objects are grouped together into coverages. Each coverage represents a particular set of data. For example, one coverage can be used to define recharge zones, and another coverage can be used to define zones of hydraulic conductivity.

Conceptual Models

Coverages are grouped into conceptual models. Conceptual models may consist of multiple coverages. In simple cases like TABS (RMA2), an "RMA2" meshing coverage may be combined with an "Area property" coverage to define material zones. When converting an RMA2 coverage to a mesh, SMS allows specifying this option. For more complex conceptual models, such as those used for TUFLOW , a simulation entry in the tree includes links to the component coverages. A TUFLOW simulation may have coverages for HX links, cross section, boundary conditions and levies.

Related Links

- Converting Feature Objects
- Feature Objects Menu
- Shapefiles

Attributes in the Feature Objects Menu

Feature object attributes are dependent on the coverage type. If a feature object has attributes, the attributes are edited by selecting the feature object and then selecting the menu command *Feature Objects* | **Attributes** or select **Attributes** from the right-click mouse menu. An *Attibutes* dialog will appear with options for the specific selected feature object in relation to the coverage type containing the feature object.

Feature Point Attributes

The following coverages can apply attributes to points or nodes.

Generic Coverage Point Types

- Observation – Uses the *Observation Coverage* dialog.
- Spectral – Uses the *Spectral Energy* dialog.
- Stamping – Uses the *Stamping Point Attributes* dialog.

Model Coverage Point Types

- ADCIRC – Uses the *Refine Attributes* dialog.
- ADH
  - ADH – Use the *ADH Boundary Condition Assignment* dialog.
  - Vessel – Use the *Vessel Node Properties* dialog.
- BOUSS-2D
  - Wavemaker – Uses the *BOUSS-2D Wave Generator Properties* dialog.
- CGWAVE – Uses the *Refine Attributes* dialog.
- CMS-Flow
  - Save Points – Uses the *Assign Save Points* dialog.
- CMS-Wave – Uses the *Refine Attributes* dialog.
- FESWMS – Uses the *'FESWMS Point/Node Attributes* dialog.
- GenCade – Uses the *Refine Attributes* dialog.
- Generic Model – Uses the *Feature Point/Node Attributes* dialog.
- PTM
  - PTM – Uses the *Feature Object Attributes* dialog.
  - Gages – Uses the *PTM Gage Attributes* dialog.
- SRH-2D
  - Obstructions – Uses the *Obstructions* dialog.
- TABS – Uses the *Feature Point/Node Options* dialog.
- TUFLOW
  - 1D Network – Uses the *Node Attributes* dialog.
  - 1D Water Level Points – Uses the *Materials Data* dialog.
  - 1D–2D BCs and Links – Uses the *Boundary Conditions* dialog.
  - 2D Flow Constriction Shape – Uses the *Flow Constriction Point* dialog.
- 2D Z Lines (advanced) – Uses the *Point Attributes* dialog.

Feature Arc Attributes

The following coverages can apply attributes to arcs.

Generic Coverage Arc Types

- Mapping – Uses the *Arc Attributes* dialog.
- Observation – Uses the *Observation Coverage* dialog.
- Stamping – Uses the *Stamping Arc Attributes* dialog.

Model Coverage Arc Types

- ADCIRC – Uses the *ADCIRC Arc/Nodestring Attributes* dialog.
- ADH
  - ADH – Use the *ADH Boundary Condition Assignment* dialog.
- BOUSS-2D – Uses the *Cartesian Grid Arc Options* dialog.
- BOUSS Runup/Overtopping
  - Probes – Uses a *Arc Attributes* dialog.
  - Damping – Uses a *Damping Properties* dialog.
  - Porosity – Uses a *Porosity Properties* dialog.
  - Transects – Uses the *XY Series Editor* .
  - Wave Maker – Uses the *BOUSS-2D Wave Generator Propertise* dialog.
- CGWAVE – Uses the *CGWAVE Boundary Conditions* dialog.
- CMS-Flow
  - Boundary Conditions – Uses the *Arc Boundary Conditions* dialog.
- FESWMS – Uses the *Feature Arc Attributes* dialog.
- GenCade – Uses the *GenCade Arc Attributes* dialog.
- Generic Model – Uses the *Feature Arc Attributes* dialog.
- PTM
  - PTM – Uses the *Feature Object Attributes* dialog.
- SRH-2D
  - Boundary Conditions – Uses the *SRH-2D Linear BC* dialog.
  - Obstructions – Uses the *Obstructions* dialog.
- TABS – Uses the '''Feature Arc Attributes *dialog.*
- TUFLOW
  - 1D Cross Section – Uses the *TUFLOW Cross Section* dialog.
  - 1D Network – Uses the *Channel Attributes* dialog.
  - 1D Water Level Lines – Uses the *Water Level Arc Attributes* dialog.
  - 1D–2D BC and Links – Uses the *Boundary Conditions* dialog.
  - 1D/2D Connections – Uses the *Arc Attributes* dialog.
  - 2D Flow Constriction Shape – Uses the *Flow Constriction Attributes* dialog.
  - 2D Grid Extents – Uses the *Cartesian Grid Arc Options* dialog.
  - 2D Z Lines (advanced) – Uses the *Z Shape* dialog.
  - 2D Z Lines/Polygons (simple) – Uses the *Choose Arc Type* dialog.

    - 2D/2D Linkage –

Polygon Attributes

The following coverages can apply attributes to polygons.

Generic Coverage Polygon Types

- Area Property – Uses the *Land Polygon Attributes* dialog.
- Mesh Generator – Uses the *2D mesh Poylgon Properties* dialog.
- Mapping – Uses a *Polygon Attributes* dialog.
- Quadtree Generator – Uses a *Polygon Attributes* dialog.

Model Coverage Polygon Types

- ADCIRC – Uses the *2D Mesh Polygon Properties* dialog.
- ADH
    - ADH – Uses the *2D Mesh Polygon Properties* dialog.
- BOUSS-2D – Use the *Polygon Attributes* dialog.
- BOUSS Runup/Overtopping
    - Roughness – Uses the *Roughness* dialog.
- CGWAVE – Uses the *2D Mesh Polygon Properties* dialog.
- FESWMS – Uses the *2D Mesh Polygon Properties* dialog.
- Generic Model – Uses the *2D Mesh Polygon Properties* dialog.
- PTM
    - PTM – Use the *Feature Objects Attributes* dailog.
- SRH-2D
    - Materials – Uses the *Assign Material Properties* dialog.
    - Sediment Materials – Uses the *Assign Material Properties* dialog.
- TABS – Uses the *2D Mesh Polygon Properties* dialog.
- TUFLOW
    - 1D–2D BC and Links – Uses the *Boundary Conditions* dialog.
    - 2D Flow Constriction Shape – Uses the *Flow Constriction Attributes* dialog.
    - 2D Flow Constriction (cell based) – Uses the *Flow Constrictions* dialog.
    - 2D Grid Extents – Uses the *Polygon Attributes* dialog.
    - 2D Miscellaneous (FLC, WRF, IWL, SRF and AD) – Uses a *Polygon Properties* dialog for the coverage property type.
    - 2D Z Lines/Polygons (simple) – Uses the *Polygon Elevation* dialog.
    - 2D/2D Linkage – Uses the *Select TUFLOW Grid* dialog.
- WAM – Uses the *Polygon Attributes* dialog.

Related Topics

- Feature Objects Menu

Map Module Selection

Beyond using the selection tools in the Map Module, there are a number of menu commands and dialogs that can be used to refine the selection of feature objects.

Select With Poly

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, it is possible to 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

The *Select By* option, in the *Edit* menu, allows selecting an object by its Material Type, Dataset Value, Area, Length, or Ambiguous Gradient.

Material Type

Selects all items of the current selection tool of a specified material. This command opens the *Materials Data* dialog with a list of the defined materials and waits for a material type to be selected. This enables all nodes or elements that reference a specific material to be selected together.

Dataset Value

Opens a dialog that asks to specify a range. 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

Opens a dialog that asks to specify a range. 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

Opens a dialog that asks to specify a range. 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.

Ambiguous Gradient

Selects all elements in a mesh or cells in a grid where the directional flow is difficult to determine due to variation in the elevation at each node.

BC Type

Brings up a *Select Arc Type* dialog if the active coverage is a boundary conditions coverage (CMS-Flow , SRH-2D , TUFLOW , etc.). Allows selecting arcs by assigned boundary condition type.

Select Connected Arcs

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

Select Connected Arcs Turning Left

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

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

Related Topics

- Map Module
- Edit Menu

3.5.c.1. Feature Object Creation

Digitize

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

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

Tips for Digitizing

The following are some useful tips for digitizing feature objects:

- Use the **Create Feature Arc** tool to make arcs. Click once to start an arc. Single click along the arc to create vertices. Double-click to end an arc.
- Nodes indicate the start and end of each arc. Vertices are points where the arc changes direction.
- Clicking on an existing node with the **Create Feature Arc** tool will start the new arc on the existing node. Nodes can belong to multiple arcs.
- Use the **Select Feature Node** or **Select Feature Vertex** to move nodes on an arc that may need adjustment after creating an arc.
- For long arcs, they can be created in using multiple arcs then merging the arcs by deleting the shared nodes where the arcs meet or converting the shared nodes into vertices using the **Convert to Vertex** command.
- Nodes can be added to existing arcs.
- Points are created separate from any arcs.
- Multiple points, arcs, nodes, or vertices can be selected holding down the *SHIFT* key while selecting.
- Polygons are not created automatically. The *Feature Objects* | **Build Polygons** command must be used to create polygons.
- Multiple arcs can be joined into an arc group. Creating an arc group allows using the **Select Feature Arc Group** tool to select all the arcs in the group at once and assign the same attributes to all arcs in the group.
- The image transparency can be adjusted to make seeing the feature objects easier.
- Different feature objects may need to be created in different map coverages depending on the intended model. For example, if intending to use the SRH-2D model, the project domains will need to be digitized in the boundary conditions coverage while any obstructions will need to be digitized in the obstructions coverage.
- The processing of digitizing can be sped up by duplicating coverages that already have many of the necessary feature objects then changing the coverage type and editing the feature objects. Duplicating a coverage is done by right-clicking on it in the Project Explorer and selecting the **Duplicate** command.
- The processing of digitizing can also be sped up by merging coverages by selecting multiple coverages using the *Ctrl* key then using the **Merge Coverages** right-click command.
- Currently, SMS does not have an undo feature. If a mistake is made when creating a feature object, the object will need to be either edited using the selection tools or the object will need to be deleted then recreated.

Related Topic

- Map Module Tools

Build Polygons

While most feature objects can be constructed with tools in the Tool Palette, polygons are constructed with the *Feature Objects* | **Build Polygons** menu command. Since polygons are defined by arcs, the first step in constructing a polygon is to create the arcs forming the boundary of the polygon. All closed loops will be formed into polygons.

Before defining material zones or creating meshes from a coverage, the **Build Polygons** command must be used.

Once polygons exist in the project, the **Select Polygon** tool becomes active. If a polygon cannot be selected with the **Select Polygon** tool, check to see if the arcs in the polygon are making a closed loop by removing any gaps, then use the **Build Polygons** command again.

Generally, nodes and vertices can be moved on a polygon to reshape it without causing the polygon to disappear. Deleting an arc in the polygon so that it no longer makes a closed loop, will cause polygon to be removed and the **Build Polygons** command will need to be used again if the closed loop is reformed. If creating an additional closed loop after using the **Build Polygons** command, such as creating a new arc to split an existing polygon, the command must be used again to create the new polygon.

Related Topics

- Feature Objects Menu
- Feature Objects Types

3.5.c.2. Feature Object Modification

Feature Object Modification: All

SMS provides a number of tools that facilitate modification of feature objects.

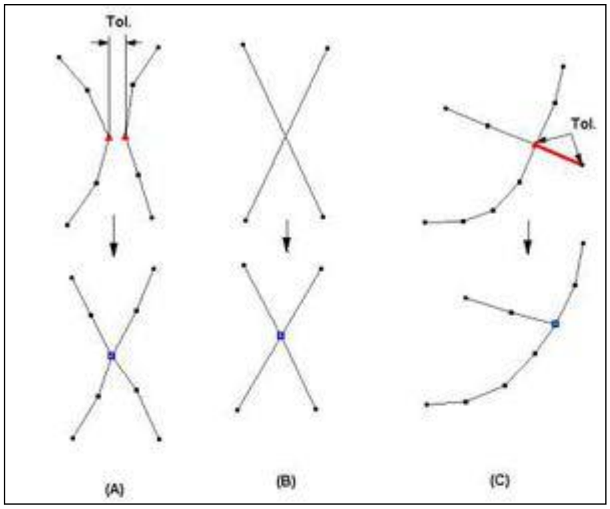
Transform Feature Objects

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

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

For more information, go to Data Transform .

Clean



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

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

Delete

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

Related Topics

- Feature Objects Menu

Converting Coverages

Map coverages can be used to generate other geometric data types in SMS such as meshes (unstructured grids), grids (Cartesian grid), Curvinlinear or boundary fitted grids, scatter sets and cross sections. This can be accomplished by right-clicking on a coverage of an appropriate type in the Project Explorer and selecting a convert command. These commands include:

- **Map → 2D Mesh** – Available with the Mesh Generator coverage and most other coverages.
- **Map → 2D Grid** – Available with the CGrid Generator coverage and many other coverages.
- **Map → Scatter** – Available with most coverages.
- **Map→1D Grid** – Available in the GenCade coverage.

- **Map → Quadtree** – Available in the Quadtree Generator coverage.

In some cases, and historically, many of these functions are accessible by selecting the similar commands from the *Feature Objects* menu.

The same coverage can be used to create multiple meshes, grids, or scatter sets, with the exception of the 1D grid. Any changes made in the coverage after converting the coverage will not implemented in the existing geometry, but a new geometry can be generated that incorporates the changes to the coverage.

Some conversion commands require specific conditions be met before the command is available.

- Converting to a grid (2D grid, 1D grid, or quadtree) requires that a grid frame has been created on the coverage. Attributes for the grid frame can be set before or during the conversion process.
- Converting to a mesh requires building polygons in the coverage. Attributes for the polygons should be set before converting to a mesh.

For more information, see Converting Feature Objects .

Related Topics

- Converting Feature Objects
- Map Coverages

Converting Feature Objects

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

Extract Cross Section

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

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

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

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

Cross Section Database

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

Map to 2D Mesh

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

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

1) Define a bathymetric source (scatter set or raster/DEM).
2) Define a map module coverage consisting of polygons that cover the modeling domain. This is the region to be covered by the mesh.
3) Assign attributes to the points/arcs/polygons in the coverage to control the mesh characteristics.
    - Point meshing attributes:
        - Used to force the creation of a mesh node at a specific location.
        - Used to specify the element density in the area of the point location by assigning refine point attributes.
    - Arc meshing attributes:
        - Used to define linear features such as a river thalweg or an embankment toe/shoulder. Mesh nodes will be created along the arc.
        - Used to control element density if a size function (scalar paving) is not utilized. Vector spacing on the arc controls mesh node spacing for all mesh generation options except scalar paving.
    - Polygon meshing attributes:
        - Specify a bathymetry source for each polygon
        - Specify a meshing type for each polygon. Choose from:
            - Patching – create quad dominant elements conforming to a topographic rectangle.
            - Paving – create triangular elements layer by layer from the polygon boundary inward.
            - Scalar Paving – create triangular elements as with paving with the spacing controlled by a size function defined on an associated scatter set
4) Optionally, define an area property coverage to define the source of material attributes.
5) Issue the **Map → 2D Mesh** command is used to create a 2D mesh using the feature objects in a 2D Mesh Coverage. When the **Map → 2D Mesh** command is selected, the *2D Mesh Options* dialog opens.
6) After completing the *2D Mesh Options* dialog, the *Mesh Name* dialog will appear. After assigning a name to the new mesh, the mesh will appear.

Mesh Name Dialog

When using the **Map → 2D Mesh** , **2D grid → 2D Mesh** , or **Scatter → 2D Mesh** commands, the *Mesh Name* or *New Mesh Name* dialog will appear. SMS requires that every mesh created be named. These dialogs give the option to enter a user specified name. A default name will be entered in the dialog and used for the new mesh if a new name is not specified.

Map to 2D Grid

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

Grid Frame Properties

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

- *Grid name* – Specified name of the grid being created.
- *Origin* – Starting location of the grid frame.
- Orientation
- Directional properties (u and v direction)
- Define cell sizes – Specified uniform cell sizes
  - *Cell size* – The cell size in the specified direction
  - *Number of cells* – Number of cells in the specified direction
- Use refine points – Refine points will be used to generate the grid
  - *Maximum cell size* – The max size the should exists when growing
  - *Maximum bias* – The max growth ratio to be used when growing
  - *Use inner growth* – Specifies whether the cell sizes should grow between two refine points
- *Grid siz* e – The grid dimension in the specified direction

When specifying "Define cell sizes", there are a few options available. These options are:

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

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

Refine Points

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

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

Depth and Vector Options

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

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

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

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

Map to 2D Scatter Points

The **Map → 2D Scatter Points** command creates a scatter point set from the points and nodes and vertices of the current coverage. The process is creates a single elevation dataset for the 2D scatter points representing the Z location of all the points, nodes and vertices.

Older versions of SMS included an option to convert one of the measurements associated with an observation coverage to a dataset on a scatterset. This utilized the *Observation Points → Scatter Points* dialog. To accomplish this in versions of SMS newer than SMS 10.2, you must copy the observation spread sheet into Excel and save the data as a text file. Importing this file into SMS will allow creation of a scatter set of observed (measurement) data.

Measurement

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

Time Step Times

This section of the dialog is only available if the selected measurement is transient. It allows defining the number of time steps, and the time step times to be created for the scatter point dataset.

- **Match all unique times**

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

- **Match time steps from model**

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

- **Specify times**

  The spreadsheet of times will be editable with this option. It is possible to copy and paste times from another program such as a spreadsheet. Also, the **Initialize Times** button becomes available which brings up a dialog that can be used to create times at a specified interval. If selecting the *Use dates/times* toggle, the *Reference time* section will become available and the times in the spreadsheets will be displayed as dates/times.

Map to Quadtree

Parameters specified to create the quadtree grid include:

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

SMS will generate a quadtree on the input parameters.

Map to 1D Grid

Related Topics

- Map Feature Objects Menu

Unstructured Grid Generation from a Conceptual Model

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

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

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

Feature Polygon Attributes

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

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

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

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

There is also an option to specify the material type that will be assigned to newly created elements.

Feature Arc Attributes

The feature arcs in the conceptual model serve three purposes.

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

Feature Point Attributes

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

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

Select/Delete Data...

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

- *Function Type*

  - *Select* – Selects objects
  - *Delete* – Deletes objects
- *Data Domain*
  - *Inside polygons(s)* – Trim data inside selected polygons.
  - *Outside polygon(s)* – Trim data outside selected polygons.
  - *Treat boundary as [Outside | Inside]* – Treat data that lies on polygon boundaries as if it were outside/inside of the polygon(s).
- *Choose Data to Select* – Choose the data type to select or delete
  - *Mesh* – Select or delete mesh data with the following options:
    - *Nodes* (Select) – Select mesh nodes only.
    - *Nodes and Elements* (Delete) – Trim mesh nodes and elements.
    - *Elements* – Select or delete mesh elements only.
  - *Scatter* – Select or delete scatter data with the following options:
    - *Points* (Select) – Select scatter points only
    - *Points and Triangles* (Delete) – Trim scatter points and triangles.
    - *Triangles* – Trim triangles only.
    - *Scatter Sets to Trim* – Click on a scatter set in the window to toggle it on/off for trimming. Push the **Active Set** button to only select the active scatter set and the **All Sets** button to turn on/off all sets. Points and/or triangles are deleted only from the selected sets.
  - *Cartesian grid cells* – Selects or deletes all grid cells.
  - *Cartesian grid cell locations* – This can be useful in working with elevation values in TUFLOW grids.

Related Topics

- Map Feature Objects Menu

Arcs

SMS has many utilities that can be used to modify feature arcs. Some of the utilities are described in this article.

Split Feature Arcs Utility

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

This utility is in the right-click menu when selecting one or more arcs. The command brings up the *Split Arcs Tool* dialog. In this dialog provide criterion for splitting the selected arc(s) into multiple arcs. If desiring to process all the arcs in a coverage, use **Select All** first.

There dialog includes three toggle boxes associated with criteria including:

- **Split long arcs** – If this is selected, specify a length (labeled as ft or m) based on the current projection. Processed arcs will be split into arcs of this specified length starting at the first of the arc. The last arc in the group will generally not be the specified length.

- **Split sharp corners** – If this is selected, specify an angle (labeled as degrees). This is the angle of deviation from one arc segment to the next. A straight line has an angle of deviation of 0.0. If the arc doubled back on itself completely, the angle of deviation is 180 degrees. This deviation is a magnitude. A left turn is processed identically to a right turn. Processed arcs will be split at vertices where the arc direction changes more than the specified angle.
- **Split long segments** – If this is selected, specify a threshold length. If a segment is longer than this threshold, the vertices at either end will be converted to nodes making it a separate arc.

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

Offset Arc

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

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

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

This command has several useful applications. These include:

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

Redistribute Vertices

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

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

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

The following options are available for redistributing vertices:

- "Specified spacing" – Will redistribute the vertices along the applying the *Average* and *Bias* parameters to determine the distance between each vertex.
- "Number of segments" – Will create or remove vertices along the arc, as well as redistribute, based on the *Number of* vertices parameter.

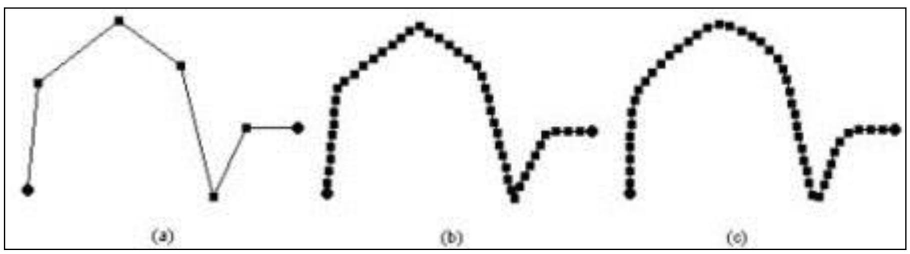
- "Min/max spacing" – Will have vertices closer together at the start of the arc and gradually increase the segment lengths up to the *Max length* parameter.
- "Source arc" – See below.
- "Size Function" – Allows using a size function dataset for redistribution.
- *Use Cubic Spline* – Option for spline interpolation .

Linear Interpolation

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

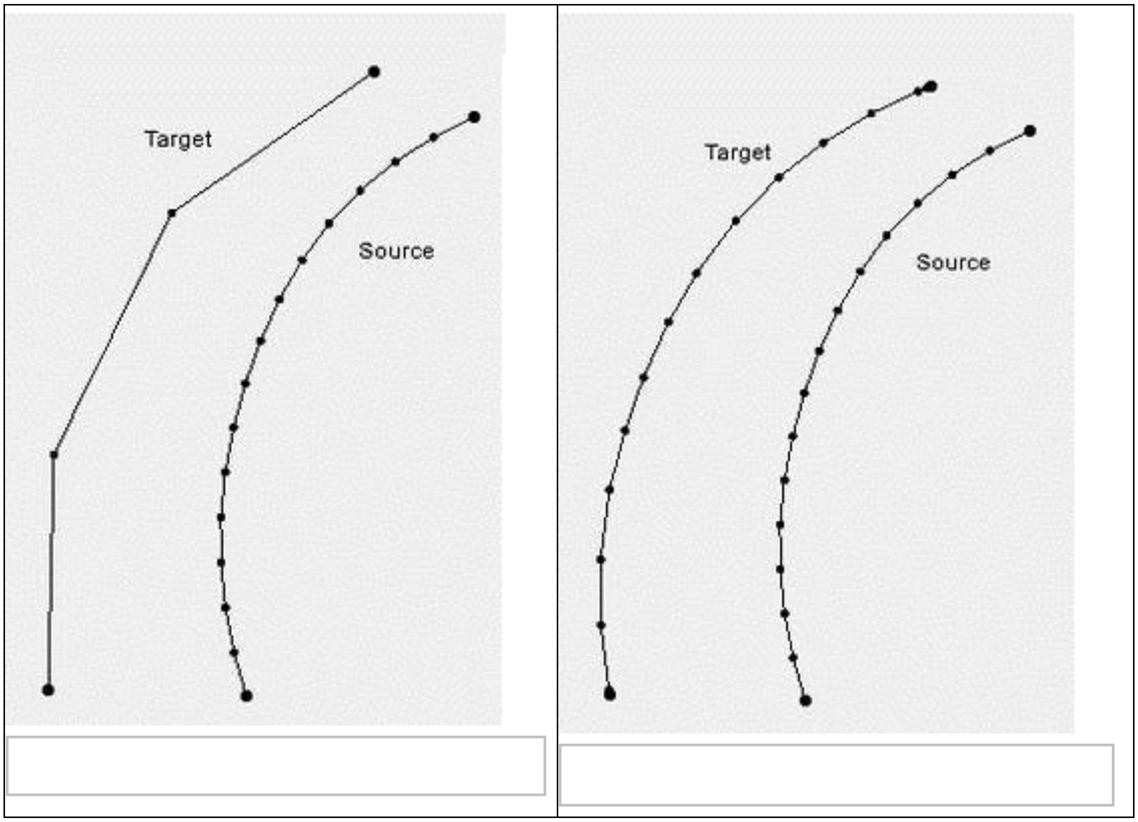
Spline Interpolation

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



Source Arc

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



Reverse Arc Direction

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

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

Smooth Arc

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

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

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

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

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

Transform

See the article Data Transform

Select Connected Arcs

See the article Select Connected Arcs .

Create Arc Group

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

Create Contour Arcs

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

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

The dialog allows specifying the following attributes:

- *Destination Coverage* – Clicking on this button will bring up a *Select Tree Item* coverage where an existing coverage can be designated to receive the contour arcs.
- *Elevation* – Designates an elevation level SMS will use to create the contour arcs. If the value enter does not match any of the existing elevation, no arcs will be generated.
- *Spacing* – Indicates the distance between vertices along the generated arcs.

Align Arc With Contour

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

Arc Attributes Dialog

Feature Arc Attributes depend on the type of coverage being used. A purely geometric arc would have no special attributes, but arcs may be used to define specific types of features such as a coastline, open ocean boundary, stream thalweg, or edge of water. In addition, feature arcs may have model specific boundary condition definitions which should be applied to a geometric object of that model type at the location of the arc.

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

Recompute All Stations

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

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

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

Filter Arc

The *Filter Arcs Tool* dialog is accessed through the right-click menu of a selected arc. The dialog allows filtering all arcs on a coverage. Options include:

- *Coverage to filter* – Selects the coverage to filter.
- *Maximum arc length* – Set a value for the maximum arc length. Arcs with a length under this value will be selected or deleted.
- *Select filtered items* – Arcs that fall below the maximum segment length will be selected.
- *Delete filtered items* – Arcs that fall below the maximum segment length will be deleted.

Filter Arc Segments

The *Filter Arcs Segments Tool* dialog is accessed through the right-click menu of a selected arc. The dialog allows filtering all arcs on a coverage. Options include:

- *Coverage to filter* – Selects the coverage to filter.
- *Maximum segment length* – Set a value for the maximum segment length. Segments with a length under this value will be selected or deleted.
- *Select filtered items* – Arc segments that fall below the maximum segment length will be selected.
- *Delete filtered items* – Arc segments that fall below the maximum segment length will be deleted.

Related Topics

- Feature Objects Types
- Map Module
- Map Feature Objects Menu

Arc Size Function

Vertices along an arc can be redistributed using a size function dataset. This options is selected in the *Redistribute Vertices* dialog. Once the "Size Function" option is set, a dataset must be assigned. Clicking the **Options** button will bring up dialogs that allow assigning the size function.

Interpolation Source and Options Dialog

The *Interpolation Source and Options* dialog is used to specifiy the size function dataset and specify the interpolation method.

- *Source geometry* – Shows all available source data in the scatter module.
- *Source dataset* – Select the specific size function dataset in the source data.
- *Time step* – Select the time step to use in a transient dataset.
- *Interpolation method* – Options include "Linear", "Inverse distance weighted", and "Natural neighbor".
- **Options** – Brings up the *Interpolate to Raster* dialog for the specified interpolation method. See below for more details.

Linear

When using linear interpolation , the *Interpolate to Raster – Linear* dialog allows the following options:

- *Truncate value* – Allows limiting the interpolation values with one of the following methods:
  - *Truncate to min/max of dataset* – The interpolation will not use values that go below the minumum value or above the maximum value of the size dataset.
  - *Truncate to specified range* – The interpolation will not use values below or above the values defined below:
    - *Min* – The lowest value to be used in the interpolation.
    - *Max* – The highest value to be used in the interpolation.
- *Clough-Tocher* – Use the Clough-Tocher interpolation technique.

Inverse Distance Weighted

When using inverse distance weighted interpolation , the *Interpolate to Raster – Inverse Distance Weighted* dialog allows the following options:

Nodal Function – Specify the nodal function to use in the interpolation.

- *Constant (Shepard's Method)* – Use the Shepard's method equation.
  - *Use classic weight function* – Option to specify the classic weight function as entered below.
  - *Weighting exponent* – Value for the weight function.
- *Gradient plane* – Use the gradient plane nodal function variation of the Shepards method.
- *Quadratic* – Use quadratic polynomials to reproduce local variations.

Computation of nodal function coefficients – Options in this section indicate to use a subset of points in the nodal function coefficient.

- *Use nearest* – Drops points that are further than the indicated value.
- *in each quadrant* – When turned on, the nearest points in each quadrant are used in the subset.
- *Use all points* – All points will be used in the computation.

Computation of interpolation weights – Options in this section indicate to use a subset of points in the interpolation weight.

- *Use nearest* – Drops points that are further than the indicated value.
  - *in each quadrant* – When turned on, the nearest points in each quadrant are used in the subset.
- *Use all points* – All points will be used in the computation.

Truncate

- *Truncate value* – Allows limiting the interpolation values with one of the following methods:

- *Truncate to min/max of dataset* – The interpolation will not use values that go below the minumum value or above the maximum value of the size dataset.
- *Truncate to specified range* – The interpolation will not use values below or above the values defined below:
  - *Min* – The lowest value to be used in the interpolation.
  - *Max* – The highest value to be used in the interpolation.

Natural Neighbor

When using natural neighbor interpolation , the *Interpolate to Raster – Natural Neighbor* dialog allows the following options:

Nodal Function

- *Constant* – Use the standard natural neighbor interpolation equation.
- *Gradient plane* – Use the gradient plane nodal function variation.
- *Quadratic* – Use quadratic polynomials to reproduce local variations.

Nodal function computed from – Options in this section indicate to use a subset of points in the nodal function.

- *Use nearest* – Drops points that are further than the indicated value.
  - *in each quadrant* – When turned on, the nearest points in each quadrant are used in the subset.
- *Use all points* – All points will be used in the computation.

Truncate

- *Truncate value* – Allows limiting the interpolation values with one of the following methods:
  - *Truncate to min/max of dataset* – The interpolation will not use values that go below the minumum value or above the maximum value of the size dataset.
  - *Truncate to specified range* – The interpolation will not use values below or above the values defined below:
    - *Min* – The lowest value to be used in the interpolation.
    - *Max* – The highest value to be used in the interpolation.

Advanced

Clicking the **Advanced** button in any of the *Interpolate to Raster* dialogs will open the *Interpolate to Raster – Advanced* dialog. The options in this dialog are rarely used in most projects. The options include:

Anosotropy – Allows accounting for data with directional tendencies.

- *Horizontal anisotropy*
- *Azimuth*
- *Vertical Anisotropy (1/z mag.)* – Minimizes the effects of clustering along vertical traces.

Extrapolation

- *Value* – Assigns a defaults value to interpolation points outside of the convex hull of the scatter set.
- *Assign extrapolation value to hidden objects* – Assign the default extrapolation value to all points that are hidden.

Related Topics

- Redistribute Vertices
- Size Function

Feature Object Commands

SMS contains a numbers commands for modifying feature objects.

General Feature Object Commands

The majority of feature objects commands can be found in the

menu.

Define Domain

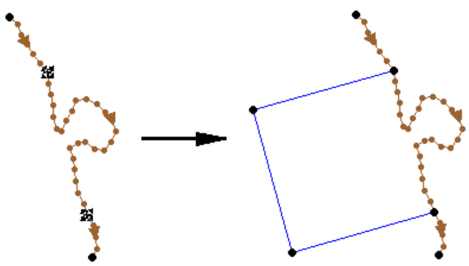
Selecting the **Define Domain** command will bring up the *Domain Options* dialog. This option is found in the *Feature Objects* menu, in the Map module, when CGWAVE or ADCIRC is the active coverage type. A domain is the region to be filled by a finite element mesh. An ocean arc is created that connects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

1) Select one or more coastline arcs. The ocean arc starts and ends at the extreme ends of the chain of arcs.
2) Select two vertices or two nodes. The ocean arc starts and ends at the nodes.

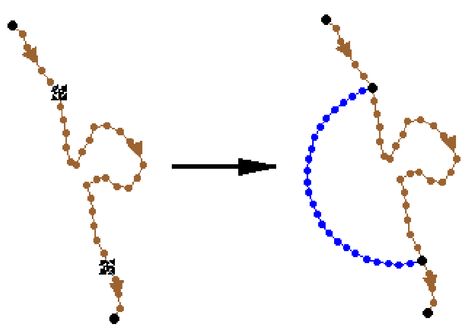
3) Select a single node or vertex along the coastline arc. Define the radius of the ocean arc and the ocean arc starts and ends where it intersects the coastline arc.
4) Select a single disjoint point. If the point is inside a closed coastline arc (an island), specify the radius of the ocean arc. The ocean arc is a circle that encloses the island.

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

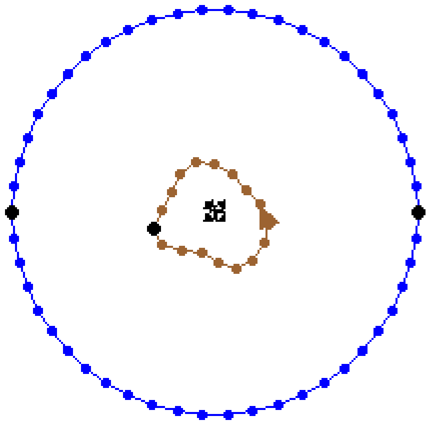
- *Rectangular* – If selecting two points or an arc, only set the *Offshore Length* . If specifying a single point, set both *Offshore Lengths* and *Along Shore Lengths* .



- *Semi-circular* – If selecting two points or an arc, do not specify the radius, the radius is the distance between the points. If specifying a single point, set the *Radius* .



- *Circular* – Sets the *Radius* . This option only works if one point is selected.



Right-Click Menu

Right-clicking on a feature object will bring up a menu. Many of the commands in this menu are standard commands accessible through other menus. Additional commands include the following:

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

Coverage Type Specific Menus

Optional menu items appear according to the active coverage type.

Generic Coverage Types

- **Create Coastline**

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

- **Stamping**

  See the article Feature Stamping for more information.

Model Coverage Types

- **ADCIRC**
  - **Model Control** – See ADCIRC Model Control for more information.
  - **Create Coastline** – See above for more information.
  - **Define Domain** – See above for more information.
- **BOUSS-2D**
  - **Create Coastline** – See above for more information.
- **CGWAVE**
  - **Model Control** – See CGWAVE Model Control for more information.
  - **Create Coastline** – See above for more information.
  - **Define Domain** – See above for more information.
- **CMS-Flow**
  - **Create Coastline** – See above for more information.

- **CMS-Wave**
  - **Create Coastline** – See above for more information.
- **GenCade**
  - **New Grid Frame** – See the article Grid Frame Properties for more information.

Related Topics

- Map Module
- Feature Objects Menu
- Feature Modification

3.6. Mesh Module

Mesh Module

The 2D Mesh Module includes tools to store, interrogate and manipulate 2D unstructured grids (referred to as a mesh inside of SMS).

The Mesh module is included with all paid editions of SMS.

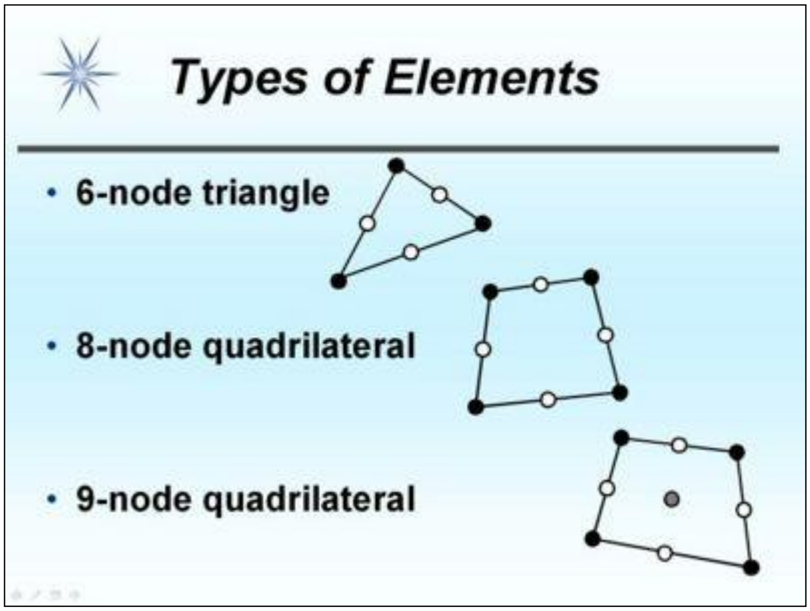
Mesh

A mesh consists of nodes that are grouped together to form elements. These nodes and elements define the computational domain of the numerical model. A numerical simulation requires a geometric definition of its domain. For many numerical analysis codes, this definition is a mesh.

Nodes

Nodes are the basic building blocks of elements in a mesh. A node consists of a location (X,Y) with an associated elevation. Other dataset values can also be associated with a node. In the traditional mesh based definition of simulations, nodes can also be used for building nodestrings and assigning boundary conditions. The density of mesh nodes helps determine the quality of solution data and can be important to model stability. See 2D Mesh Nodes Menu for more information.

Elements



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

1D elements

- Three-node line

Triangular Elements

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

Quadratic (order of solution) Elements

- Eight-node "serendipity" quadrilateral
- Nine-node "Lagrangian" quadrilateral

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

Mesh Datasets

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

Mesh Based Simulation

Traditionally, the geometry has been the basis of the simulation in SMS. This is referred to as a geometry centric model. For mesh based models, this means that in addition to its geometric attributes (nodes and elements), the mesh also stores the simulation parameters, including model settings, boundary conditions and areal properties such as material types.

Starting with version 9 of SMS, an alternative, explicit simulation based modeling approach was introduced for the TUFLOW model. Since that version a slow migration has been occurring moving away from the geometry based method.

Node/Element properties

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

Nodestrings

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

Mesh Generation

See Mesh Generation for more information.

Editing a Mesh

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

Mesh Visualization

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

Advantages of a Mesh

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

Mesh Models

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

- ADCIRC
- ADH
- CGWAVE
- FESWMS-2DH
- Generic Mesh – SRH-2D
- Generic Model
- TABS – (RMA2, RMA4)

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

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

Current Numerical Model

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

Mesh Module Tools

See Mesh Module Tools for more information.

Mesh Module Menus

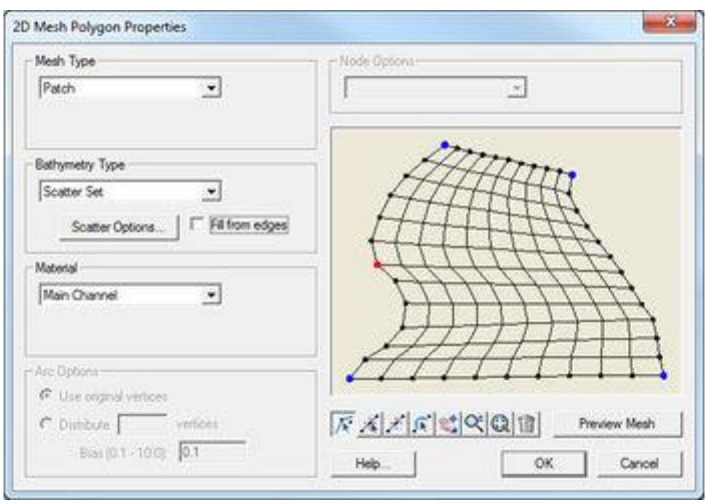
See 2D Mesh Module Menus for more information.

Related Topics

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

3.6.a. Mesh Generation

Mesh Generation



At a glance

- 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 *Dataset Toolbox*, or LTEA (linear truncation error analysis) (ADCIRC)

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

Using a Conceptual Model

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

The mesh generation capability is based on feature objects .

Mesh From Coverage

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

- Patch
- Paving
- Scalar Paving Density

Mesh Generation Toolbox

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

Mesh Generator Coverage

Starting in SMS 12.0, a generic Mesh Generator coverage can be created. This coverage replaces previous coverages for creating a mesh from the Map module.

Manually Creating a 2D Mesh

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

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

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

Creating a 2D Mesh from Existing Geometry

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

- **TIN → 2D Mesh**

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

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

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

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

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

Related Links

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

Refine Attributes Dialog

The *Refine Attributes* dialog is used to set the attributes for a refine point represented by a feature point in a 2D Mesh model coverage. The dialog is reached by right-clicking on a point in the Map module and selecting the **Refine Attributes** command. This option is only available if the active coverage is a 2D Mesh model coverage type (i.e. mesh generator, ADCIRC, FESWMS, etc.).

Attributes that can be specified for each refine point include:

- *Refine point* – When checked on, assigns the point as a refine point. A refine point is a feature point that is created inside the boundary of a polygon and assigned a size value. When the finite element mesh is created, a corner node will be created at the location of the refine point and all element edges that touch the node will be the exact length specified by the refine point size value.
  - *Element size* – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.

- *Create mesh node at this location* – A corner node will be created at the location of this point when the finite element mesh is generated.

Model specific options may also be available in this dialog. See the model documentation for information about these options.

Related Topics

- Feature Objects Menu
- Mesh Generation

2D Mesh Options Dialog

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

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

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

Related Topics

- Mesh Generation
- Map Feature Objects Menu

2D Mesh Polygon Properties

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

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

The following polygon attributes can be set:

- *Mesh Type* – Specify the mesh generation algorithm. The following options are available:
  - None
  - Patch
  - Paving
  - Scalar Paving Density
  - Existing Nodes
- *Bathymetry Type* – Specify the bathymetry source for assignment to the mesh. The following options are available:

  - Constant – Assigns a constant elevation value to all nodes in the mesh.
  - Scatter Set – Interpolates elevation values from the specified scatter set .
  - Existing Mesh – Interpolates elevation values from an existing 2D mesh.
  - Raster – Interpolates elevation values from an existing raster image.
- *Material* – Material type to assign to elements created within the polygon.
- *Arc Options* – Used to modify the feature vertices along the arc. The arc must be selected in the polygon preview window. The following options are available:
  - Use Original Vertices
  - Distribute Vertices – Change the number of vertices along the arc. A bias can be specified rather than distributing the vertices with a constant spacing.
- *Node Options* – When the mesh type is set to Patch , the node options are used to define the 3 or 4 sides for the patch mesh generation. If a feature point is selected in the polygon preview window, the following options are available:
  - Split – The mesh generation algorithm will treat the two arcs meeting at the feature point as separate sides.
  - Merge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a single side.
  - Degenerate Edge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a "degenerate edge." When using a degenerate edge, the Patching algorithm will require only 3 sides to be defined. This option is only valid for meshes which allow triangular elements. Only one degenerate edge can be specified per feature polygon.

Mesh Preview

The preview area allows seeing and editing how the mesh will appear after the polygon has been converted to the Mesh module. This area has the following tools and options:

- **Preview Mesh** – The preview area does not update automatically. Clicking this button will display how the mesh will appear after conversion. After any options in the dialog are changed, this button must be clicked again to show the changes.

Preview Tools

The following tools are available for the *2D Mesh Polygon Properties* dialog. Changes that are made to the polygon, such as adding a vertices or moving a node, will be made to the polygon once the dialog is closed by clicking **OK** . When nodes or vertices are moved, a preview can also be seen in the main graphics window if the dialog is not covering the changed area.

- **Select Feature Node** – Allows selecting a node which can then be moved to change the shape of the polygon.
- **Select Feature Vertex** – Allows selecting a vertex which can be deleted or moved to change the shape of the polygon.
- **Create Feature Vertex –** Adds a vertex along an arc belonging to the polygon.
- **Pan** – Moves the what portion of the polygon is visible in the preview area.
- **Zoom** – Magnifies or shrinks the polygon in the preivew area.
- **Frame** – Centers and resizes the polygon to fit in the preview area.
- **Delete Selected Feature Vertices –** Removes the selected vertex from the polygon.

Related Topics

- Feature Objects
- Map Module Menus

Advancing Front Triangulation

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

Boundary Spaced Advancing Front

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

Scalar Advancing Front

SMS supports the option to control the spacing between layers of elements using a size dataset. This is the scalar advancing front method and requires selecting a spatial dataset that is everywhere positive to define the local spacing of the desired mesh. This may come from a variety of sources.

See the tutorials on mesh generation for CGWAVE and ADCirc for more information.

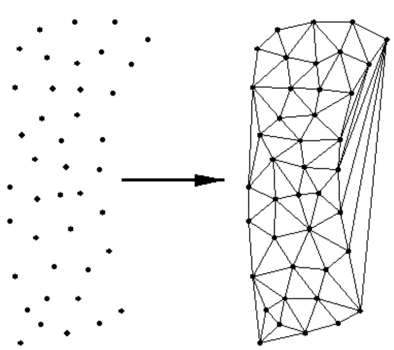
Related Topics

- Mesh Generation
- Adaptive Tesselation
- Patches

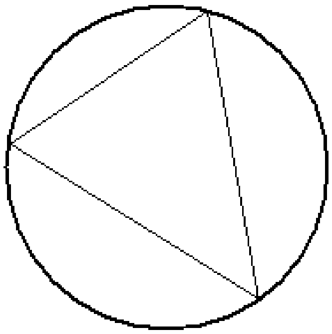
Mesh Node Triangulation

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

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



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



Optimize Triangulation

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

Related Topics

- Boundary Triangles
- 2D Mesh Elements Menu

Merge 2D Meshes

Merge Command

Two meshes can be merged to form a new mesh which will be called "Merged" (or "Merged (##)" if that name is already used) in the Project Explorer. No model data will be transferred, and the type of the new mesh will be the same as the default mesh model set in the Preferences regardless of the model type of the meshes used in the merge (to check the Preferences, go to *Edit* | **Preferences...** and check the *Defaults* tab for the Default 2D Mesh Model). The **Merge 2D Meshes** command is accessed by selecting two meshes, right-clicking, and selecting **Merge 2D Meshes** . This will bring up the *Mesh 2D Merge Options* dialog.

If the two meshes being merged overlap, SMS will report this and give the option of aborting the merge.

When merging two meshes that overlap, SMS includes all nodes from both meshes, and forces one of the previous mesh's boundaries to be honored in the merged mesh. Any disjoint nodes in the meshes will be deleted.

Mesh 2D Merge Options

The *Mesh 2D Merge Options* dialog has the following options to determine whitch method SMS will use in merging 2D meshes.

Merge method

- "Automatic" – Checks for overlapping elements in the meshes to be merged. If no overlap is found, then merging will be done the same as if "Non-overlapping" was chosen. Otherwise, the "Overlapping" merge will be performed. Checking for overlapping elements can be time consuming for large meshes.
- "Overlapping" – Safely merges meshes with overlapping elements.
- "Non-overlapping" – The fastest merge method. However, if this method is used with meshes that overlap, the resulting merged mesh will be invalid.

Base mesh

- When merging meshes, one mesh will be the principal mesh. The principal mesh is the mesh that is considered the mesh that the other mesh will be added to. In the case of an overlapping merge, the boundary conditions of the principal mesh are the ones that are preserved.

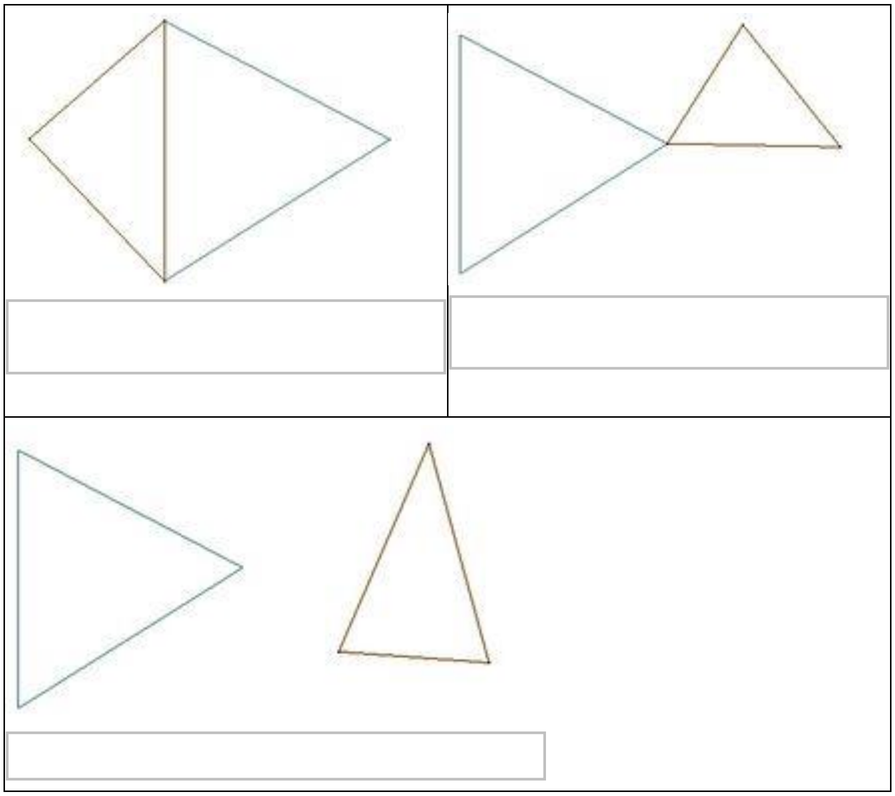
Merging with Priority

One common feature that may be desired would be to only include nodes from one of the two meshes in the overlap region. This feature is under consideration. To get this result with current functionality, the following steps may be followed:

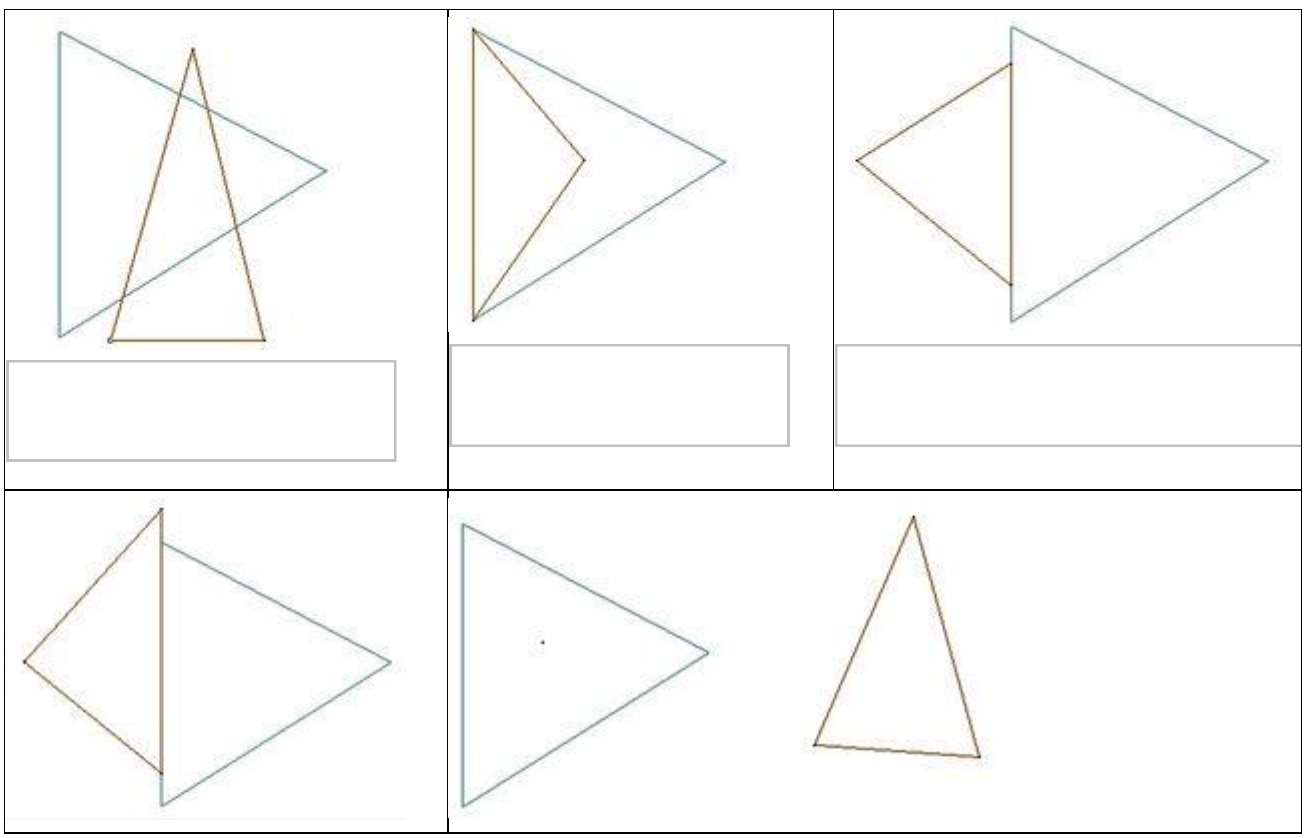
1) Convert the boundary of the mesh with higher priority (the one for which the nodes will be preserved) to a feature polygon (Use the right click menu **Convert to feature** command).
2) Build polygons for the new regions just created in the map coverage.
3) Using the *Nodes* | **Options** command in the mesh module, make sure the option to retriangulate voids is enabled.
4) Select the polygons created in the previous step.
5) In the *Edit* menu, use the **Select/Delete Data** command to select all the nodes from the low priority mesh that lie inside the high priority mesh. Delete these nodes.
6) Now merge the two meshes. The nodes from the low priority mesh that were in the overlap region have been deleted, so they do not appear in the merged mesh.

(Note: If wanting to maintain the original meshes in this process, create a copy of the low priority mesh to edit before performing the steps. This copy can be deleted after the merge process if desired.)

Examples of Meshes that are not considered overlapping



Examples of Meshes that are considered overlapping



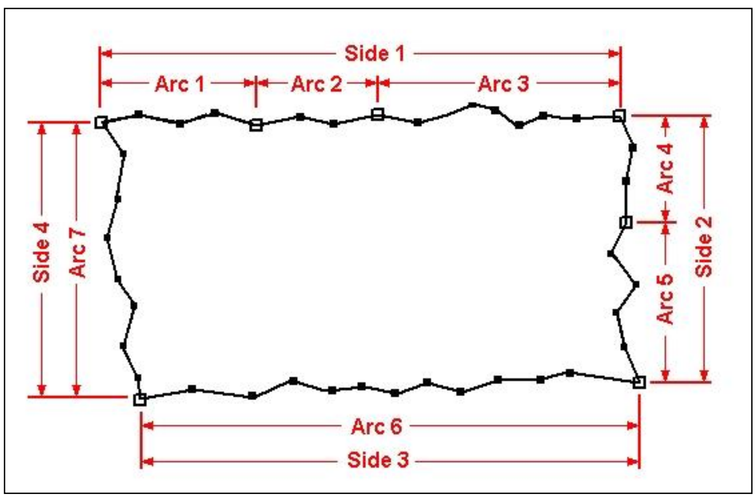
Related Topics

- [2D Mesh Module Menus]

Patch

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

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



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

- Switch to the Map Module
- Select the menu *Feature Objects* | **Build Polygons**
- Switch to the **Select Polygon** tool
- Select the polygon to set meshing attributes for
- Select the menu *Feature Objects* | **Attributes**
- Set Mesh Type to *Patch*
- Set Bathymetry Type
- Set Material Type
- Set Node Options if the polygon consists of more than four arcs
    - Switch to the **Select Feature Point** tool in the dialog
    - Select the node to "merge"

  - In the *Node Options* combo box, change the selection to "merge"
  - The two arcs meeting at the "merge" node will now be treated as a single arc for the mesh generation within the current polygon.
- Press **OK** to exit the dialog and save the polygon attributes

Related Topics

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

External Links

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

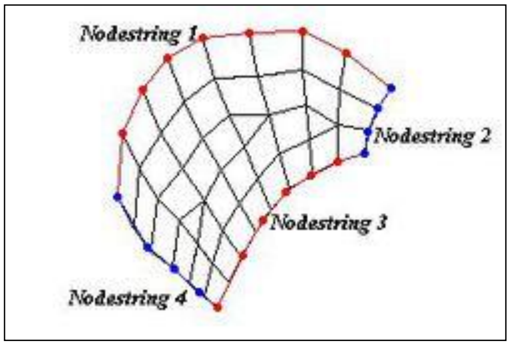
Patches

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

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

It is recommended that a patch be applied before generating a mesh using the *2D Mesh Polygon Properties* dialog.

Rectangular Patch



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

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

- *Use original nodes* – This option causes the original nodes from the nodestring to be used as corner nodes of elements along the boundary.
- *Distribute nodes* – This option distributes the specified number of nodes as corner nodes of elements along the boundary. If elements already exist on the boundary, then this option is unavailable.

- *Bias* – This is used with the *Distribute nodes* option. It causes the spacing of nodes along the nodestring to be weighted more to one of the corners.

After the spacing on each side is defined, click the **Preview** button to see how the patch will look. If changes are desired, they can be made. When the patch looks good, click the **OK** button to accept it. The patch can be canceled by clicking the **Cancel** button. Be careful to use the preview button because **THERE IS NO UNDO FOR THIS OPERATION** .

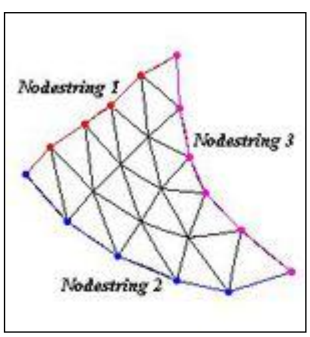
The elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

Rectagualar Patch Hints

The following are some hints when using rectangular patches:

- The curvature of the patch can change somewhat, but it should not switch directions. If it does, then the patch should be split at the inflection point of the curve.
- Although opposite sides in the rectangular patch are not required to have the same number of nodes, the best patches occur when this is close. In the example shown above, the two ends have the same number of nodes and the two sides only differ by three nodes.

Triangular Patch



Elements can be made to fill a triangular area by choosing the **Triangular Patch** command from the *Elements* menu. To define a triangular patch, three nodestrings must be selected. The nodestrings must connect at the ends.

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the nodestrings as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. The following options are available for each edge of the triangular patch:

- *Use original nodes* – This option causes the original nodes from the nodestring to be used as corner nodes of elements along the boundary.

- *Distribute nodes* – This option distributes the specified number of nodes as corner nodes of elements along the boundary. If elements already exist on the boundary, then this option is unavailable.
- *Bias* – This is used with the *Distribute nodes* option. It causes the spacing of nodes along the nodestring to be weighted more to one of the corners.

All three sides of a triangular patch must have the same number of nodes. After the spacing on each side is defined, click the **Preview** button to see how the patch will look. If changes are desired, they can be made. When the patch looks good, click the **OK** button to accept it. The patch can be canceled by clicking the **CANCEL** button.

Be careful to use the preview button because **THERE IS NO UNDO FOR THIS OPERATION** .

The elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

Triangular Patch Hints

All three sides of a triangular patch must have the same number of nodes.

Errors

When the patch is previewed in the *Polygon Attributes* dialog, the elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

If a polygon cannot be patched, a help string under the preview window in the *Polygon Attributes* dialog explains what needs to be changed.

Related Topics

- Mesh Generation
- 2D Mesh Elements Menu
- Adaptive Tesselation
- Adaptive Front Triangulation

Paving

The paving method uses an advancing front technique to fill the polygon with elements. Based on the vertex distribution on the boundaries, equilateral triangles are created on the interior to define a smaller interior polygon. Overlapping regions are removed and the process is repeated until the region is filled. Interior nodal locations are relaxed to create better quality elements.

Both the Paving method and the Scalar Paving Density can be selected in the *2D Mesh Polygon Properties* dialog. The dialog is accessed by right-clicking on the a polygon and selecting the **Attributes** command.

Scalar Paving Density

Scalar paving density utilizes the same approach as paving with the added component of a size dataset. A size dataset defines the desired spacing of nodes in a spatial fashion. A scattered dataset provides the geometric basis for the size dataset, and a dataset on the scatter set provides the values for the size dataset. SMS redistributes the vertices on the boundaries of the polygon to match the underlying size dataset.

Related Topics

- Patch Meshing Algorithm
- Mesh Generation

Adaptive Tesselation

Adaptive tessellation is a mesh generation technique used to fill the interior of a polygon. The method is based on overlaying a quad tree on the polygon, and recursively splitting the quads until the size approaches the desired spacing. SMS derives the desired spacing based on either the spacing of the original polygon, or based on a user specified spatially varying scalar dataset (for a scattered dataset). A polygon is automatically processed with adaptive tessellation and is filled with the **Map to 2D Mesh** command.

The adaptive tessellation technique is robust and relatively quick, however, it often results in discrete increments in resolution as the overlying quad tree grid transitions from one resolution level to another. For this reason, the Advancing Front Triangulation method is preferred.

Boundary Spaced Adaptive Tessellation

Adaptive tessellation uses the existing spacing on the polygons to determine the element sizes on the interior. Any interior arcs and refine points are forced into the new mesh. If the input polygon has varying node densities along its perimeter, SMS attempts to create a smooth element size transition between these areas of differing densities. Altering the size bias indicates whether SMS should favor the creation of large or small elements. Decreasing the bias will result in smaller elements; increasing the bias will result in larger elements. In either case, the elements in the interior of the mesh will honor the arc edges and the element sizes specified at nodes. The bias simply controls the element sizes in the transition region.

Scalar Adaptive Tessellation

SMS supports the option to control the local target size of elements using a size dataset. This is the scalar adaptive tessellation method and requires selecting a spatial dataset that is everywhere positive to define the local spacing of the desired mesh. This may come from a variety of sources.

See the tutorials on mesh generation for CGWAVE and ADCIRC for more information.

Related Topics

- Mesh Generation
- Advancing Front Triangulation
- Patches

Size Function

A size function is a multiple that guides the size of elements to be created in SMS.

A size function determines the element size based off of a dataset that will be created by SMS. Each point is assigned a size value. This size value is the approximate size of the elements to be created in the region where the point is located. The mesh will be denser where the size values are smaller.

Using the *Data Calculator* allows created size function datasets in SMS. The size function dataset can then be used to redistribute vertices along an arc or used as the bathymetry for polygons.

Size functions can be based off of different criteria. For example, they may be based on either depth, slope, or curvature of the model.

Size Function Based on Depth

Many coastal models utilize a size function based on depth. As the depth gets shallower, the elements should get smaller. The model will become finer near areas of interest and coarser at deep water areas that are less significant.

A size function based on depth uses the following equation:

$$\left(\left(\frac{positive\ depth\ -\ minimum\ depth}{maximum\ depth\ -\ minimum\ depth}\right) * (maximum\ size - minimum\ size) + minimum\ size\right)$$

Size Function Based on Slope

Size functions based on slope are helpful when analyzing slope data because as the rate of change of the gradient increases, the smaller the mesh element becomes. Size functions based on slope are mostly applied to riverine models.

A size function based on slope uses the following equation:

$$maximum\ size - \left(\frac{slope\ -\ minimum\ slope}{maximum\ slope\ -\ minimum\ slope}\right) * (maximum\ size - minimum\ size)$$

Related Topics

- Mesh Generation
- Arc Size Function

3.6.b. Interface Components

3.6.b.1. Mesh Module Display Options

Mesh Module Display Options

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

- *Nodes* – A circle is filled around each node. It's possibly to specify the radius and color of these circles. The **Options** button is used to set the display of nodal boundary condition data. The dialog that opens when this button is clicked depends on the current numerical model.
- *Nodal BC* – Some models support nodal boundary conditions. For those that do, each type of boundary condition can be displayed by highlighting the node with a symbol associated with that boundary condition. The **Options** button next to the *Nodal BC* entry of the display options allows selecting/modifying the symbols associated with each boundary condition.
- *Elements* – Element edges.
- *Functional Surface* – Show surfaces representing one of the functional datasets associated with a mesh, grid or TIN.
- *Contours* – The mesh contours are drawn for the active scalar dataset. Use the contours tab to change contour options.
- *Vectors* – The mesh vectors are drawn for the active vector dataset. Use the vectors tab to change vector options.
- *Nodestrings* – The color in which a nodestring is drawn depends upon its type. Unassigned nodestrings are drawn in the color/thickness/style shown at the left of the toggle box. For the display of boundary condition nodestrings, click the Options button. The dialog that opens when this button is clicked depends on the current numerical model.
- *Mesh Quality* – The mesh quality shows potential problems with the finite element mesh layout. An element is highlighted in a color corresponding to the criterion which it violates. The **Options** button opens the *Mesh Quality Options* dialog to specify display options for the mesh quality criteria.
- *Mesh Boundary* – A line is drawn around the perimeter of the mesh.
- *Inactive Mesh* – Determines the color of all inactive meshes when using multiple meshes.

- *Materials* – Elements can be filled with the color and pattern which define their materials. Materials and their display properties can be edited by choosing *Edit* | **Materials** from the menu.
- *Material Boundary* – The boundary between zones of elements with a common material type is drawn using specified line attributes.
- *Material Numbers* – The material id number can be displayed in the center of each element. The font and color can be selected.
- *Node Numbers* – The node id number can be displayed next to each node. The font and color can be selected.
- *Element Numbers* – The id of each element.
- *Nodal Elevations* – Displays the z elevation at each node.
- *Wet/Dry Boundary* – After a simulation has been opened, the interface between wet and dry nodes can be displayed.

Model specific options

Each model may include model specific display options. These appear at the bottom of the *display options* dialog and include such things as 1D contour options for RMA2 and tidal ellipses for ADCIRC .

Related Topics

- Mesh Module
- Display Options

Mesh Quality

In the *Mesh* tab of the *Display Options* dialog. selecting the **Options...** button next to the *Mesh Quality* toggle will bring up the *Element Quality Checks* dialog. Several rules of mesh element construction, if adhered to, will help in creation of a well-behaved finite element network. Violations of the following mesh quality checks should be avoided. Violations of these mesh quality checks can be displayed in SMS (see Mesh Display Options):

- *Minimum / Maximum interior angle* – For triangular elements, if the angle is between 10 and 150 degrees, computation problems will usually be avoided. Care must also be taken when curved edges are defined (non-linear midside nodes) to prevent overlap of element sides.
- *Concave quadrilaterals* – For quadrilateral elements, if the angle is between 30 and 150 degrees, computation problems will usually be avoided. Care must also be taken when curved edges are defined (non-linear midside nodes) to prevent overlap of element sides.
- *Maximum slope* – Rapid changes in slope can cause computational instabilities.
- *Element area change* – Nodes need to be more plentiful and elements smaller in areas where the solution variables (u,v, and h) change rapidly. Such areas may be located near channel or floodplain constrictions, in channel bends, or at sudden changes in bed slope. The network should be dense in the critical areas of interest. The density of a network can vary through the solution domain. Areas that are of little interest and have stable flow characteristics should not be as dense as critical areas. The size of elements needs to change gradually when moving from an area described by small elements to an area modeled with large elements, or vice versa. A rule of thumb is to keep the areas of neighboring elements within a factor of two, meaning an element is twice as big or half as big as its adjacent elements.
- *Connecting elements* – Avoid creating "pinwheels" by limiting the number of elements connecting at a node to fewer than eight.
- *Ambiguous gradient* – All triangular elements are planar by their definition. However, quadrilateral elements may vary significantly from a plane. It is a good idea to construct elements as close to a plane as possible. This precludes the existence of elements whose slope, or direction of drainage is ambiguous.
- *Display Legend* – Checking this box will display the legend in the Graphics Window.

- **Options...** – Brings up the *Legend Options* dialog.

Related Topics

- Mesh Display Options
- Mesh Module
- Display Options

3.6.b.2. 2D Mesh Module Tools

2D Mesh Module Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Mesh Module is active. Tools specific to a model interface are described with the corresponding model. Only one tool is active at any given time. The action that takes place when clicking in the Graphics Window depends on the current tool. The following sections describe the tools in the 2D Grid tool palette.

Create Mesh Nodes

The **Create Mesh Nodes** tool is used to manually create a node using the mouse. A node will be created at the location where the mouse button is clicked inside the Graphics Window. If the node is created inside the triangulated area of a current mesh, and the Insert nodes into triangulated mesh option is turned on, then the new node will be added as part of the mesh. If the new node is not added as part of the current mesh, then the z-value assigned depends on the Nodal z-value option.

Select Mesh Nodes

The **Select Mesh Nodes** tool is used to select nodes. A single node is selected by clicking on it. A second node can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple nodes can be selected at once by dragging a box around them. A selected node can be deselected by holding the *SHIFT* key as it is clicked.

If the nodes are not locked (see the menu command Nodes | Locked), then a single node can be clicked and dragged to a new location. As the node is being dragged, its new location is shown in the *Edit Window* . If a single node selected, the *X, Y, and Z Coordinate* fields in the *Edit Window* become available to set the node location exactly. If multiple nodes are selected, the *Z Coordinate* field in the *Edit Window* becomes available. The value shown is the average elevation value of all selected nodes. If this value is changed, the new value will be assigned to all selected nodes.

With one node selected, the *Edit Window* shows the node id number and the number of elements to which it is attached. With two nodes selected, the *Edit Window* shows both node id numbers and the distance between the nodes. With multiple nodes selected, the *Edit Window* shows the number of selected nodes.

Right-Click Menu

See the 2D Mesh Nodes Menu and 2D Mesh Elements Menu articles.

Create Nodestrings

The **Create Nodestrings** tool is used to create node string. Nodestrings are used for operations such as assigning boundary conditions, forcing breaklines into the mesh, and renumbering the mesh. To create a nodestring:

1) Click on a node. The node will be highlighted in red and a prompt will be shown in the *Help Window* .
2) Click on any node to add it to the nodestring. The selected node is also highlighted in red and a solid red line is drawn between the two nodes. Continue adding nodes to the nodestring in this manner.
    1) Note: For most operations, nodes in the nodestring should be adjacent, but this is not required. A breakline, for example, will usually be made of nodes which are not adjacent.
    2) Press the *BACKSPACE* key to backup one node. Press the *ESC* key to abort the nodestring creation.
    3) Double-click a node or press the *ENTER* key to end the nodestring creation.

3) The *SHIFT* and *CTRL* keys assist in creating large nodestrings which are made up of adjacent nodes. These can be used after at least one node has been selected and function as follows:
    1) *SHIFT* . Holding down the *SHIFT* key and selecting another node will add to the nodestring all nodes between the two. The path chosen is the shortest distance between the two nodes. This is useful for creating continuity strings which run along a cross section of the mesh.
    2) *CTRL* . Holding down the *CTRL* key and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going counter clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.
    3) *CTRL* + *SHIFT* . Holding down both the *CTRL* and *SHIFT* keys and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.

Select Nodestrings

The **Select Nodestrings** tool is used to select nodestrings. When this tool is chosen, a small icon appears near the center of each nodestring. A nodestring is selected by clicking inside this icon. A second nodestring can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple nodestrings can be selected by dragging a box around their icons. A selected nodestring can be deselected by holding the *SHIFT* key as its icon is clicked.

When nodestrings are selected, the Z Coordinate field in the *Edit Window* becomes available. The value shown is the average elevation value of all nodes in the selected nodestrings. If this value is changed, the new value will be assigned to all nodes in the selected nodestrings.

With one nodestring selected, the *Edit Window* shows the number of nodes in the nodestring, its type, and its length. With multiple nodestrings selected, the *Edit Window* shows the number of selected nodestrings and their total length.

Right-Click Menu

See the [2D Mesh Nodestrings Menu] article.

Create Elements

Most elements in SMS will be created using automatic mesh generation techniques. At times, however, it is necessary to manually create a single element or a small group of elements.

Although SMS supports various types of elements, only those element types supported by the current numerical model will be available in the tool palette. Some of these element types are linear while others are quadratic. A linear element has only corner nodes, while a quadratic element has midside nodes between the corner nodes.

The following linear elements are supported:

- **2-node lines**
- **3-node triangles**
- **4-node quadrilaterals**

The following quadratic elements are supported:

- **3-node lines**

- **6-node triangles**
- **8-node quadrilaterals**
- **9-node quadrilaterals**

Linear and quadratic elements cannot coexist in a single mesh. If linear elements exist in a mesh, then the quadratic element creation tools are dimmed out. Similarly, if quadratic elements exist in a mesh, then the linear element creation tools are dimmed out. To create a single linear or quadratic element:

1) Select the tool which corresponds with the type of element to be created.
2) Click on the corner nodes which will make the element, one-by-one. Do not click midside nodes. As each node is clicked, it becomes highlighted in red.
3) Alternatively, a box can be dragged around the corner nodes which will make the element. A beep will sound if the box does not surround the exact number of corner nodes required by the selected tool.

The mid-side nodes in quadratic elements are created automatically, as is the center node of a nine-node quadrilateral. Before a new element is created, SMS performs the following quality checks:

- The new element cannot overlap other elements.
- A quadrilateral element cannot twist or overlap itself.
- A quadrilateral element cannot be concave.

If any of the these fails, the new element is not created.

Select Elements

The **Select Elements** tool is used to select elements. A single element is selected by clicking inside it. A second element can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple elements can be selected at once by dragging a box around them. Holding the *CTRL* key and dragging the mouse selects any elements through which the line is drawn. A selected element can be deselected by holding the *SHIFT* key as it is clicked.

When elements are selected, the *Z Coordinate* field in the Edit Window becomes available. The value shown is the average elevation value of all nodes in the selected elements. If this value is changed, the new value will be assigned to all nodes attached to the selected elements. Caution must be used when changing node elevations in this manner. Do not create large flat areas where surrounding elements may become dry because this can cause ponds to form when the finite element analysis is performed.

With one element selected, the Edit Window shows the element id number, its type, and its area. With multiple elements selected, the Edit Window shows the number of selected elements and their combined area.

Right-Click Menu

See the 2D Mesh Elements Menu article.

Swap Edges

The **Swap Edges** tool is used to manually swap the edges of two adjacent triangles. This is useful in such cases as preserving a geometrical feature in the mesh or avoiding an artificial dam in a channel.

Two adjacent triangles form a quadrilateral with an element edge down one diagonal. When the diagonal is clicked, it gets swapped to the other diagonal, as long as the quadrilateral is not concave.

Merge/Split Elements

The **Merge/Split Elements** tool is used to either merge two triangles into a quadrilateral or split a quadrilateral into two triangles. This is a useful tool to use when trying to avoid certain mesh drying problems.

To split a quadrilateral element, click inside it. An element edge appears on the diagonal which will make the triangles uphold the Delaney criteria. To merge two adjacent triangles into a quadrilateral, click the common edge. A quadrilateral will form as long as it is not concave.

Label Contours

The **Label Contours** tool is used to manually create a contour label using the mouse. To add a label, click on the point where the label should be created. The label will remain on the screen until either it is manually removed or the automatic contour label options are changed. To manually remove a contour label, hold the *SHIFT* key and clicking on it. There are also available automatic contour label options.

Related Topics

- Editing 2D Meshes
- Mesh Module

Editing 2D Meshes

2D Mesh nodes can be inserted, deleted , or moved .

2D Mesh elements can be edited in the following ways:

- Elements can be converted between linear and quadratic.
- The type of element can be changed from a 3 node element to a 4 node element by merging triangles .
- A 4 node element can be converted to a 3 node element by splitting the 4 node element .
- Elements can be refined automatically.
- The material assigned to an element can be changed.
- Poorly shaped boundary triangles can be automatically selected for deletion.
- Breaklines can be inserted into the mesh

Deleting Nodes

A set of selected nodes can be deleted by hitting the *DELETE* key or selecting the **Delete** command from the *Edit* menu. Elements attached to the nodes are also deleted.

If the *Confirm Deletions* option in the *Edit* menu is active, SMS will prompt to confirm each deletion. This feature is helpful in preventing accidental deletions. The *Confirm Deletions* item is toggled by selecting it from the menu.

Options in the *Node Options* dialog will affect how nodes are deleted.

Editing Node Coordinates

The coordinates of a 2D Mesh node can be edited by selecting the mesh node and entering the new coordinates in the edit boxes in the *Edit Window* . It is also possible to drag an existing node to a new location by clicking on the node and moving the mouse with the button held down until the node is in the desired position.

If the snap to grid option in the *Drawing Grid Options* dialog is set, the node will move in increments corresponding to the drawing grid. If the node being dragged is connected to one or more elements, SMS will not allow the node to be dragged to a position where one of the surrounding elements would become ill-formed.

Since it is possible to accidentally drag points, nodes can be "locked" to prevent them from being dragged by selecting the **Lock All Nodes** item from the *Nodes* menu. The nodes can be unlocked by unselecting **Lock All Nodes** from the *Nodes* menu.

Merging Triangles

The triangulate operation creates a mesh composed entirely of triangles. In some cases it is desirable to have the mesh composed primarily of quadrilateral elements. Quadrilateral elements result in less elements which leads to faster solutions, and quadrilateral elements are often more stable numerically. To address this need, two options are provided for converting triangular elements to quadrilateral elements:

The Merge Triangles Command

The **Merge Triangles** command in the *Elements* menu can be used to automatically merge pairs of adjacent triangular elements into quadrilateral elements. Upon selecting the **Merge Triangles** command, a prompt appears to input a minimum interior angle. This angle should be between 0 and 90 degrees. If no elements are selected, all of the triangular elements in the mesh are then processed. If some elements have been selected, only the selected elements are processed.

The conversion process works as follows:

1) The set of elements to be processed is traversed one element at a time. Each triangular element that is found is compared with each of its three adjacent elements. If the adjacent element is a triangle, the trapezoid formed by the triangle and the adjacent triangle is checked.
2) Each of the four interior angles of the trapezoid is computed and compared to a minimum interior angle. If all of the angles are greater than the user specified minimum interior angle, then the two triangles are merged into a single quadrilateral element.

This process is repeated for all of the elements. The merging scheme will not always result in a mesh composed entirely of quadrilateral elements. Some triangular elements are often necessary in highly irregular meshes to provide transitions from one region to the next.

The Merge/Split Tool

The other option for merging triangles involves the use of the **Merge/Split** tool in the Tool Palette. This tool can be used to manually merge triangles one pair at a time rather than using the automatic scheme described above.

The manual method is also useful to edit or override the results of the automatic merging scheme in selected areas. The Merge/Split tool can also be used to undo a merge. A quadrilateral element can be split into two triangles by clicking anywhere in the interior of the element. This tool is useful if a pair of triangles is inadvertently merged.

Splitting Quadrilaterals

Occasionally it is necessary to split quadrilateral elements into triangular elements. For example, in order for new nodes to be automatically inserted into a mesh, the elements in the region where the node is inserted must be triangular. Also, in order to process a breakline, the elements in the region of the breakline must be triangular. In such situations, it may be necessary to split a group of quadrilateral elements into triangular elements. Two options are provided for splitting quadrilateral elements:

The Split Quads Command

The **Split Quads** command in the *Mesh* menu can be used to split a group of quadrilateral elements into triangular elements. If no elements are selected, all of the quadrilateral elements in the mesh are split. If some elements have been selected, only the selected quadrilateral elements are split.

The Merge/Split Tool

The other option for splitting quadrilateral elements involves the use of the **Merge/Split** tool in the Tool Palette. If the **Merge/Split** tool is selected, clicking anywhere in the interior of a quadrilateral element with the mouse cursor will cause the element to be split into two triangles. The shortest diagonal through the quadrilateral is chosen as the common edge of the two new triangular elements.

Subset Edit Mode

When working with large meshes, even simple operations can take a long time. Therefore, it can be useful to work on only a portion of the mesh. This is referred to as *Subset Edit Mode* .

Entering Mesh Subset Edit Mode

To enter subset edit mode:

1) Select elements that cover the area to be edited

2) Enter subset edit mode in one of the two following ways
   1) Right-click on the selected elements and select *Edit Subset*
   2) Right-click on the mesh tree item and select *Edit Subset*

The icon in the tree menu for the mesh being edited is updated to indicate that a subset of this mesh is being edited.

Caution: Datasets other than the depth/elevation dataset are deleted upon entering mesh subset edit.

Prohibited Actions While in Subset Edit Mode

- Making changes on the border of the mesh subset
- Making changes outside of the mesh subset
- Reading in meshes from file
- Creating meshes from feature map or scatter data
- Running numerical models
- Changing attributes such as boundary conditions on nodes, nodestrings, and elements
- Renumbering node/nodestring/element ids

Exiting Mesh Subset Edit Mode

As noted above, several operations are not allowed while in subset edit mode. It is intended that this mode only be used to facilitate editing sections of the mesh. SMS includes to commands to exit subset edit mode when the desired edits are completed, or to revert and undo any local edits.

Commit Mesh

Merge the changes made to the subset of the mesh with the rest of the mesh by selecting **Commit mesh** from the right-click menu of either the mesh tree item or the **Select Element** tool. **Commit mesh** will exit subset edit mode.

Revert Mesh

Revert to how the mesh was upon entering mesh subset edit mode by selecting **Revert mesh** from the right-click menu of either the mesh tree item or the **select element** tool. **Revert mesh** will exit subset edit mode.

Related Topics

- 2D Mesh Module Tools

2D Mesh Module Tools Right-Click Menus

The following tools are contained in the Dynamic Tools portion of the tool palette when the Mesh Module is active. These are tools with corresponding right-click menus.

Select Mesh Nodes

Right-clicking on a mesh node while using the **Select Mesh Nodes** will bring up the following menu options:

- **Delete** – Delete selected mesh Nodes.
- **Assign BC** – Assign a boundary condition to selected mesh nodes.
- **Transform** – Transform the selected nodes either by scaling, translation, or rotation.
- **Triangulate** – Triangulate the selected nodes to form triangle elements.

Select Nodestrings

Right-clicking on a selected nodestring while using the **Select Nodestring** tool will bring up the following options:

- **Delete Selected** – Delete selected nodestrings.
- **Assign BC** – Assign a boundary condition to selected nodestrings.

- **Reverse Direction** – Reverse direction of selected nodestrings.
- **Force Breaklines** – Force element edges to follow the selected nodestrings.
- **Renumber Nodes** – Renumber the nodes starting with an ID of 1.
- **Renumber Nodestrings** – renumber the nodestrings starting with an ID of 1.
- **Smooth** – Smooth the mesh boundary along the path of a selected nodestring by moving midside nodes. Only used for quadratic elements.

Select Elements

Right-clicking on a selected element while using the **Select Elements** tool will bring up the following options:

- **Delete** – Delete selected element.
- **Refine** – Split all selected elements into four elements.
- **Relax** – Moves all the nodes on the interior of the selected elements to the centroid of their contributing area. Iterates based on option set in the options command.
- **Assign Material Type** – Assign material type to selected elements.

Related Topics

- Editing 2D Meshes
- Mesh Module

3.6.b.3. 2D Mesh Module Menus

2D Mesh Module Menus

The following menus are available in the 2D Mesh Module :

Standard Menus

Includes the *File*, *Edit*, *Display*, and other standard menus. See SMS Menus for more information.

Module Specific Menus

- Data
- Nodes
- Nodestrings
- Elements

Right-Click Menu

Right-clicking on the Mesh Data folder in the Project Explorer will bring up the *Display Options* dialog. Right-clicking on a Mesh Item in the Project Explorer will bring up the following menu options:

- **New Folder** – Creates a new folder under the *Mesh* item
- **Delete** – Deletes the active mesh.
- **Duplicate** – this command will create a duplicate of the active mesh, along with its model data. The name of the new mesh will be the same as the original mesh, but with an appended number.
- **Rename** – Allows changing the active mesh name. Names need to be unique, hence if a non-unique name is entered, a warning message will appear and the name will be reverted.

- **Merge 2D Meshes** – merges two meshes together. See the article Merge 2D Meshes .
- **Convert** – Converts mesh to Map or 2D Scatter.
- **Projection** – Allows setting the projection of the mesh.
- **Reproject** – Allows reprojecting the projection of the mesh.
- **Metadata** – Allows making annotations.
- **Create Quality Mesh Scatter Set** – This command creates a scatter set consisting of one vertex at the center of each element in the mesh. Six datasets are created for this scatter set including the six quality measures defined in the ARR quality plot . These datasets range from 0.0 to 1.0. The higher the value, the higher the quality of the element. (If SMS supported element centered datasets, these quantities could be displayed directly on the mesh.) The creation of this scatter set gives a spatial feeling for the quality of the mesh.
- **Zoom to Mesh** – Zooms to where the mesh is located within the *Graphic Window* .
- **Edit Subset** – This command is only available when elements are selected. In this case, the command appears. Selecting the command creates a subset from the selected elements and enters subset editing mode .

Model Specific Menus

- ADCIRC
- ADH
- CGWAVE
- FESWMS
- Generic Mesh Model
- TABS
  - RMA2
  - RMA4

2D Mesh Nodestrings Menu

The use of nodestrings has varied from one version of SMS to another. Historically, nodestrings served the following purposes:

1) Breakline to enforce element edges.
2) Location for boundary condition assignment.
3) Location for mesh renumbering for efficient numerical analysis.

With the development of a [[SMS:Renumber|global renumbering] option the need to check matrix efficiency from a number of starting points was eliminated. The *renumber* command that had been in the nodestring menu was moved to the nodes menu. This feature was added in SMS 11.1 . The option for local renumbering was retained as a right click command when clicking on a nodestring.

As the modeling approach moves to a simulation based approach, rather than a geomtry based approach, the use of nodestrings to assign boundary conditions is transitioning to the assignment on arcs in a boundary condition coverage.

The *Nodestrings* functions includes:

Nodestrings Menu

- **Options** – This command invokes the *Nodestring Options* dialog. This dialog is described below.
- **Force Breaklines** – This command is only available when at least one nodestring is selected. The command forces element edges to follow the selected nodestring(s). This can be accomplished by swapping element edges or by inserting nodes where the nodestring crosses an element edge. This capability is described here .

- **Smooth** – This command is only available when at least one nodestring is selected and the mesh includes quadratic elements (midside nodes). The result of the command is to move the midside nodes to create a geometrically smooth curve from one element edge to the next. This continuity is based on the edges being interpreted as quadratic curves rather than linear segments. If the angle between adjacent element edges in the nodestring is greater than 60 degrees, the break in the smoothing is left to avoid severe distortion of the quadratic edge.
- **Renumber** – This command is only available in version 11.0 and older. It is only available when one nodestring is selected. The function reorders the nodes and elements (resetting the ID values) with the nodes on the selected nodestring being the starting point and sweeping through the domain as described in renumbering .
- **Renumber Nodestrings –** This command appears in the menu starting at version 11.1. This allows to explicitly remove any gaps in the nodestring ID numbers.
- **Extract Weir Elevations... –** This command appears in the menu starting at version 11.1 for ADCIRC meshes when two nodestrings making up a weir or island barrier are selected. The command invokes the **Extract Weir Elevations** dialog that allows extracting the elevations for the weir to a coverage.
- **Merge** – This command is available if more than one nodestring is selected. It looks for nodestrings that share a single point (end to end) and merges them into a single nodestring if these conditions exist.
- **Split** – This command is available if a mesh node is selected. When the command is issued, SMS looks for nodestrings that use the selected node. When such a nodestring is found, the string is split into two nodestrings at the selected mesh node.
- **Reverse Direction** – This command reverses the direction of selected nodestrings. Selecting a nodestring causes the direction arrows to be displayed and can be used to verify the nodestring direction. This is typically only useful for extracted 2D plots.

Nodestring Right-click Commands

Many of the *Nodestring* menu commands are also available by right-clicking on a selected nodestring. In addition to the commands that are described in the previous section, the following are available as right-click commands on nodestring.

- **Delete Selected –** (Standard right click menu command)
- **Add Weir... –** This command was added for version 12.0 of SMS for ADCIRC meshes. It appears only when the selected nodestring(s) are fully internal to the mesh. It invokes the *Add Weir* dialog which specifies a width (in m) for a new weir to be inserted into the mesh along the nodestring.
- **Remove Weir...** – This command was added for version 12.0 of SMS for ADCIRC meshes. It appears only when a pair of selected nodestring(s) define an ADCIRC weir or island barrier boundary condition. It invokes the *Remove Weir* dialog which specifies a method for removing the weir. These include an option to pave over the weir, which fills the area between the nodestrings with elements, or to merge the nodestrings, which creates a new node for each pair of nodes in the weir at the midpoint between the nodes on the weir. By default the command will also renumber the mesh since the edit changes the mesh definition.
- **Assign BC...** – This command applies to mesh based simulations and invokes the model specific boundary condition dialog to allow a boundary condition to be applied at the nodestring.
- **Renumber Nodes** – This command was added when the global renumber capability was implemented in the SMS and the menu command was removed.
- **Clear Selection** – (Standard right-click menu command)
- **Invert Selection –** (Standard right-click menu command)
- **Zoom to Selection** – (Standard right-click menu command)

Nodestring Options

The nodestring options dialog, invoked by the menu command described above, includes the following.

Breakline Options

Controls how breaklines are processed.

- *Insert new nodes* – Triangles intersected by the breakline are modified by adding new nodes at necessary locations to ensure that the edges of the triangles will conform to the breakline. The elevations of the new nodes are based on a linear interpolation of the breakline segments. The locations of the new nodes are determined in such a way that the Delaunay criterion is satisfied.
- *Swap element edges* – Triangles intersected by the breakline are modified by swapping element edges to ensure that the edges of the triangles will conform to the breakline.

Renumber Options (SMS version 11.0 and earlier only)

- *Band Width* – See Front Width and Band Width for more information.
- *Front Width* – See Front Width and Band Width for more information.

Related Topics

- Mesh Module

Mesh Data Menu

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

Command	**Description**
Steering Module	Model command – launches the *steering* dialog to connect multiple model runs
Switch Current Model	Model command – changes the current numerical model associated with the mesh
Data Calculator	Dataset command – invokes the *data calculator* for the mesh
Dataset Toolbox	Opens the *Data Toolbox* containing various tools to work with datasets.
Map Elevation	Dataset command – assigns a new "elevation" or "depth" dataset to the current mesh. See below.
Zonal Classification	Dataset command – creates polygons matching user specified criterion from the current mesh and its datasets
Vector Options	Visualization command – invoke the *vector options* dialog (display options)
Contour Options	Visualization command – invoke the *contour options* dialog (display options)
Set Contour Min/Max	Sets the contour options based on the current options and the selected nodes/vertices or zoom level.
Contour Range Options	Controls if the **Set Contour Min/Max** command applies to dataset specific contour options or the general contour options (for the mesh or scatter modules). It also sets the flags for precision and fill above and below.
Film loop	Visualization command – launches the film loop generation wizard
Mesh → Scatterpoint	Data conversion command – converts the current mesh to a scatter point set. Also available as a right-click command. See Below.
Mesh → Map	Data conversion command – converts the current mesh to a map module coverage. Also available as a right-click command. See Below.

Map Elevation

SMS requires that a mesh or grid have bathymetry, or bottom elevation data, associated with the nodes or cells. By default, SMS creates a dataset named "elevation" to store the elevation values. The dataset being used to store elevations is referred to as the mapped dataset.

The *Data* menus in several modules include a command to use another functional dataset as this mapped elevation function. When this command is performed the *Select Dataset* dialog opens to allow any existing scalar dataset to be chosen. Any time step of any scalar dataset can be used as the mapped dataset and override the previous nodal elevation values. This is used mainly when interpolating new elevation data from scatter points.

Mesh to Map

The **Mesh → Map** command in the *Data* menu (mesh module) is used to convert mesh data into feature data (map module). This can be useful for creating a conceptual model from an existing numeric model. The **Mesh → Map** command converts the mesh data and places it in the active coverage. This process makes use of the *Mesh → Map* dialog which provides conversion options. If wanting the new data in a new coverage, click on the **Create New Coverage** button to create a new coverage and make it active.

Material Regions → Polygons

This option converts the materials in the current mesh into polygons in the map module. If the coverage supports materials (area property or most mesh model coverages support this), the polygons will have attributes reflecting the correct material values.

Mesh Boundaries → Polygons

This option converts the current mesh boundaries into polygons in the map module.

Mesh Contours → Arcs

This option creates an arc at a specific contour value (based upon the active dataset). It is not necessary for the value to be associated with a displayed contour line as SMS will determine where the line would be if it existed.

Mesh Nodestrings → Arcs

This option creates an arc for each nodestring in the mesh.

Mesh to Scatterpoint

The **Mesh → Scatterpoint** command in the *Mesh Data* menu is used to create scatter point data from existing mesh nodes. When this command is performed, the *Convert Mesh to Scatter Points* dialog appears to request a name for the new scatter point set.

When the scatter point set is created, it contains one scatter point for each mesh node, including midside nodes and center nodes. Any mesh datasets that have been read into SMS are copied into the new scatter point set. The scatter point data can then be used for interpolation.

Obsolete Commands

- Create Datasets – Creates specific datasets for the mesh based on user specified options. No longer available as of SMS 10.1. Has been replaced with the *Dataset Toolbox* .

Related Topics

- Mesh Module

3.6.b.3.1. 2D Mesh Elements Menu

2D Mesh Elements Menu

The *Elements* Menu includes the following commands:

Options

General command – set up the default element options.

Select Thin Triangles

General command – selects all thin triangular element that meet the definition specified in the options command.

During the process of triangulation, a mesh of triangular elements is created around existing nodes. This usually creates triangular elements outside the desired mesh boundary. Many of these exterior triangles are very skinny, and some are virtually invisible. The **Select Thin Triangles** command from the *Elements* menu finds and selects skinny triangular elements which are on the mesh boundary.

Thin triangles interior to the mesh will not be selected when this command is performed, since deletion of interior triangles would result in gaps in the mesh. After the thin triangles have been selected, they can be removed by selecting the **Delete** macro.

Find

General command – finds the element with a specified ID or location.

The **Find Element** command from the *Elements* Menu is used to locate an element either with a specific ID, or surrounding a specific location. When this command is executed the *Find Element* dialog opens.

When the *Find by ID* option is selected, then the element with the specified ID is highlighted in red. If there is no element with the specified ID, then an error message is given. Conversely, when the Find by nearest (x,y) coordinate option is selected, the element which surrounds to the specified (x, y) location is highlighted in red. With either of these methods, if the current tool is the **Select Elements** tool, then the found element becomes selected in addition to being highlighted.

Assign Material Type

General command – requires a selected element. Sets the material type of the selected elements based on the option defined in the options command.

Each element in the mesh is assigned a material type. The default material ID can be set in the *Element Options* dialog. A selected element is assigned a new material type by choosing the **Assign Material Type** command from the *Elements* menu. If the *Assign default material* option is selected in the *Element Options* dialog, then the default material is automatically assigned to the selected element. If the *Prompt for material when assigning* option is selected in the *Element Options* dialog, then the *Materials Data* dialog opens from which a material type can be chosen.

Merge Triangles

Conversion command – merges triangle pairs that meet the standard for rectangles defined in the options command. Can operate on selected elements.

Split Quadrilaterals

Conversion command – splits quadrilateral elements into two triangular elements. Can operate on selected elements.

QUAD8 ↔ QUAD9

Conversion command – converts all QUAD8 to QUAD9 elements and vice versa. Only applies to the FESWMS model. Can operate on selected elements.

Linear ↔ Quadratic

Conversion command – converts all elements from linear to quad or vice versa. Only applies to TABS and FESWMS models.

Triangulate

Generation command – triangulates the selected nodes to form triangle elements.

Optimize Triangulation

Generation command – swaps edges of triangular elements to meet the Dulanay criterion.

Refine

Generation command – splits all selected elements into four elements.

At times, there is not enough definition in a finite element mesh. The **Refine** command from the *Elements* menu splits each of the selected elements into smaller elements. After the selected elements have been refined, SMS automatically creates transitions, from the refined area of higher density to the unrefined area of lower density, using triangular elements. Refine options are set in the *Element Options* dialog.

Relax

Generation command – moves all the nodes on the interior of the selected elements to the centroid of their contributing area. Iterates based on option set in the options command.

The process of creating and editing a finite element mesh can result in poor quality elements . These elements may have poor interior angles or may violate the area change guideline for adjacent elements. The **Relax** command from the *Elements* menu can improve adjacent element areas and interior angles by moving nodes. This command moves nodes to improve the elements shape. Several options are available. Relaxation is an iterative process. The number of iterations performed and other options are specified in the *Element Options* dialog. If no elements are selected, then the relaxation is performed on all elements in the mesh.

Fix Bad Area Transitions

Generation command – removes nodes that cause bad area transitions as defined in the element quality control.

The process of creating and editing a finite element mesh can result in poor quality elements . These elements may violate the area change guideline for adjacent elements, specifically, the area of the smaller of the adjacent elements divided by the area of the large element may be less than a recommended ratio. This ratio is set in the *Mesh Quality* entry of the *2D Mesh Display Options* .

The **Fix Bad Area Transitions** command from the *Elements* menu can improve adjacent element areas by removing nodes. The adjacent elements of each node in the mesh are examined for guideline violations. If more that one are found, a calculation is made to determine if the removal of the node and the retriangulation this would cause would maintain compliance for the newly formed elements and their neighbors. If this is the case the node is removed.

Rectangular Patch

Patch command – creates elements from four selected node strings. See the article Patches for more information.

Triangular Patch

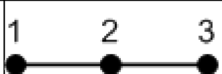
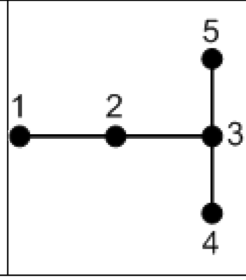
Patch command – creates elements from three selected node strings. See the article Patches for more information.

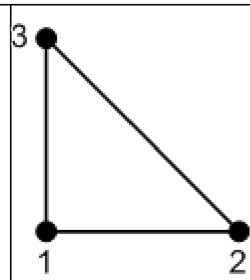
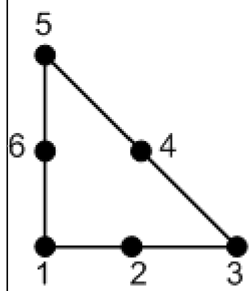
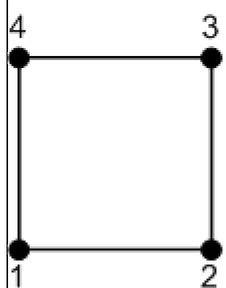
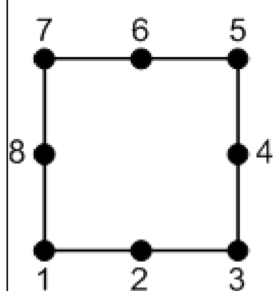
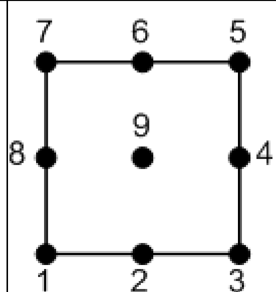
Related Topics

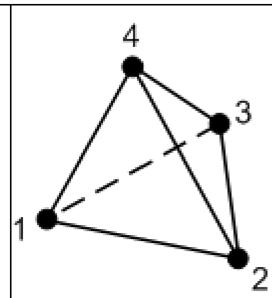
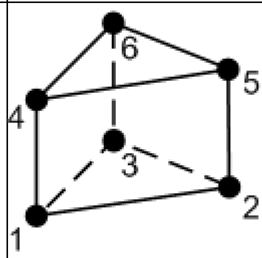
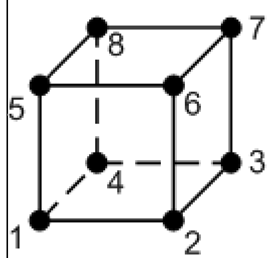
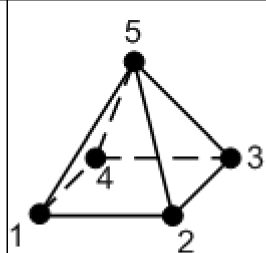
- Mesh Module
- Element Options Dialog

Element types

Element types used in XMS software.

Element Type	Image	Faces
1D linear element with 2 nodes		
1D linear element with 3 nodes		
transition element		

2D linear triangle		
2D quadratic triangle		
2D linear quadrilateral		
2D quadratic quadrilateral		
2D quadratic quadrilateral with center node		

<table>
<tr><td>3D linear tetrahedron</td><td></td><td>
<table>
<tr><td>FaceID</td><td>Node Indices</td></tr>
<tr><td>1</td><td>2,3,4</td></tr>
<tr><td>2</td><td>1,4,3</td></tr>
<tr><td>3</td><td>1,2,4</td></tr>
<tr><td>4</td><td>1,3,2</td></tr>
</table>
</td></tr>
<tr><td>3D linear prism</td><td></td><td>
<table>
<tr><td>FaceID</td><td>Node Indices</td></tr>
<tr><td>1</td><td>1,3,2</td></tr>
<tr><td>2</td><td>4,5,6</td></tr>
<tr><td>3</td><td>1,2,5,4</td></tr>
<tr><td>4</td><td>2,3,6,5</td></tr>
<tr><td>5</td><td>3,1,4,6</td></tr>
</table>
</td></tr>
<tr><td>3D linear hexahedron</td><td></td><td>
<table>
<tr><td>FaceID</td><td>Node Indices</td></tr>
<tr><td>1</td><td>1,4,3,1</td></tr>
<tr><td>2</td><td>5,6,7,8</td></tr>
<tr><td>3</td><td>1,2,6,5</td></tr>
<tr><td>4</td><td>2,3,7,6</td></tr>
<tr><td>5</td><td>3,4,8,7</td></tr>
<tr><td>6</td><td>4,1,5,8</td></tr>
</table>
</td></tr>
<tr><td>3D linear pyramid</td><td></td><td>
<table>
<tr><td>FaceID</td><td>Node Indices</td></tr>
<tr><td>1</td><td>1,4,3,2</td></tr>
<tr><td>2</td><td>1,2,5</td></tr>
<tr><td>3</td><td>2,3,5</td></tr>
<tr><td>4</td><td>3,4,5</td></tr>
<tr><td>5</td><td>4,1,5</td></tr>
</table>
</td></tr>
</table>

Back to XMS

Boundary Triangles

The perimeter of the mesh resulting from the triangulation process corresponds to the convex hull of the data points. This may result in some long thin triangles or "slivers" on the perimeter of the triangulated region.

Select Thin Triangles

There are several ways to select and delete long thin triangles.

Long thin triangles on the perimeter of the mesh can be automatically selected using the **Select Thin Triangles** item from the mesh *Elements* menu or scatter *Triangles* menu. The triangles on the outer boundary are checked first and if the aspect ratio of a triangle is less than a critical value, the triangle is selected and the triangles adjacent to the triangle are then checked. The process continues inward until none of the adjacent triangles violate the minimum aspect ratio.

The "drag line" method for selecting elements was designed specifically for this purpose. Elements can be selected with a line by selecting the **Select Elements** tool, holding down the *CTRL* key, and dragging a line through all of the elements to be selected. The selected elements can then be deleted.

SMS makes use of an aspect ratio to determine which triangles to select. Triangles with an aspect ratio below a specified value will be selected when the **Select Thin Triangles** command is used. The aspect ratio value can be changed in the *Element Options* dialog or the *Scatter Options* dialog.

Related Links

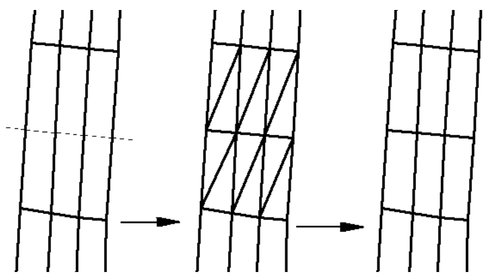
- Editing 2D Meshes
- Mesh Node Triangulation
- Triangulation

Convert Elements

Several commands are available to modify the current elements these include:

Add Breakline

The **Add Breaklines** command from the *Elements* menu can be executed when at least one nodestring has been selected. This forces element edges along the nodestring line. When this command is performed, elements are sliced along the nodestring to ensure that the edges will conform to the breakline. The elevations of any new nodes are interpolated from the original mesh. All new triangles satisfy the Delauney criterion.



A breakline example is shown. This example has some long, skinny quadrilaterals which will be split across the width. The dotted line in the left part of the figure represents the location of the breakline. When the elements are split, triangles are formed. These can be merged together using the **Split/Merge** tool , as shown in the right part of the figure.

Merge Triangles

The **Split/Merge** tool can be used to merge individual pairs of triangles. Doing this manually for large numbers of elements takes a lot of time. The **Merge Triangles** command from the *Elements* menu automatically merges a selected set of triangles simultaneously. If no elements are selected when this command is executed, all triangles in the finite element mesh will be processed.

This command uses the *Merge triangles feature angle* specified in *Element Option* dialog. In order to form quadrilateral elements with the best aspect ratios, SMS starts with a feature angle of ninety degrees and checks for any elements that can be merged. Then, a series of steps are performed, each time lowering the feature angle and checking for elements that can be merged. This ensures that the quadrilaterals which are formed are as close to rectangular as possible. In general, after the automatic merging process is complete, a limited number of triangles will still exist.

Split Quadrilaterals

The **Split Quadrilaterals** command in the *Elements* menu is used to split a set of quadrilaterals into triangles. If no elements are selected, all quadrilateral elements in the mesh will be split. The quadrilaterals are split along the shortest diagonal.

Quad8 ↔ Quad9

The **Quad8↔Quad9** item from the *Elements* menu is used to convert between eight- and nine- noded quadrilaterals. FESWMS supports nine-noded quadrilaterals. Both FESWMS and TABS support eight-noded quadrilaterals. If no elements are selected when this command is performed, all elements are converted.

Linear ↔ Quadratic

Linear elements (three node triangles and four node quadrilaterals) can be converted to quadratic elements (six node triangles and eight node quadrilaterals) and vice versa by selecting the **Linear ↔ Quadratic** item from the *Elements* menu. A finite element mesh must be made of either all linear elements or all quadratic elements. Linear elements do not have midside nodes while quadratic elements do.

Refine

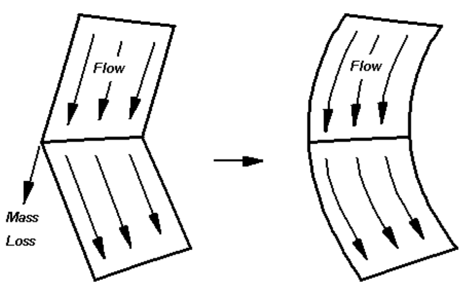
At times, there is not enough definition in a finite element mesh. The **Refine** command from the *Elements* menu splits each of the selected elements into smaller elements. After the selected elements have been refined, SMS automatically creates transitions, from the refined area of higher density to the unrefined area of lower density, using triangular elements. Refine options are set in the *Element Options* Dialog .

Relax

The process of creating and editing a finite element mesh can result in poor quality elements . These elements may have poor interior angles or may violate the area change guideline for adjacent elements. The **Relax** command from the *Elements* menu can improve adjacent element areas and interior angles by moving nodes. This command moves nodes to improve the elements shape. Several options are available. Relaxation is an iterative process. The number of iterations performed and other options are specified in the *Element Options* dialog. If no elements are selected, then the relaxation is performed on all elements in the mesh.

Smooth Nodestring

Quadratic elements have a node located at the midpoint of each edge. These nodes are generally referred to as midside nodes. The angular corners resulting from such elements are discontinuous. Such a discontinuity may result in inaccuracy in the numerical model sometimes referred to as a mass loss. Mass loss occurs because water artificially flows out of the mesh.



To minimize the abrupt change in flow direction, element edges can be curved by slightly moving the midside node. This can be done by hand using the **Select Mesh Nodes** tool with the nodes unlocked . Moving large numbers of nodes becomes tedious. However, element edges along a selected nodestring can be smoothed by SMS with the *Nodestrings* | **Smooth** command.

Normally, element edge smoothness is only a concern along the mesh boundary. However, if the analysis includes regions that become dry, interior boundaries should also be smoothed. To avoid smoothing corners that should be sharp, SMS provides a *Smooth nodestring feature angle* in the *Element Options* dialog. A corner will only be smoothed if it is less than the specified angle.

Related Topics

- 2D Mesh Elements Menu

Mesh Element Options

Certain parameters governing the creation and manipulation of nodes are set using the *Element Options* dialog, which is opened by selecting the **Options** command from the *Elements* menu available when the Mesh Module is selected. This dialog is divided into four sections.

General Options

The *General Options* section of the *Element Options* dialog specifies the following parameters for general element operations:

- *Select thin triangle aspect ratio*

  When SMS finds thin elements, only elements with an aspect ratio (element width divided by element length) less than this value are selected. This value is also used by in the model checker mesh quality checks.

- *Merge triangle feature angle*

  This angle should be between zero and ninety degrees. Any two adjacent triangles are merged into a quadrilateral if all angles in the resulting quadrilateral are greater than the merge triangles feature angle.

- *Smooth nodestring feature angle*

  When a nodestring is smoothed, the smoothing will not be applied around a corner whose angle is greater than this value. See the convert elements article for a discussion on nodestring smoothing.

- *Preserve material boundaries*

  When turned on, triangles will not be merged into quadrilaterals if they are assigned different materials types, even if they satisfy the merge triangle feature angle criteria.

Materials

The *Materials* section of the *Element Options* dialog controls how materials are assigned to elements using the following options:

- **Set Default Material**

  Brings up the *Materials Data* dialog. This defines the default material assigned to elements as they are created.

- *Assign default material*

  When turned on, the material selected as the default material is assigned to selected elements when the assign material command is issued.

- *Prompt for material when assigning*

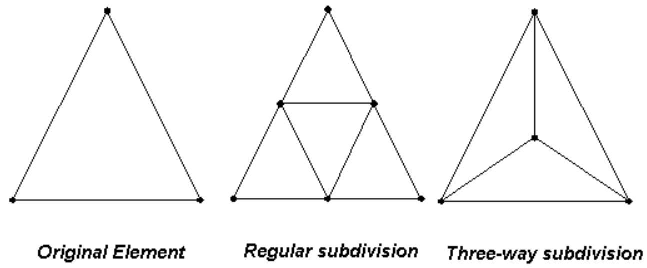
  If this option is selected, choose from a list of existing materials to assign to the selected elements when the assign material command is issued.

Refine Elements

In some cases, a mesh does not have enough elements in a particular region of the mesh to ensure stability. Rather than inserting supplemental nodes and re-creating the mesh, it is possible to refine a selected region of the mesh using the **Refine Elements** command in the *Mesh* menu. This increases the mesh density of a selected area of the mesh. If no elements are selected, the entire mesh is refined. The elevations of the new nodes are interpolated from the existing nodes.

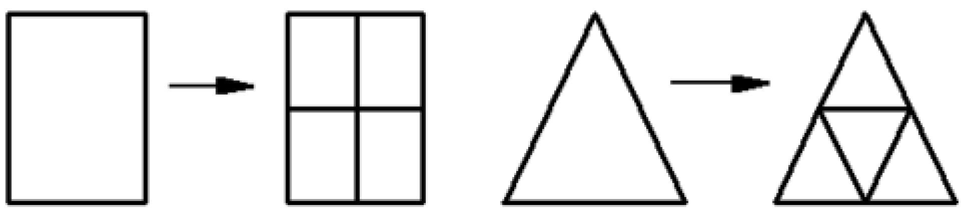
- *Three-way subdivision*

  Refines triangular element into three elements. Quadrilateral elements are not refined.



- *Regular subdivision*

  Refines triangular and quadrilateral elements into four elements.



Relax Elements

The Relax Elements section of the *Element Options* dialog controls the following relaxation parameters:

- *Number of iterations*

  This is the number of iterations to perform during the relaxation process.

- *Interpolate Z from existing mesh*

  When turned on, the nodal Z coordinate is interpolated from the old mesh so that the contours do not change. When this is turned off, the nodal Z coordinates are not changed when they are moved.

- *Lock nodes on nodestrings*

  Preserve material boundaries when relaxing. Previous versions of SMS would lock any nodes on a material or mesh boundary. Starting with version 7.0 of SMS, nodes on these boundaries will slide along the boundary unless it is part of a nodestring and this option is turned on.

- *Area relax*

  Equalize the area of elements adjacent to each node.

- *Scatter relax*

  Space the nodes according to the specified size dataset.

- *Angle relax*

  Equalize the angle of elements adjacent to each node.

Boundary Relax

- *Allow sliding on mesh boundary*

  When turned on, relaxation may modify the location of nodes on the mesh boundary.

- *Material boundaries*
  - *Allow relax*

    Allows nodes located on material boundaries to move in all directions.

  - *Preserve*

Prevents nodes located on material boundaries from moving.

- *Allow sliding*

  Allows nodes located on material boundaries to move along the material boundary.

- *Sliding angle*

  When a node is smoothed, the smoothing will not be applied around a material boundary whose angle is greater than this value.

Related Topics

- 2D Mesh Elements Menu
- Convert Elements

3.6.b.3.2. 2D Mesh Nodes Menu

2D Mesh Nodes Menu

2D Mesh Nodes are the basic building blocks of elements in finite element meshes. Nodes are also required to create nodestrings and assign boundary conditions. The following commands are available when working with 2D Mesh Nodes (under the *Nodes* menu when the Mesh Module is activated):

Interpolation Options

Opens the *Node Interpolation Options* dialog. Using the options that are set in this dialog, a set of new nodes can be interpolated between any two selected nodes.

If two nodes are selected when this dialog is invoked, the distance between the two nodes is displayed at the top of the dialog. The number of new nodes can be specified in three ways:

- *Number of intervals in string.* If this option is chosen, the number of new nodes is one less than the number of intervals specified.
- *Number of interpolated nodes.* If this option is chosen, the number of new nodes is exactly specified.

- *Total number of nodes in string.* If this option is chosen, the number of new nodes is two less than the number of nodes specified.

The *Bias factor* controls the distribution spacing of the new nodes. This factor can be any number between 0.1 and 10.0. A smaller factor will make new nodes be closer to the first selected node while a larger factor will make new nodes be closer to the second selected node. For example, a bias of 2.0 makes the first new node spaced twice as far as the last new node.

The *Linear/Arc* option controls the distribution shape of the new nodes. The Linear option causes all new nodes to be in a straight line while the *Arc* option causes all new nodes to form an arc. If the arc option is used, a Radius must also be specified. The arc will be created counter-clockwise from the first selected node to the second.

Interpolate

After the interpolation options are set up, nodes can be interpolated between any two selected nodes by choosing the Interpolate item from the *Nodes* menu. This operation may be performed multiple times with a single set of interpolation options by selecting any two nodes and invoking the command again.

The elevation of each new node depends on the *Insert nodes into triangulated mesh* option in the *Node Options* dialog (see section 1.6.8). If this option is turned on and the new node is inside the finite element mesh, then the elevation is interpolated from the mesh. If this option is turned off or the new node is not inside the finite element mesh, the elevation is interpolated from the two selected nodes.

Find Node

Opens the *Find 2D Mesh Node* dialog which can initiate a search for a specific 2D mesh node.

The *Find 2D Mesh Node* dialog can locate a node using the following methods:

- *Find by ID* &nash; When the *Find by ID* option is selected, then the node with the specified ID is highlighted with a red circle. If there is no node with the specified id, then an error message is given.
- *Find by Nearest (x,y) coordinate* – When the *Find by nearest (x,y) coordinate* option is selected, the node closest to the specified (x, y) location is highlighted with a red circle.

With either of these methods, if the current tool is the **Select Mesh Nodes** tool, then the found node becomes selected in addition to being highlighted.

Select or Delete Duplicate Nodes

Duplicate nodes are either selected or deleted, according to the option defined in the *Node Options* dialog. The menu item shows either *Select Duplicate nodes* or *Delete duplicate nodes* based on the setting. Two nodes are considered to be duplicates if they are closer together than the *Tolerance* in the *Node Options* dialog. When deleting duplicate nodes, elements attached to deleted nodes will also be removed, unless the *Merge adjacent elements when deleting* option is turned on in the *Node Options* dialog.

Select Disjoint Nodes

Disjoint nodes can be found automatically and selected by choosing the **Select Disjoint Nodes** option from the *Nodes* menu. Disjoint nodes are nodes that are not connected to any elements. Before saving a simulation, it is important to make sure there are no disjoint nodes in the mesh.

Locked

The nodes in a mesh can be dragged with the mouse cursor if they are unlocked and the **Select Mesh Nodes** tool is selected. The **Locked** item in the *Nodes* menu toggles on and off the node locked status. If nodes are locked, a check mark is shown next to the menu text. The default status is locked so that nodes are not accidentally moved.

Reduce Nodal Connectivity

Searches through the active mesh looking for wagon wheel nodes. When such a node is found, SMS reduces the nodal connectivity by inserting new node(s).

Renumber

Used to order the IDs of the nodes and elements to make numeric calculations more efficient. The goal is to make the matrix used in calculations as diagonal as possible by having related nodes numbered with indices as close as possible to each other. SMS now utilizes a Cuthill-McKee global renumbering scheme to update these indices. When a mesh is generated, it is resequenced, however, after performing hand edits the mesh should be sequenced again.

Multiple invocations of the scheme may result in slightly different sequences.

Transform

Opens *Transform* dialog which is used to move a group of selected nodes. If there are no selected nodes, the transformation will be applied to all nodes of in mesh. When this command is executed, the *Nodes Transform* dialog opens.

In this dialog, the transformation type can be chosen and then appropriate parameters can be entered. The following transformation types are available:

Scaling, translation, datum conversions, and rotations are supported.

By default, the image will be framed after the transformation takes place. However, this can be turned off by using the **Frame** image after transformation option.

Options

Parameters governing the creation and manipulation of nodes are set using the *Node Options* dialog.

Interpolate Nodal Boundary Conditions

If two non-adjacent boundary nodes have been assigned boundary conditions, and the two nodes are selected, this command interpolates the boundary conditions to each of the boundary nodes between the two.

Related Topics

- Mesh Module Tools
- Mesh Module

2D Mesh Node Options Dialog

Parameters governing the creation and manipulation of nodes are set using the *Node Options* dialog, which is opened by selecting the menu *Nodes* | **Options** .

Individual Node Options

- *Insert nodes into triangulated mesh* – When a node is created inside the mesh boundary with the **Create Mesh Nodes** tool, it can become part of the mesh. If this option is turned off, new nodes are not added to the mesh triangulation and remain disjoint. This option also applies to nodes created using the **Interpolate** command from the *Nodes* menu.
- *Retriangulate voids when deleting* – See below.
- *Node Z value* – The z-value of a node created with the *'Create Mesh Nodes* tool is based on the chosen option:
    - Interpolate z-value from mesh – The Z coordinate is determined by interpolation from the existing mesh. If this option is turned off or the node is created outside the existing mesh boundary, the default Z coordinate is assigned.
    - Assign default z-value – The Z coordinate is assigned the default value.
    - Prompt for z-value – A dialog will prompt for the Z coordinate of each node after it is created.

- Interpolate z-value from active scatter – The Z coordinate is determined by interpolation from the active scatter set. If the node is created outside the active scatter set boundary, the default Z coordinate is assigned.

Retriangulate Voids When Deleting

When deleting a node is deleted, all elements attached to the node are also deleted. The void in the mesh left by the deleted elements can be automatically filled by triangulating the surrounding nodes. If this option is turned off, then the void will remain.

If the *Retriangulate voids when deleting* option is turned on, the void created when a node and the elements surrounding the node are deleted is re-triangulated or filled in with triangles. This feature makes it possible to selectively "unrefine" a region of the mesh or reduce the density of the nodes in a region of the mesh without having to completely recreate all of the elements in the region.

When deleting selected mesh nodes, if the node count exceeds ~1000 there will be a noticeable delay if the *Retriagulate voids when deleting* box is checked.

Duplicate Node Options

- *Merge adjacent elements when deleting* – When a duplicate node is removed, the adjacent mesh elements are merged.
- *Tolerance* – Two nodes closer than this tolerance value will be considered identical for selection and deletion of one of them. Also used by the automated mesh generation algorithms of SMS as a minimum node spacing. The tolerance should be specified in feet or meters. If using a Geographic Coordinate System , the tolerance is automatically converted by SMS to meters.
- *Select/Delete duplicate nodes* – The *Nodes* | **Select/Delete Duplicate Nodes** command is based on this selection.

Related Topics

- 2D Mesh Nodes Menu
- Mesh Module

Renumber

Renumbering a mesh improves the computational efficiency (how fast a model produces a result) of a numeric mesh but should not affect the end results.

Global Renumber

To renumber a mesh select the **Renumber** command in the *Nodes* menu.

Upon execution of this command, the nodes and elements are renumbered using a global renumbering process known as the Cuthill-McKee or Inverse Cuthill-McKee scheme. Other global resequencing methods may be added in future versions. The Cuthill-McKee method searches for a global optimum, but since are often multiple options with the same efficiency level (bandwidth), invoking the command multiple times usually results in different numbering patterns. Each time a mesh is generated, SMS invokes a renumbering command. When nodes are manually added/removed from a mesh, it should be renumbered.

It is important to realize that after renumbering the finite element mesh, any previous boundary condition file or solution file may no longer be valid!

In the case of boundary conditions, SMS associates the specified conditions with the nodestrings, elements or nodes, but the model specific files must be resaved with the new numbering scheme.

In the case of solution files, the numeric engines output values associated with a node id, so the solution is associated with a specific mesh, right down to the numbering of the nodes. The solution would not map to the renumbered mesh correctly and must be regenerated. (The old solution is still valid for the old mesh, but renumbering in effect creates a new mesh.)

Front Width and Band Width

There are two measures of efficiency of a matrix. These include front width and band width. Both can be computed in multiple ways from the grid. The Cuthill-McKee scheme has its own method of computing band width and reports the band width before and after renumbering. In addition, SMS provides an estimate as to how large the front width and half band width may become when running the finite element solver. These estimates are shown in the Mesh Information dialog, which can be opened by performing the *File* | **Get Info** command while in the Mesh Module.

Background information

Due to the number of questions that are asked regarding this subject, this section will attempt to describe, in a broad sense, why renumbering is important.

The finite element solvers use an iterative, banded numerical solver to solve the governing differential equations. If the computer had to simultaneously solve the thousands of equations, much more memory would be required and the process is much less efficient (more time). Meshes with gaps in numbering could lead to errors or singular matrices resulting in no solution with many finite element solvers.

Renumbering organizes the equations in the system of equation so that they can be decomposed and efficiently solved.

Historical

In the past, SMS used a trial an error method of renumbering. It required selecting a nodestring and issue the **Renumber** command either from the *Nodestrings* menu or when right-clicking on a selected nodestring. SMS would use a sweeping algorithm progressing from the selected nodestring to reassign the node and element numbers. The "row" of elements and nodes adjacent to the string is numbered first. The elements and nodes adjacent to the first set of nodes and elements are numbered next, and so on until all of the nodes and elements have been renumbered.

Since the front proceeds from one set of elements to an adjacent set of elements, disjoint portions of the mesh were not visited in the renumbering process. Unvisited nodes and elements were numbered arbitrarily. It was then up to the user to try various starting points, comparing the computed front widths, and keep the numbering that resulted in the smallest front width.

This process did not ensure an optimal numbering, and could become tedious. With version 11.1 of SMS, the global renumbering methodology was added, making the process more efficient. When this modification was implemented, the command to renumber was moved from the *Nodestrings* menu to the *Nodes* menu. However, for convenience, the command was left in the right-click menu on selected nodestrings. SMS uses the global renumbering algorithm even when the command is issued from this right-click menu.

Related Topics

- Nodestrings Menu

Reduce Nodal Connectivity

Many finite element engines (including ADCIRC) have limits on the number of elements that may be attached to a single node. When many elements attach to a single element, the node appears as a hub with many spokes radiating from it. Thus, it is termed a wagon wheel node.

When many elements are connected to a single node, each element has a smaller interior angle. This results in more severe deformations in numerical space. Ideal triangular elements have internal angles of 60 degrees. Ideal quadrilateral elements have internal angles of 90 degrees. For triangles, this would result in six elements connected (or constructed using) a single node. For quadrilaterals, at most four elements would converge at each node.

The *node* menu of the Mesh module includes the **Reduce Nodal Connectivity** command which inserts one or more new nodes in the area of wagon wheel nodes resulting in a maximum of 7 adjacent elements.

The command may need to be applied recursively. The first time may reduce connectivity from 12 or more to 8. Then a second application reduces connectivity to 6.

Specifically, if the current connectivity is:

- 8 elements – 1 new node added
- 9 elements – 2 new nodes added
- 10 elements – 3 new nodes added
- 11 elements – 4 new nodes added
- 12 elements – 6 new nodes added.

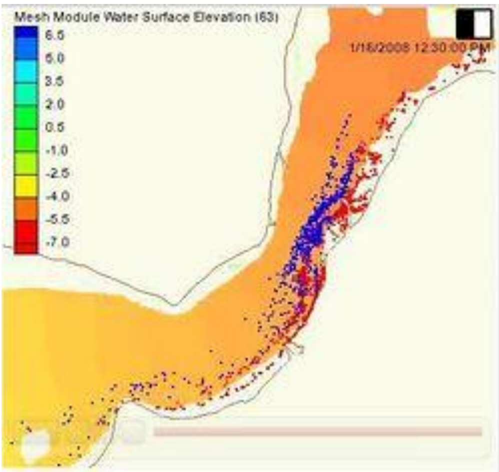
In each of these cases, the resulting elements all have connectivity of 6 elements. The patterns of insertion are illustrated below:

Related Topics

- Mesh Module Menus
- Mesh Module Tools

3.7 Particle Module

Particle Module



At a glance

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

The particle module contains tools used to work with particle data. Particles can have time varying location and scalar data. The particle module currently includes interfaces for:

- PTM – Lagrangian particle tracker designed to allow simulating particle transport processes.

The Particle module can be added to a paid edition of SMS.

Particle Module Tools

The Particle Module tools are contained in the Dynamic Tools portion of the tool palette when the Particle Module is active.

Select Particles

The **Select Particles** tool selects a single particle with a left mouse click. A group of particles can be selected by dragging a box around them. Particles may be added to the selection by holding the *SHIFT* key and selecting additional particles. The main use of selecting particles in the particle module is to query the particle properties. When a single particle is selected, the *Info Window* will show the ID and the edit window will show the location and value of the active particle dataset. If exactly two particles are selected, the *Info Window* will also show the exact distance between the two selected particles. If more than one particle is selected, the minimum, maximum, and average value of the active particle dataset will be shown.

Particle Module Menus

See Particle Module Menus for more information.

Particle Module Display Options

See the article Particle Module Display Options .

Related Topics

- SMS Modules
- Particle Tracking Model (PTM)

Particle Module Display Options

The properties of all particle tracking data that SMS displays on the screen can be controlled through the *Particle* tab of the *Display Options* dialog. This dialog is opened by selecting *Display* | **Display Options** from the menu bar, the **Display Options** macro, or the *Ctrl+D* quick keys.

The display options defined will only be applied to the active Particle Set displayed in the Project Explorer. The active Particle Set is listed at the top of this display options tab. To change to another set, close this window, left-click on the desired Particle Set in the Project Explorer, and then reenter the display options.

The entities associated with the Particle module with display options are shown below. These entities also show an **Options** button to the right. For these entities, additional display controls are available. The available particle tracking display options are:

- *Particles*

  A symbol is placed around each particle. The symbol, size, and color of these representations can be specified by using the **Options** button. The toggle controls the display of the particles. The Color based on sets whether the particles will be colored normally or dynamically. The choices include:

  - *Default* – Displayed using the color defined by the **Options** button.
  - *Active Dataset* – Displayed using the coloring of the active dataset. Available only when a Particle Set exists with a dataset.
  - *Defined Dataset* – Displayed using the coloring of the specified dataset defined using the **Select Set…** button. The name of the specified set will be displayed beside this button. Available only when a Particle Set exists with a dataset.

- *Particle tails*

  A tail is drawn out from the particle as it moves through time. The farther the particle has moved, the longer the tail may be; if the particle has remained stationary, then a tail may not appear. The tail style, width, and color can be specified by using the **Options** button. The toggle controls the display of the particles. Tail Length, in seconds, controls the amount of tail displayed. The longer (time length) the particle has a tail, the longer the tail may be; if the particle has remained stationary for the duration of the tail length, then no tail will appear. Lengths can be fractions of time steps and the default is 10 time steps. Include symbol can be turned on to display little filled circles at time step positions within the tail. The Color based on sets whether the particle tails will be colored normally or dynamically. The choices include:

  - *Default* – Displayed using the color defined by the **Options** button.
  - *Active dataset* – Displayed using the coloring of the active dataset. Available only when a Particle Set exists with a dataset.
  - *Defined dataset* – Displayed using the coloring of the specified dataset defined using the **Select Set…** button. The name of the specified set will be displayed beside this button. Available only when a Particle Set exists with a dataset.
  - *Same as above* – Displayed using the coloring of the same dataset specified from the Particles. Available only when the Particles are colored based on a defined set and the set has been selected.

- *Particle path lines*

  A path line is drawn from the original position of a particle to every position the particle inhabits thereafter. The path line will remain even after the particle has settled or has crossed the edge of the domain. Specify the path line style, width, and color using the **Options** button.

All off unchecks all three particle display options (Particles, Particle tails, and Particle path lines). This also disables the *Particle Display Filter* since nothing is selected to be displayed.

All on checks all three particle display options.

Particle Display Filter specifies the range of Particles, Particle tails, and Particle path lines to be displayed. Filtering the particle set can increase the displaying speed and improve visibility of specific particles or groups of particles. The total Number of particles within the active particle set is displayed for convenience. *Display every* displays series of particles, i.e. an input of 1 displays every particle, but an input of 7 will display particles 1, 8, 15, 22, 29… and so forth. Begin with particle denotes the first particle within the range to be displayed. The input cannot be less than 1 or more than the displayed particle set total. End with particle denotes the last particle within the range to be considered for display. If the input is not a multiple of the Display every input plus one, then the particle will not be displayed. For example, an input of 46 with Display every input of 15 will display the last particle because 3 * 15 + 1 = 46. If the input was 45, the particles 1, 16, and 31 will only be displayed (particle 45 is considered, but is filtered out). The input cannot be less than the Begin with particle input or more than the displayed particle set total. One or more of the display options must be on to enable the filter controls.

Specific Dataset Color Options selects a specific dataset and adjust its color options (similar to geometry contour options, but specific to each Particle Set dataset) by clicking on **Options** . Color options is available only when a Particle Set exists with a dataset.

Related Topics

- Display Options
- Particle Module

Particle Module Menus

The following menus are available in the Particle Module :

Standard Menus

See SMS Menus for more information.

Module Specific Menus

Beside the standard menus, the Particle Module has the

menu.

Particle Module Data Menu

The Particle Module *Data* menu commands include:

- Data Calculator
- Dataset Toolbox
- Filter Options
- Create Datasets
- Film Loop
- Compute Grid Datasets

Right-Click Menus

Right-clicking on the Particle Data folder in the Project Explorer brings up the standard right-click folder menu commands as well as the **Display Options** command to access the Particle Display Options .

Right-clicking on a particle set in the Project Explorer brings up the following commands:

- Copy Simulation Inputs
- Filter Objects
- Extract Subset

Model Specific Menus

- PTM

Extract Particle Subset

This command writes a portion of a particle set to a new a particle file. This portion may be a subset of the time steps for the particle solution, a subset of the particles, or both. To get to this command, right-click on a particle set and choose **Extract subset (times/particles)** .

The required information for this command includes:

- The file name to write the particle subset to.
- The first and last time steps that define the range of time the user wants export.
- Whether to write every time step, every other time step, every third time step, etc.

It is possible to define a subset of the particles to export by setting up filters (see PTM Particle Filters). For example, deciding to only write particles with a specific range of grain size, or particles from a specific source.

The extracted particle set will have the same datasets that exist in the original particle set.

Related Topics

- Particle Module

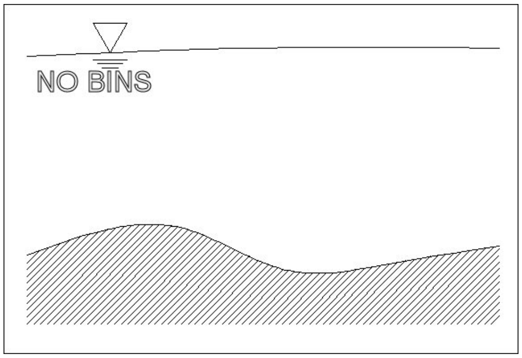
3.7.a. Particle Module Datasets

Particle Grid Dataset Bin Elevations

This article documents a feature that is under development

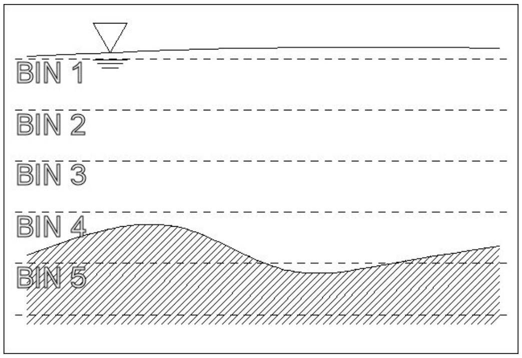
Datum Definition

The particle module compute grid dataset bin elevations dialog is accessed through the *Compute grid datasets* dialog . The bin elevations are specified according to the datum selected in the *Compute grid datasets* dialog.



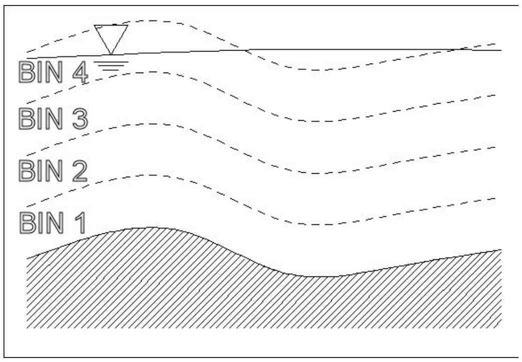
Fixed Datum

When using a fixed datum, the bins have a constant z-elevation. Be careful if the model uses a depth dataset. For example, if the water surface is at a constant value of 0.0 meters and the depth is a constant value of 10.0 meters, the bins should range from 0.0 to -10.0 meters.



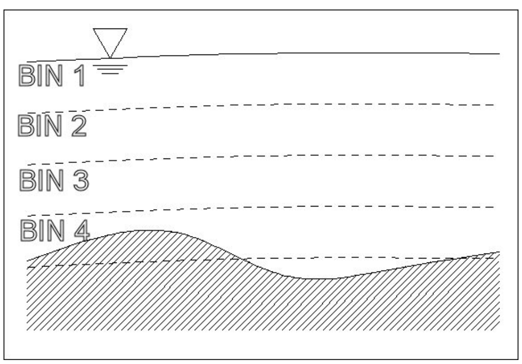
Bathymetry as Datum

When using the bathymetry as a datum, the bins are offsets from the specified bathymetry dataset. This is useful when determining the effects of concentrations on bottom dwelling species, oyster beds, etc. Offsets should always be positive values.



Water Surface Elevation as Datum

When using the bathymetry as a datum, the bins are offsets from the specified bathymetry dataset. This is useful when determining the effects of concentrations on fish migrating at a specified depth, etc. Offsets should always be positive values.



Related Topics

- Particle Module Compute Grid Datasets Dialog

Particle Module Compute Grid Datasets

The *Particle Module Compute Grid Datasets* dialog is accessed through the particle module *Data* menu . This dialog is also used to compute datasets on fence diagrams (defined in a coverage).

The datasets are added to the active cartesian grid . It should be noted that the resolution of the grid will have an impact on the numerical values computed for several of these datasets. For example, the accumulation in a cell is computed as the volume of particles which have settled in a cell divided by the cell area. If the cells are larger, this will result in a smaller accumulation. Many projects require experimenting with a few different grid resolutions.

Currently the following grid datasets can be created in the *Particle Module Compute Grid Datasets* dialog:

- Particle Count

  The number of particles in the Cartesian grid cell.

- Accumulation

  The depth of particles in the Cartesian grid cell. The volume of particles is calculated using the particle mass and density dataset for particles which are inactive (based on the state dataset) and in the cell. The volume in each cell is divided by the area of the cell to calculate an average depth in the cell. No voids ratio is included at this time, however the Data Calculator can be used to change the resulting Cartesian grid dataset.

- Rate of accumulation

  The change in accumulation (as described above) over time.

- Deposition

  The change in depth of particles in the cartesian grid cell during the focus time. The volume of particles is calculated using the particle mass and density dataset for particles which have become inactive (based on the state dataset) during the focus time. This volume is then divided by the area of the cell to calculate an average depth in the cell. No voids ratio is included at this time, however the Data Calculator can be used to change the resulting cartesian grid dataset.

- Concentration

  The concentration of particles in the Cartesian grid cell. The volume of particles is calculated using the particle mass and density dataset for particles which are active (based on the state dataset) and in the cell. This volume is then divided by the volume of the cell using the bathymetry and water surface elevation datasets. For this purpose, the bathymetry must be specified as an *elevation* dataset. This means that the values must be specified as positive upwards from the same datum the water surface dataset is measured from. For models such as CMS-Flow and ADCIRC, which require that specifying positive depths (values measured down from mean sea level), the geometry (Z) dataset must be inverted to use for this application. The depth of the water column in a cell of the resulting grid is computed as *WSE – Elevation* . This depth is combined with the cell area to compute a cell volume. SMS is capable of extracting ground elevations (bathymetry) and water surface elevations from either a finite element mesh, or a scattered dataset.

  If the data is only available on a Cartesian grid, convert this to a scatter set. Right-clicking on the grid and selecting the *Convert* | **2D Grid → 2D Scatter** , will accomplish this. The datasets on the grid will be converted to the scattered data as well.

- Exposure

  The cumulative concentration relative to time in the Cartesian grid cell.

- Dosage

  The exposure in the Cartesian grid cell during the focus time.

Related Topics

- Bin elevations dialog

Particle Module Create Datasets

The particle module create datasets dialog is accessed through the particle module data menu . Currently the following particle datasets can be created in the particle module create datasets dialog:

- *Distance traveled*

  For each time step, computes the total distance each particle has traveled since the particle was born.
- *Average velocity (since last time step)*

  For each time step, computes the average velocity of the particle since the previous time step.
- *Number of particles within vicinity*
  - *Vincinity radius*
- *Dataset name*

  A name for that particle dataset can be entered in in each field under this option.

Related Topics

- Particle Module Menus

PTM Create Grid Datasets – Fence Diagrams

The Particle Module Compute Grid Datasets page describes how to represent particle data on a rectilinear grid. The computations include things such as count, accumulation, and concentrations on the grid cells. In addition to computing these values on 2D grid cells, some of the datasets can be computed in layers creating 3D data. These datasets include concentration, exposure and dosage.

3D Fence Options

If the "create fence diagram" option is selected SMS will build a 3D mesh and datasets for each of the selected 3D datasets. SMS will also turn on the option to display fences (found in the display options dialog) and set the coverage used for fences. See 3D Fence Diagrams below for information on adjusting the display of the fence.

There will be two datasets generated for each type of 3D data computed. One dataset represents the concentration/exposure/dosage that is experienced by the cell. These values only make sense when applied to a volume. These datasets will display as a block filled value in each cell. Some people prefer viewing smooth contours rather than block filled values. The second dataset that is created represents the cell based values averaged to the nodes to provide for smooth contours. These datasets have "smoothed" in their names to distinguish them from the cell based data.

3D Fence Diagrams

3D fence diagrams allow viewing a cross section of a 3D solution. To create/view a 3D fence do the following:

Displaying 3D fences requires:

1) A 3D mesh with solution datasets.
2) A coverage of any type that has one or more feature arcs without any vertices. This defines where the fences will be located. The arcs cannot have vertices since only planar surfaces can be represented.

3D fences can be turned on in the display options dialog. The coverage used for the fence definitions is specified in the display options dialog. The fences will use the current contour settings and are always represented with color-filled contours.

Remember to rotate out of plan view to see the fence.

3.8 Quadtree Module

Quadtree Module

The Quadtree Module contains tools used to construct and edit Quadtrees. A quadtree is a tree data structure in which each internal node has exactly four children. Quadtrees are most often used to partition a two-dimensional space by recursively subdividing it into four quadrants or regions.

In SMS, Quadtrees are synonymous with telescoping grids because currently only the CMS-Flow model can utilize them and they are referred to as telescoping grids by that model and its developers. Since SMS version 12 does not include the interface for the new version of CMS-Flow, this article is only a skeleton and will be filled in as that interface is released.

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

- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone

Creating and Editing Quadtrees

Creating Quadtrees

A quadtree is generated using a Quadtree Generator coverage and uses a technique from previous versions of SMS to generate telescoping grids. The rules for generating such a grid are described in the Telescoping Grid article.

Quadtree Generator Coverage

A generic coverage that can be used for creating feature objects to be converted to a quadtree. The **Create 2D-Grid Frame** tool and **Select 2D-Grid Frame** tool are available with this coverage. A grid frame is required before converting the coverage to a quadtree.

Polygon attributes can be assigned by double-clicking on a polygon or right-clicking on the polygon then selecting the **Attributes** command. The *Polygon Attributes* dialog allows setting the *Maximum grid cell size* .

The quadtree generator coverage has a right-click menu. This menu has the standard coverage menu commands and access to the **Map → Quadtree Grid** command.

Map → Quadtree

The **Map → Quadtree** command is used to construct a quadtree grid using a grid frame feature object in a the current quadtree generator coverage. When the **Map → Quadtree** command is selected, the *Map → Quadtree* dialog appears.

Parameters specified to create the grid include:

- **Grid Geometry** – This section specifies the origin, orientation and size of the grid. The fields of these quantities are populated with default values based on the three points. The orientation is measured as an angle from the positive X axis.
- **Cell Options** – This section specifies the number of cells in each direction in the grid. Several options are available. Specify sizes in the I (Delta U)and J (Delta V) directions or a number of columns and rows. If the *Use Grid Frame Size* toggle is checked, the grid will exactly match the dimensions specified in the *Grid Geometry* section. If that option is not checked, the last row and column may extend beyond the specified lengths. This allows specifying exact grid size or exact cell size.

- **Depth Options** – The elevations or depths assigned to each cell or node can be specified as a single value, or select a dataset to interpolate from.

SMS will generate a quadtree on the input parameters.

Editing Quadtrees

Each of the cells in a quadtree can be subdivided into four subcells by selecting the cell, right-clicking, and selecting the **Split** command. Multiple cells may be selected and split in a single command.

Four sub cells can be merged by selecting them, right-clicking, and selecting the **Merge** command.

Smoothing Quadtrees

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

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

When right-clicking on quadtree data in the Project Explorer, operations for the quadtree appear in a pop up window. One of these is the smooth operation.

Quadtree Smoothing Options

This dialog is accessed by right-clicking on the quadtree data in the Project Explorer and selecting the **Smooth** command. The *Quadtree Smoothing Options* dialog contains the following options:

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

Converting Quadtrees

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

Project Explorer

The following Project Explorer mouse right-click menus are available when the mouse right-click is performed on a Quadtree item. See Quadtree Menus for more information.

Quadtree Module Tools

See Quadtree Module Tools for more information.

Quadtree Module Menus

See Quadtree Module Menus for more information.

How do I?

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

Related Coverages

The grid module currently includes interfaces for:

- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone

Related Topics

- SMS Modules

Telescoping Grids

One of the most restrictive attributes of a Cartesian grid is the limited variability in resolution. By the purest definition, a Cartesian grid consists of square cells, meaning a constant resolution over the entire domain. A method that can be employed to support variable resolution involves the creation/use of what can be called a *Telescoping Grid* (sometimes referred to as a Quad Tree). A Quad Tree is a two-dimensional recursive spatial subdivision. Each region can be subdivided into four child regions. In the SMS application, the regions are always square or rectangular.

The CMS-Flow model is the only numeric engine currently supported by the SMS which allows computation on a telescoping grid.

The generation of a telescoping grid involves:

- A base grid
- Refinement features

Base Grid

The SMS generates a telescoping grid from a user defined base grid. Define this base grid with the approach that is used in any other grid generation operation in the SMS consisting of the definition of a grid frame to define grid extents and either a base cell dimension or number of cells in each of the coordinate directions (I,J). The base grid parameters are specified as a grid frame properties or when using **Map→2D Grid** .

Choosing Base Grid Cell Size

Normally this is a single value for both dimensions (I,J) that represents the typical variation in the geometry. Specifying a value of 50 m indicates that the grid will have a depth every 50 m and any feature smaller than about 100 m will be smoothed away. If a single Cartesian grid is used, this dimension must be small enough to capture the smallest desired feature. For a variable cell grid it must be small enough to capture any feature along the row or column. For a telescoping grid, this can be a general value to represent the general geometric shape of the domain. In some situations there may be justification to have a larger base cell size in one direction that another. For example, all of the features may be aligned with a coastline and the cells could generally be elongated in that direction. This situation would be rare, so a square base cell size is a general recommendation.

Refinement Features

Theoretically a refinement feature could be a single point, a linear feature (arc) or a region (polygon). Future capabilities may allow for a variety of refinement features, but currently the SMS telescoping grid generation capability allows defining feature polygons that enclose areas for which a specific resolution is desired. Assign a "maximum grid cell size" as an attribute on the polygon (double-click on the polygon and assign polygon attributes).

When the SMS generates the telescoping grid, when a cell is generated, the polygon containing the cell centroid is found (if it exists). If either cell dimension is larger than the maximum specified size for the polygon, the cell is split into four sub-cells and the process repeated.

Note, when rectangular cells are being generated (as specified on the base grid), this constraint will force the larger dimension of the cell to be smaller than the specified *maximum size* , so the cell may be significantly smaller than the specified size.

Choosing a Maximum Grid Cell Size

The maximum grid cell size for a polygon should be based on a physical length. One example would be a channel that must be represented in the domain. If the base cell size is 250 m, but the channel has a width of approximately 100 m, the channel would not be represented by the base grid. By specifying a cell size of 20 m for a polygon enclosing the channel, it is possible to enforce approximately five cells to represent the shape of the channel. The feature being represented may be a structure such as a jetty, a natural feature like a channel, or a numerically represented feature such as an eddy current. Geometric features can be detected before generating any grids. Numerical features may require modifications based on preliminary simulations.

Impact of Feature Size on Base Cell Size

In addition to the limitation noted above between a single maximum size and a two dimensional cell, the relationship between the selected base cell size and the maximum specified sizes can also cause what appear to be overly refined grids. Since the cells are created in discrete increments (half of the parent cell dimension), the actual generated size may be smaller than the specified target simply due to binary limitations. For example, if a maximum size of 5 is specified, and the base cell size is 30, the first acceptable cell size that meets the criteria of 5 is actually 3.75 (30/2^3 - 30, 15, 7.5, 3.75). In this case, 3.75 is only 75% of the specified maximum, resulting in cells that are 75% smaller than the user specified acceptable resolution (in the larger direction).

This perceived difference can be reduced by specifying a compatible base cell size. In the previous example, a base cell size of 20 (instead of 30) would result in a cell size of 5 (20/2^2), exactly matching the specified maximum. However, since there are two dimensions to the base cell size, and only one target size, and there can be multiple refinement polygons, each with a specified maximum cell size, the relationship can seem complex.

The SMS includes a tool for telescoping or quad tree grids to compute an ideal base cell size. This option appears in the SMS Grid Frame Dialog which is accessible either by right clicking on the grid frame or when using **Map→2D Grid** . This tool will increase the base cell sizes defined for a telescoping grid so that the largest dimension will be a multiple of the smallest specified grid size. The specified aspect ratio of the cells will be maintained.

> *Note: Telescoping Cartesian grids will be replaced with a more memory efficient Quad Tree structure in SMS 12.0.*

Related Topics

- Cartesian Grid Module

Quadtree Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Quadtree module is active.

- **Select Cell**

The **Select Cell** tool is used to select a grid cell. A single cell is selected by clicking on it. A second cell can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple cells can be selected at once by dragging a box around them. A selected cell can be de-selected by holding the *SHIFT* key as it is clicked.

When a single cell is selected, its Z coordinate is shown in the Edit Window . The Z coordinates can be changed by typing in the edit field, which updates the depth function. If multiple cells are selected, the Z Coordinate field in the Edit Window shows the average depth of all selected cells. If this value is changed, the new value will be assigned to all selected points.

With one cell selected, the Edit Window shows the point i,j location. With multiple cells selected, the Edit Window shows the number of selected cells. The number and size of the cells can be changed in the *Model Control* .

Right-Click Menu

Right-clicking on an active cell while using the **Select Cell** tool will bring up a menu with the following options:

- **Split Cells** – Subdivides a cell into four subcells.
- **Merge Cells** – Merges a group of four subcells back into a single cell.
- **Smooth** – Opens the *Quadtree Smoothing Options* dialog.

Related Topics

- Quadtree Module

Quadtree Menus

The following menus are available in the Quadtree Module :

Standard Menus

See SMS Menus for more information.

Module Specific Menus

Beside the standard menus, the Quadtree Module has the *Data* menu and the *Cells* menu.

Quadtree Module Data Menu

The Quadtree Module *Data* menu commands include:

- **Dataset Toolbox**
- **VTK Data Calculator**
- **Film Loop**
- **Transform**
- **Zonal Classification**
- **Set Contour Min/Max** – This command sets the contour options based on the current options and the selected nodes/vertices or zoom level.
- **Contour Range Options** – This allows controlling if the **Set Contour Min/Max** command applies to dataset specific contour options or the general contour options (for the mesh or scatter modules). It also sets the flags for precision and fill above and below.

Cells Menu

The Quadtree Module *Cells* menu commands include:

- **Split Cells** – Subdivides a cell into four subcells.

- **Merge Cells** – Merges a group of four subcells back into a single cell.

Project Explorer

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

Quadtree Module Root Folder Right-Click Menus

Right-clicking on the Quadtree module root folder in the Project Explorer invokes an options menu with the following options:

- Display Options

Quadtree 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:

- **Delete** – Removes the selected quadtree dataset from the project.
- **Duplicate** – Creates a copy of the selected quadtree dataset.
- **Rename** – Allows changing the selected quadtree dataset.
- **Interpolate to** – Brings up the *Interpolation Options* dialog for the Quadtree Module.
- *Convert to*
  - **Quadtree → VTKMesh** – Creates a new VTK Mesh using the Quadtree dataset. A dialog will appear that lets the user name the new VTK Mesh. After giving the new VTK mesh a name and clicking **OK** , the new VTK Mesh will appear in the Project Explorer.
- **Projection** – Opens the *Object Projection* dialog.
- **Metadata** – Brings up the *Metadata* dialog.
- **Zoom to Quadtree** – Will frame the quadtree in the Graphic Window.
- **Smooth** – Opens the *Quadtree Smoothing Options* dialog.

Quadtree Tool Menus

Some tools in the Quadtree Module have menus that can be accessed by right-clicking while using the tool. See Quadtree Tools for more information.

Related Topics

- Quadtree Module

Quadtree Display Options

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

- Cell edges
- *Contours* – The mesh contours are drawn for the active scalar dataset. Use the contours tab to change contour options.
- *Vectors* – The cartesian grid vectors are drawn for the active vector dataset. All standard vector display options are supported.

- *Grid Boundary* – A line around the perimeter of the quadtree grid can be drawn. This is useful when the cells are turned off. Options allow specifying line color and thickness.
- Cell id
- IJ triad
- Grid name
- Inactive quadtree boundaries
- *Functional Surface* – Show surfaces representing one of the functional datasets associated with a mesh, grid or TIN.

Related Topics

- Quadtree Module
- Display Options

3.9 Raster Module

Raster Module

At a glance

NOTE : the functionality of this module was incorporated into the GIS module starting with SMS version 12.0

- Open and visualize raster data
- Supports many gridded elevation file formats. A complete list can be found at: www.globalmapper.com/product/formats.htm
- One or more rasters are placed under a raster set in the project explorer.
- Convert raster to TIN (scatter set)
- Interpolate data from raster to TINs /2D Mesh/2D Grid
- Use rasters with observation profile plots

Starting with SMS 12.0, rasters are managed in SMS as objects in the GIS module. All previous functionality is available on raster entities in that module.

Rasters contain data (usually elevation) stored in pixels. Their resolution can vary, depending on the number of x and y cells the raster contains. The Raster Module allows opening and visualizing rasters of various formats and convert them into TIN (scatter sets), and interpolate their data into scatter sets, 2D meshes, and 2D grids.

The Raster module is included with all paid editions of SMS.

Raster Sets

Each raster will be stored in a raster set. A raster set may contain multiple rasters. Ofter the data in one raster is also associated to another raster, when each raster is used to cover an area. When this is the case, group the rasters together under a single raster set. Raster sets are used in doing interpolation and plotting.

A new raster set can be created by right-clicking on the root raster item in the project explorer, then select **New Raster Set** . Existing raster's can be moved into a raster set by simply dragging the raster into the raster set.

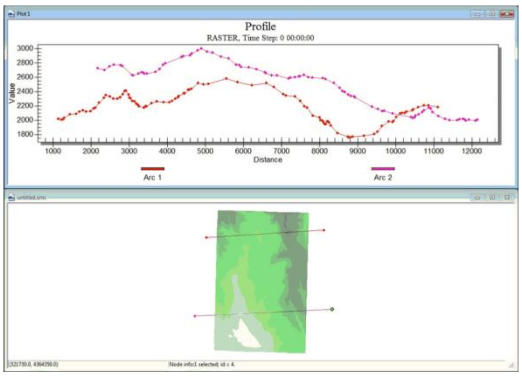
Related Topics

- Raster Module Interface
- Raster Functionalities
- Raster Options

Raster Functionalities

Profile Plot

SMS can generate many different types of 2D plots. Scalar data contained in visible rasters may be plotted using a specific type of 2D plot called a profile plot . When creating a profile plot for a raster using the Plot Wizard and 'specified data' is selected in Step 2, be sure to set the appropriate "Raster Set". Data will be plotted from the rasters in the selected raster set.



Interpolation

Values contained in rasters can be interpolated to existing scattersets, 2D meshes, and 2D grids. To interpolate the values from a raster, right-click on the raster set in the *project explorer* and choose **Interpolate** . Then choose the appropriate option for the object to interpolate to.

Convert to Scatter

Values from rasters can also be converted into scatter data. This is done by right-clicking on the raster item in the *Project Explorer* and selecting *Convert* | **Raster→2D Scatter** .

In special cases, one would want to convert only a selected portion of the raster to scatter data. For more information, see Raster Tools .

Raster Values as Elevation / Z Data

By default data associated with the raster is stored as elevation/z data. If a rasters projection is changed and the units change from meters to feet, the z values also would change. This becomes problematic if the raster values is used to represent NCLD land data and not elevation data. To keep NCLD land data from being projected, specify this in the raster set . This is done by right-clicking on the raster set and selecting **Options..** . A dialog appears to specify if the values are elevations or not elevations. If raster values are set as not elevations, when doing a projection the z values will remain unchanged.

Related Topics

- Plot Window

Raster Module Interface

Clicking the raster module icon in the module tool bar will bring the raster module interface to display on the screen. The raster module interface can also be brought to display by clicking on a raster item in the Project Explorer. The raster module interface consists of the menu, tools, and Project Explorer right-click menus.

Raster Menu Items

The menu bar that appears in the Raster Module:

- *File* – Has standard features as in other models.
- *Edit* – Has standard features as in other models.
- *Display* – Under the *Raster* tab in *Display Options* , choose to *Display as raster* or *Display as surface* . *Display as raster* is a 2D representation, while *Display as surface* uses data from pixels as elevation to show a 3D surface. It should be noted that these display options affect performance. *Display as raster* takes less memory and performs faster because it is a more efficient data structure. If *Display as surface* is selected, options are available to turn on or off contours, edges, or boundaries.
- *Web* – Has standard features as in other models.
- *Window* – Has standard features as in other models.
- *Help* – Has standard features as in other models.

Raster Tools

The Raster Module contains only one dynamic tool called the **Select Points** tool. Using this tool, one can edit/select one or more corners (points) of the pixels which make up the raster. Selecting cells with the **Select Points** tool will show information about the points in the status bar.

Right-Click Option

When one or more points are selected, one can right-click in the graphics window, and the option to convert selected raster points to a scatter set appears. If this option is clicked, a new scatter set will appear in the project explorer.

Raster Right-Click Menus

Right-Click Menu of "Raster Set" Project Explorer Item

Right-clicking on the raster item in the Project Explorer will bring up the following options:

- **Delete** – Removes the current coverage.
- **Rename** – Allows changing the current coverage name.
- **Options** – Clicking this will bring up a *raster options* dialog. This specifies if the z values are elevations . If NCLD land data is being used, specify that raster values are not elevations. Then if the projections are changed, the z values that store the land data would not be affected.
- **Interpolate** – Data from raster sets may be interpolated to already existing TINs, meshes and 2D grids.

Right-Click Menu of "Raster" Project Explorer Item

Right-clicking on the raster item in the Project Explorer will bring up the following options:

- **Delete** – Removes the raster set.
- **Rename** – Allows changing the raster set name.
- **Convert** – SMS allows rasters to be converted into TINs.
- **Zoom to Raster**

Right-Click Menu of Dynamic Tools

See Raster Tools above.

Related Topics

- Project Explorer

- Toolbars
- SMS Menus
- Dynamic Tools

3.10 Scatter Module

Scatter Module

At a glance

- Used to create, edit, and visualize triangulated irregular networks
- DEMs can be read in and converted to TINs
- Filter scatter sets to eliminate redundant data
- Datasets can be interpolated to other modules (meshes, grids, etc)

Overview

The Scatter Module (previously known as the Scattered Data Module) is used to interpolate spatial data values from groups of scattered data points or ordered grids (DEMs) to the other data types (i.e., meshes and grids). SMS supports three interpolation schemes including linear, natural neighbor and inverse distance weighted. The module is also used to view and edit survey data (i.e. SHOALS data).

Interpolation is useful for setting up input data for analysis codes. Generally, the data gathered from a site to be modeled varies in density. Generating a finite element mesh directly from these points would result in a very low quality mesh. Further this data does not lie in a grid for use as a finite difference grid. Interpolation allows the gathered data points to be used as background information that can then be used to generate a base mesh or grid in the Mesh Module, the Grid Module or the Map Module. The only consideration of bathymetry for such a mesh or grid would be the definition of element edges along geometric or property features. The actual bathymetry comes from the scattered data. SMS interpolates this data to the created mesh or grid points.

Interpolation may also be used to create datasets for one mesh from data related to another mesh of the same region. For example using a mesh of a river reach for which analysis has been preformed. If a bridge is to be added to the reach, the mesh topology changes. The data from the first mesh can be converted to a scattered dataset and then interpolated to the second mesh. This data may be used as initial conditions for the second mesh, or compared to results of analysis run on the second mesh using the Dataset Toolbox .

A third purpose of interpolation is to create additional datasets from either observed, or calculated data.

The Scatter module is included with all paid editions of SMS.

Data Sources

There are various potential sources for background data in an SMS project. These include:

- **local surveys**

  Local surveys must be formatted into an SMS supported format. The most intuitive format and easiest to use is a tabular file of coordinates.

  If this data is to be augmented with previous models or historical surveys, the coordinate system of the local survey must be defined relative to the historical survey or a global coordinate system.

- **historical surveys**

  There are several sources of historic surveys. These include previous studies done by a modeler or company and compiled databases such as GEO-DAS or ETOPO. These data sources can be imported into SMS and used either as the basis of a finite computation domain (mesh or grid), or as a scattered dataset or DEM. Care must be taken into account the age and quality of the data and make sure all data sources are converted to a single coordinate system.

- **digital elevation maps**

  Digital Elevation Maps (DEMs) are regular structured grids of elevation values. Since the data is structured, it can be read, stored, displayed and utilized more efficiently than scattered datasets. These data sources are becoming more prevalent and can be obtained for topographic regions of the entire United States and several other area of the world from web sites such as Terraserver.

  Unfortunately, most DEMs available online do not include bathymetric portions of the domain, which makes their use in SMS limited. New data bases are being developed, but due to the dynamic nature of bathymetric information, the feasibility of an extensive database is very slight.

  It may be useful to convert scattered datasets into DEMs for faster processing inside of SMS.

- **electronic charts**

  Since surveys can be expensive to obtain, and DEMs may not be applicable, another option available for the hydraulic modeler is the use of topographic/bathymetric charts or historic nautical charts. If these types of maps can be digitized into an electronic format, they can be read into SMS and displayed on the screen. The goal is to create a scattered dataset from this electronic chart. The steps to do this include:

  1) Scan the paper map and save it as an image (*.tif, *.jpg, ...).
  2) Register the image (if desired, mark the map with the register points prior to scanning it).
  3) Select the Create Vertex tool in the Data Module.
  4) Digitize (click on the image on the screen) to create a vertex on a contour line in the image.
  5) In the z edit box of the edit window set the z value to the contour value of the line.
  6) Digitize along the specified contour value (the spacing of points along the contour lines should be approximately the same distance as the spacing between adjacent contours).
  7) Repeat steps 4–6 for each contour line. Spot elevations can be entered by setting the z value to the value of the spot elevation and then creating a vertex at that location.
  8) Triangulate the vertices once done.

  This method becomes tedious for larger areas, but is ideally suited for smaller areas where there are not too many contours to be digitized.

In addition, when DEM data is brought into SMS, the data is triangulated and stored as a scatter set.

It is also possible to convert CAD and GIS data into scatter sets. This is accomplished by right-clicking on the object in the project explorer and selecting the **Map → Scatter** command. This command searches the data for triangular and quadrilateral faces and converts them to triangles in a triangulated surface (TIN). Points along contours or polylines are not converted using this command. In order to use these points in a scattered dataset format, first convert them to feature objects in the Map module.

Scatter Interface Components

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

Scatter Display Options

Accessed through the *Display Options* dialog. Provides options as to how Scatter Module elements will be displayed. See Scatter Module Display Options for more information.

Scatter Module Menus

The following menus are available in the the Scatter Module:

Standard Menus

Standard menus such as the *File* , *Edit* , and *Display* menus are available. See SMS Menus for more information.

Module Specific Menus

- Data
- Vertices
- Breaklines
- Triangles
- Scatter

Scatter Right-Click Menus

The Scatter Module has right-click menus for root folders and scatter set items in the Project Explorer. See Scatter Module Right-Click Menus for more information.

Scatter Module Tools

The Scatter Module has eight unique tools for creating and manipulating scatter data in the Graphics Window. See Scatter Module Tools for more information.

Scattered Datasets

The Scatter Point Module is used to visualize and apply various types of data. This data typically comes from surveys, digital maps, previous numerical analysis or digitization on screen. The data is stored as sets or groups of 2D scattered data points with associated values. The most common value is bathymetry and is used to create the geometric representation of the area being modeled.

SMS connects the scattered data points into triangles forming a Triangulated Irregular Network (TIN). TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for a source of bathymetric or other data in a numerical model. TINs can also be used to compute areas, volume, distances, gradients and several other geometric parameters.

SMS applies data from scattered datasets to finite element networks or grids via interpolation. This allows poorly distributed elevation data to be assigned to a well-structured set of elements to create the bathymetry of the entire mesh. A variety of interpolation schemes are supported. Internally, the scattered data sets are triangulated to create surfaces for continuous interpolation. Since the connectivity of the triangulation affects the interpolation, SMS provides tools to allow for the manipulation of this triangulation. The triangulation also allows contouring of the scattered data set to visualize the data.

Multiple scatter point sets can exist at one time in memory. One of the scatter sets is always designated as the "active" scatter point set. The active scatter set can be changed by changing the *Scatter Set* combo box in the top *Edit Window* . Whenever a new scatter set is created, it becomes the active set.

Data Interpolation

Scatter dataset can be interpolated to other data types (grid, mesh, etc.). See Scatter Interpolation for more information.

Practical Notes

Does SMS have a way of measuring the difference in volume between two bathymetric surveys?

To do this, the following needs to be done:

1) Interpolate the elevation from one survey onto the other.
2) Use the *data calculator* to compute the difference between the two elevations. If desired, do max(0.0, z1-z2) as well as max(0.0, z2-z1) to get both deposition and erosion volume.
3) Turn on the *Volume* option in the *Info Options* dialog.
4) Select the triangles of interested. The volume appears in the info window at the bottom of the screen. It is also possible to direct these values to a file or another window through the *Info Options* settings.

How do I compare datasets from different scatter point sets?

Datasets within a scatter set are associated with the geometry of that scatter set. To compare datasets from different scatter sets, it is necessary to first interpolate the datasets to a common geometry. Below are guidelines on how to do this with a mesh and with a scatter grid.

- Mesh
  1) Interpolate the first dataset to mesh.
  2) Interpolate the second dataset to the mesh.
  3) Use the *data calculator* (*Data* | **Data Calculator**) to compare the two datasets.
- Scatter Grid
  1) Select first dataset.
  2) Select *Scatter* | *Interpolate to Scatter* | **…to Scatter Grid** . Specify extents and resolution of grid.
  3) Select second dataset.
  4) Select *Scatter* | *Interpolate to Scatter* | **…from other scatter set** . Specify the second scatter set.
  5) Select *Data* | **Data Calculator** with the new scatter grid selected to compare the two datasets.

Related Topics

- Digital Elevation Maps
- Mesh Generation
- Scatter Interpolation
- Scatter Module Display Options

External Links

- Jun 2002 ERDC/CHL CHETN-IV-43 SHOALS Toolbox: Software to Support Visualization and Analysis of Large, High-Density Datasets [71]

3.10.a. Interface Components

Scatter Module Display Options

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

- *Points* – A symbol is drawn at each point. The type, radius, and color of these symbols can be specified. The toggle below the *Points* item specifies that rather than coloring the symbols with the specified color, a contour color should be used based on the current scalar value at the point. This gives a contouring effect without generating/displaying the contours.
- *Triangles* – Triangle edges are drawn using the specified line attributes. Line attributes include color, thickness, and style (dashed/solid).
- *Boundary* – A line around the perimeter of the scatter set can be drawn. This is useful when the triangles are turned off. Line color and thickness can be specified.
- *Contours* – The scatter contours are drawn for the active scalar data set for the active scatter set. All standard contour display options are supported for scatter contours.
- *Velocity Vectors* – The scatter vectors are drawn for the active vector data set of the active scatter set. Display options are set through the *Vector Display Options* dialog.
- *Inactive Color* – Only the active scatter set is displayed in its color. All other scatter sets are displayed using the inactive color. This helps to avoid clutter on the screen.
- *Nautical Grid*
- *Point Names* – The name of the selected scatter set can be changed.
- *Point Numbers* – The scatter point id number can be displayed next to each node. Font and color can be selected.
- *Scalar values* – The scalar value of the active function is displayed next to each point. The options button opens the *Scalar Value Options* dialog.
  - *Use contour color scheme* – Use the color ramp specified for the contours for text color rather than a specified color.

The display of individual scatter sets can be turned on or off through the Project Explorer .

Related Topics

- Display Options
- Scatter Module

Scatter Project Explorer Items

In the Project Explorer, the Scatter Data folder houses all of scatter data information that is manipulated by the Scatter module. The Scatter Data folder will not appear in the Project Explorer until scatter data is opened in SMS. Once scatter data is opened, the Scatter Data folder will appear with the Scatter set item below it. There will also be elevation information that appears under the Scatter set item. The Scatter module may be activated by clicking on the Scatter set item in the Project Explorer. Once active, the *Scatter* module tool bar menu will appear to the right of the Project Explorer.

Scatter Module Right-Click Menus

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

Scatter Module Root Folder Right-Click Menus

Right-clicking on the Scatter module root folder in the Project Pxplorer invokes an options menu with the following options:

- **New Scatter Set** – Creates a new, empty scatter set.
- **Display Options** – Brings up the *Display Options* dialog.

Scatter Set Item Right-Click Menus

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

- **Split** – Creates a new scatter set containing the selected scatter vertices. Selected scatter vertices are removed from the original scatter set.
- **Autogenerate breaklines** – Automatically creates breaklines following specified elevations.

Related Topics

- Scatter Module

Scatter Module Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Scatter Module is active. Only one tool is active at any given time. The action that takes place when clicking in the Graphics Window depends on the current tool. The following sections describe the tools in the Scatter tool palette.

Select Scatter Point

The **Select Scatter Point** tool is used to select scatter points (also known as vertices). A single point is selected by left-clicking directly on it. Multiple points can be selected at once by dragging a box. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. Additional scatter points can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new points without holding the *SHIFT* key will first clear the selection list and then add the newly selected points. A selected point can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool.

Scatter points are locked by default so they are not accidentally dragged, but can be unlocked using the *Scatter Vertices* Menu . When a single point is selected, its location is shown in the *Edit Window* . The Z coordinates can be changed by typing in the edit field. The active scalar function is updated when the Z coordinate changes. If multiple points are selected, the Z Coordinate value shown is the average scalar value of all selected points. If this value is changed, the new value will be assigned to all selected points.

The *Graphics Window's* status bar will display information on the selected items depending on the settings find through the *File* | **Info Options** command in the *File* Menu .

Selected scatter points can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* Menu , by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Triangles attached to the deleted scatter points are deleted. The resulting void can be retriangulated.

This tool is available when one or more scatter points exist.

Right-Click Menu

- **Find** – Available when nothing selected. Brings up the "Find..." dialog to search for a vertex by ID or location. Like the menu command.
- **Scatter Options** – Available when nothing selected. Brings up the *Scatter Options...* dialog. Like the menu command.
- **Triangulate** – Available when nothing selected. Triangulates the entire scatter set. Like the menu command.
- **Select Thin Triangles** – Available when nothing selected. Selects thin triangles around the boundary. 'Thin' is defined by the aspect ration in the scatter options. Like the menu command.
- **Delete Long Triangles** – Available when nothing selected. Deletes long triangles around the boundary. 'Long' is defined by the aspect ration in the scatter options. Like the menu command.
- **Select All** – Available when nothing selected. Selects all vertices.
- **Delete** – Available when one or more vertices selected. Deletes the selected scatter vertex/vertices.
- **Split Breaklines** – Available when one or more vertices selected. Splits all breaklines that go through the selected vertex into two breaklines (one on each side of the selected vertex). If the vertex is the end of the breakline or is not used in any breaklines, the command has no impact.
- **Assign Point Name...** – Available when one or more vertices selected. Prompts for a name for the selected point. The default name will be "Point #" where '#' is the ID of the point. Points do not have names unless specifically specified. Names can be displayed in the Scatter Module display options.
- **Clear Selection** – Available when one or more vertices selected. Unselects all the selected vertices.
- **Invert Selection** – Available when one or more vertices selected. Selects all the unselected vertices and unselects all the selected vertices. This can be useful when many vertices are to be deleted. First select those to be kept and then invert the selection.
- **Zoom to Selection** – Available when one or more vertices selected. Recursively zooms in on the selected objects. Invoke this command multiple times to zoom in tighter.

Create Scatter Point

The **Create Scatter Point** tool is used to place new scatter point in a Scatter Set. A single point or vertex is created at a time by left-clicking at the coordinate desired. The newly created point is selected to allow Z Coordinate changes in the Edit Window . Scatter points are locked so X and Y Coordinates can not be edited.

This tool is always available, however, creating a scatter point is only allowed while in plan view.

Select Breaklines

The **Select Breaklines** tool is used to select breaklines . Holding the *SHIFT* key while selecting breaklines will add breaklines to the selection that are not already selected and remove breaklines that are already selected from the selection. Multiple nodestrings can be selected by dragging a box around the breaklines. Holding the *CTRL* key and clicking will enable selection with a polygon. Holding the *CTRL* key and dragging will enable selection with an arrow.

The *Edit Window* shows the number of breaklines selected.

Right-Click Menu

- **Delete Selected** – Deletes the selected scatter breakline(s).
- **Merge Selected** – Merges multiple breaklines that connect end to end into a single breakline. An error is reported if two (or more) breaklines are selected that do not have common end points or a single break line is selected.
- **Force Breaklines** – Swaps the triangle edges that cross the selected breakline(s) to ensure no triangles cross the breakline(s).
- **Clear Selection** – Unselects all the selected breaklines.
- **Zoom to Selection** – Recursively zooms in on the selected objects. Invoke this command multiple times to zoom in tighter. If no breaklines are selected, this command zooms to all breaklines.
- **Select All** – Selects all breaklines.

Create Breaklines

The **Create Breaklines** tool is used to create breaklines . Breaklinesare used to control the connectivity of a scatter set. To create a breakline:

1) Click on a scatter vertex. The vertex will be highlighted in red and a prompt will be shown in the *Help Window* .
2) Click on any vertex to add it to the breakline. The selected vertex is also highlighted in red and a solid red line is drawn between the two vertices. Continue adding vertices to the breakline in this manner.
    1) Note: Vertices in the breakline can be adjacent, but this is not required. A breakline will usually be made of vertices which are not adjacent.
    2) Press the *BACKSPACE* key to backup one vertex. Press the *ESC* key to abort the breakline creation.
    3) Double-click a vertex or press the *ENTER* key to end the breakline creation.

3) The *SHIFT* and *CTRL* keys assist in creating large breaklines which include sections made up of adjacent nodes. These can be used after at least one vertex has been selected and function as follows:
    1) *SHIFT* – Holding down the *SHIFT* key and selecting another vertex will add to the breakline all vertices between the two. The path chosen is the shortest distance between the two vertices that follows triangle edges.
    2) *CTRL* – Holding down the *CTRL* key and selecting another vertex will add to the breakline all vertices on the scatter set boundary between the two, going counter clockwise from the first vertex to the second vertex. Both vertices must be on the boundary of the scatter set or SMS will beep.
    3) *CTRL* + *SHIFT* – Holding down both the *CTRL* and *SHIFT* keys and selecting another vertex will add to the breakline all vertices on the scatter set boundary between the two, going clockwise from the first vertex to the second vertex. Both vertices must be on the boundary of the scatter set or SMS will beep.

Select Triangle

The **Select Triangle** tool is used to select triangles. A single triangle is selected by clicking inside it. Multiple triangles can be selected at once by dragging a box or an arrow. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. To drag a selection arrow, left-click while holding the *CTRL* key (the *CTRL* key can be released after the click) and hold the mouse button and drag the mouse to form an arrow of the desired length and direction; release the button to impale and select the triangles through which the arrow passes. Additional triangles can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new triangles without holding the *SHIFT* key will first clear the selection list and then add the newly selected triangles. A selected triangle can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESCAPE* key will clear the entire selection list. Right-clicking will open a menu specific to this tool.

When a single triangle is selected, its centroid location is shown in the *Edit Window* . If multiple points are selected, the Z value shown is the average scalar value of all selected triangles.

The *Graphics Window's* status bar will display information on the selected items depending on the settings find through the *File* | **Info Options** command in the *File* Menu .

Selected triangles points can be deleted by selecting the *Edit* | **Delete** menu command on the *Edit* Menu , by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. The Scatter vertices (points) of the deleted triangles are not deleted. The resulting void can be retriangulated.

This tool is available when one or more scatter triangles exist.

Right-Click Menu

- **Select Thin Triangles** – Selects thin triangles around the boundary. 'Thin' is defined by the aspect ration in the scatter options. Like the menu command.
- **Delete Long Triangles** – Deletes long triangles around the boundary. 'Long' is defined by the aspect ration in the scatter options. Like the menu command.
- **Select All** – Available when nothing selected. Selects all vertices.
- **Process Boundary Triangles** – Available when nothing selected. Like the menu command.
- **Delete** – Available when one or more triangles selected. Deletes the selected scatter triangle(s).
- **Clear Selection** – Available when one or more triangles selected. Unselects all the selected triangles.
- **Invert Selection** – Available when one or more triangles selected. Selects all the unselected triangles and unselects all the selected triangles . This can be useful when many triangles are to be deleted. First select those to be kept and then invert the selection.
- **Zoom to Selection** – Available when one or more triangles selected. Recursively zooms in on the selected objects. Invoke this command multiple times to zoom in tighter.

Create Triangle

Most triangles in SMS will be created using automatic triangulation. At times, however, it is necessary to create a triangle, especially after deleting triangles. This is done using the **Create Triangle** tool.

To create a single triangle, select the vertices of the desired triangle by left-clicking on each scatter point or by dragging a selection box. As points are selected individually, they will be highlighted. Also, triangle edge will be drawn between the first and second vertex. To remove the last highlighted point from the desired triangle, press the *DELETE* or *BACKSPACE* keys. To abort the creation of a triangle, press the Escape key. When three points are selected, SMS will try to create the desired triangle. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. The selection box must only contain the three desired vertices to create the triangle. If the new triangle will overlap existing triangles the triangle will not be created. If more then three points are selected (via a selection box) then SMS will not attempt to create a new triangle.

This tool is available when three or more scatter points exist.

Swap Triangle Edge

The **Swap Edges** tool is used to manually swap the edges of two adjacent triangles. This is useful in such cases as avoiding an artificial dam in a channel. Left-click on the desired triangle edge to swap it. The edges of the triangles will only be swapped if the resulting triangles are valid.

This tool is available when two or more scatter triangles exist. Swapping edges is only allowed while in plan view.

Create Mesh Nodes Contour

The **Create Mesh Nodes Contour** tool is used to temporarily display a Z value on a TIN. Left-click at a location to display the value. The labels will not reappear if the graphic window view changes.

This tool is available when at least one scatter triangle exists. Contour labeling is only allowed while in plan view.

Related Topics

- Scatter Module

3.10.a.1. Scatter Module Menus

Scatter Menu

The items in the *Scatter* menu in the Scatter module are described below. The menu items all work with the active scatter set unless otherwise noted.

Listed below are the general commands in the *Scatter* menu.

Scatter Options

Scatter options are accessed through the *Scatter* menu, *Scatter Options* dialog in the Scatter Module.

Merge Scatter Sets

Multiple scatter sets can be merged into a single scatter set using the *Merge Scatter Sets* dialog. The menu command *Scatter* | **Merge Sets** opens the *Merge Scatter Sets* dialog.

Selecting scatter sets to merge

The *Merge Scatter Sets* dialog contains a spreadsheet listing all of the scatter sets currently loaded into SMS. Scatter sets to merge are specified by checking the *Merge* box in the *Merge column* of the spreadsheet. When merging scatter sets, only one dataset is transferred to the merged scatter set. The dataset to be transferred is specified for each scatter set in the *Dataset* column of the spreadsheet.

The *Priority* column of the spreadsheet is only used if the overlapping region option is set to *Delete lower priority scatter points* . This option is explained below.

Merge Options

The following options are available when merging scatter sets:

- *Name* – Specify the name for the new, merged scatter set.
- *Delete original scatter sets* – The scatter sets to be merged are deleted after the new, merged set is created.

Overlapping region options

- *Merge all scatter points* – All scatter points from all scatter sets to be merged are combined into one set and retriangulated.

- *Delete lower priority scatter points* – In regions where scatter points and triangles overlap, the scatter points and triangles from the lower priority scatter set are deleted. The priority is based on the *Priority* column of the *Select scatter sets to merge* spreadsheet. The **Move up** and **Move down** buttons can be used to adjust the priority of the scatter sets when this option is selected.
    - *Maintain triangulation* – The triangulation of the original scatter sets is maintained. New triangles are created to connect the original scatter sets into a single, merged scatter set.

Merge Report

When the merge finishes a merge report will be displayed on the screen. This report shows statistics for the scatter sets that were merge such as number of vertices and triangles before and after the merge. If desired vertices are being deleted, check the duplicate points tolerance. This is found at *Scatter→Scatter Options* dialog.

Assign Point Name

Active when a scatter point, or group of points, have been selected. This brings up a dialog that allows assigning a unique name to the selected point.

Interpolate to Mesh

If mesh nodes exist, the *Interpolation* dialog appears where the interpolation options are set. The scatter point datasets are then interpolated to the mesh nodes using the user specified interpolation options.

Interpolate to Cartesian Grid

If a cartesian grid exists, the *Interpolation* dialog appears where the interpolation options are set. The scatter point function values are then interpolated to the center of each grid cell.

Interpolate to Quadtree

If a quadtree exists, the *Interpolation* dialog appears where the interpolation options are set. The scatter point function values are then interpolated to the center of each grid cell.

Interpolate to Scatter

Interpolating on scatter set to another scatter set has three options:

Interpolate to Scatter Grid

If a scattered dataset exists, the **Interpolate to Scatter Grid** menu item (Scatter module, *Scatter* menu) brings up the *Grid Frame* dialog. This dialog, positions the purple grid frame and sets up the number of rows and columns in the grid. After pushing **OK** , a new scatter set is created with scatter points at the corners of each grid cell. The original scatter set is interpolated to the new scatter grid set using linear interpolation.

Using the scatter grid is a form of data decimation: a dense scatter set can be represented as a less dense scatter set.

Interpolate from Other Scatter

This option interpolates one scattered dataset to another set. Two sets must exist for this option to be enabled. An *Interpolation* dialog appears where the scatter set to interpolate from is selected. That scatter set is interpolated to the active scatter set. The interpolation uses an extrapolation value of 0.0.

Interpolate to Nautical Grid

This option creates a nautical chart. A nautical chart divides a scatter set into bins and finds the maximum, minimum, and average depth over each bin. The **Interpolate to Nautical Grid** menu item (Scatter module, *Scatter* menu) brings up the *Grid Frame* dialog. The dialog positions the purple grid frame and sets up the number of rows and columns in the grid. After pushing *OK* , a new scatter set is created with scatter points at the center of each grid cell. Three functions are created for each scatter point from the active scalar function of the original scatter set:

- **Average** – The average depth over each bin.
- **Minimum** – The minimum depth over each bin.
- **Maximum –** The maximum depth over each bin.

Requirements to interpolate to a nautical grid inlcude:

1) A scatter dataset must exist.
2) Active coverage type must allow grid frames to be created. The Cartesian Grid Module model coverage types allow the creation of grid frames.

Interpolate to Map

This option will assign the elevation values from on scatter set to the active map coverage. The *Interpolation* dialog will appear, where the interpolation method can be selected for assigning elevation data to the map coverage.

Obsolete Commands

The following commands are no longer available in current released of SMS:

Create Scatter Subset

All selected points from the original scatter set are moved from the original set into a new scatter set. A prompt will ask for the name of the new scatter set. If all points for the current scatter set are selected, nothing occurs. The two scatter sets, the original and the new, are retriangulated.

Delete Scatter Set

This option is found in the Scatter module in the *Scatter* menu. If one scatter set exists, a prompt will ask if wanting to delete the active scatter set. If more than one scatter set exists, a dialog appears. The scatter sets can be flagged for deletion in this dialog. Double-clicking on a scatter set in the window or pushing the **Delete** button flags or unflags a scatter set for deletion. **Select All** or **Deselect All** will flag or unflag all sets. A set is flagged if the letter "d" appears to the left of the scatter name.

Related Topics

- Scatter Module
- Scatter Module Menus

- Scatter Triangles Menu

Scatter Data Menu

Most of the SMS modules have a *Data* menu, but the items in this menu are different for each module. The menu items work with the active scatter set unless otherwise noted. The Scatter Module commands include:

Dataset Commands

Dataset Toolbox

Opens the *Dataset Toolbox* containing various tools to work with datasets. Includes the *Data Calculator* options.

Smooth Size Dataset

Opens the *Smooth Dataset* dialog that can be used to condition scattered data scalar values.

Transform

Brings up the *Transform* dialog. The **Transform** command is used to move scatter points.

Zonal Classification

Can be used to identify areas that meet a set of criteria. The criteria can be based upon scalar dataset values and/or specific material ids in a coverage. Opens the *Classification Wizard* .

Visualization Commands / Options

Contour Options

Brings up the *Display Options* dialogue. See Contour Options for more information.

Vector Options

Brings up the *Display Options* dialogue. See Vector Display Options for more information.

Film Loop

Opens the *Film Loop Setup Wizard* . See Animation for more information.

Data Conversion Commands

Scatter → Mesh

The scatter points are converted to a mesh by this command. All functional data, scalar and vector, is copied to the mesh. The mesh nodes can be triangulated in the Mesh module using the menu command *Elements* | **Triangulate** .

Scatter Contour → Feature

When this command is invoked, the *Create Contour Arcs* dialog opens. For more information, see Create Contour Arcs .

Boundary → Feature

The outer boundary of the scatter set is converted to Map module arcs. Arcs are created where scatter triangles do not border another triangle.

Scatter Commands

Find

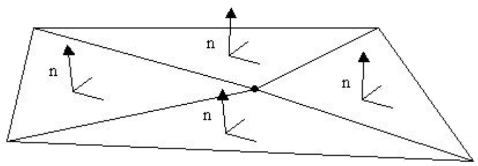
Finds a specific scatter point either by specifying its ID or by specifying the nearest (x,y) coordinate. This can be useful if one specific scatter point needs to be located in a large scatter set.

Scatter Filter

There are two filter options. The first option is to filter by adjacent triangle normal angle filter. The second option is to filter by using the VTK Decimate Pro. Access the *Filter Options* dialog by going to *Data* | **Filter...** while in the Scatter Module.

Filter – Adjacent Triangle Normal Angle Filter

Redundant and overlapping data may exist in a scatter. SMS offers the ability to filter the data and remove unnecessary data points in relatively flat areas in the *Filter* option from the Scatter module, *Data* menu. Specify an angle. Each data point is checked to see if it is in a flat region by dotting the normals of the surrounding triangles.



If the normals are all within the specified angle, the region is flat and the point is deleted.

Filter – VTK Decimate Pro

VTK Decimate Pro is a filter to reduce the number of triangles in a triangle mesh, forming a good approximation to the original geometry. To get a detailed description of the options to set when performing this kind of filter, please visit the website:
http://www.vtk.org/doc/release/4.0/html/classvtkDecimatePro.html

Select/Delete Duplicate Points

This menu item changes according to the option set in the *Scatter Options* dialog. In the options, choose to select or delete duplicate points and set a tolerance. The check works by checking each point and selecting/deleting any point that is within the tolerance. Points with lower ids are checked first; the point with the higher id is therefore selected/deleted.

Obsolete Commands

The following commands have been removed from current versions of SMS.

Create Datasets

Brings up the *Create Datasets* dialogue for creating functions for the active scatter set. No longer available starting in SMS 10.1. Has been replaced by the *Dataset Toolbox*.

Data Calculator

Can be used to perform mathematical operations with datasets to create new datasets.

Related Topics

- Scatter Module
- Scatter Module Menus
- Create Coastline
- Coastline Files

Scatter Breakline Menu

The *Breaklines* menu provides commands for processing breaklines in the scatter set. Breaklines can either be imported or created manually in SMS.

The menu items operate on the active scatter set unless otherwise noted:

Force Breaklines

Processes breaklines in a scatter set by requiring scatter triangle edges to follow the selected breaklines . SMS will swap triangle edges to get the edges to conform to the breakline. Contour values of the elements will be adjusted to fit the new triangulation. If no breaklines are selected, all breaklines will be forced. This command is also available in the **Select Breakline** tool right-click menu.

Merge

Joins selected breaklines to form a single breakline. Available if more than one breakline is selected. The selected breaklines must share the same endpoint(s) in order to be merged. Identical to the **Merge Breaklines** command in the **Select Breakline** tool right-click menu.

Split

Divides a single breakline into multiple breaklines at the selected scatter vertex. Available if a scatter vertex is selected that is connected to (and not at the end of) a breakline. Functions identical to the **Split Breakline** command in the **Select Scatter Point** tool right-click menu.

Related Topics

- Breaklines
- Scatter Module
- Scatter Module Menus
- Importing Scatter Breaklines

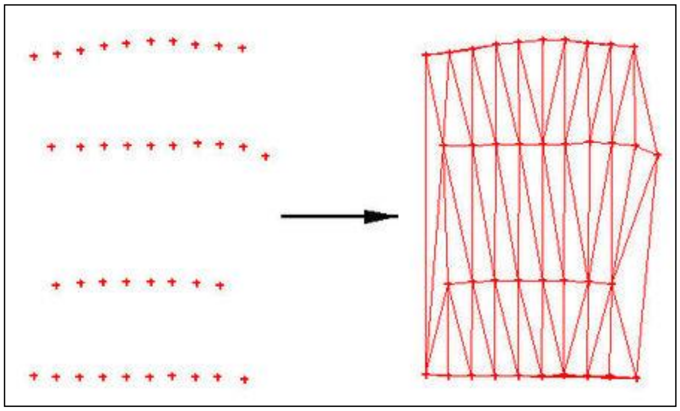
Scatter Triangles Menu

The items unique to the Scatter module are listed below. The menu items operate on the active scatter set unless otherwise noted:

General Commands

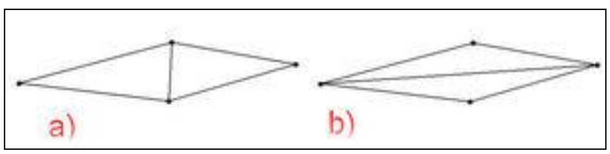
Triangulate

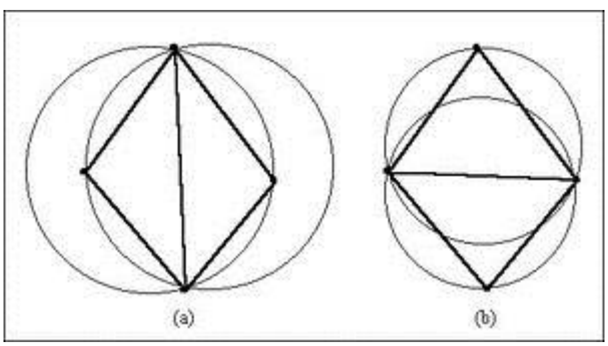
Scatter points or mesh nodes can be triangulated to form piecewise linear surfaces. For scattered data, these surfaces are also referred to as TINs (Triangular Irregular Networks). For mesh nodes, they form a finite element mesh. The points/nodes are connected into surfaces as scatter sets or meshes are created, but at times it may be necessary to reconnect the points (i.e. after deleting individual points/nodes or triangles/elements). New triangles are constructed in mass by triangulating a set of points when the **Triangulate** command from the *Triangles* menu is executed. The selected points are connected with a series of triangles. If points are not selected, then all points will be triangulated.



Delaunay Criterion

The resulting triangulation satisfies the Delaunay criterion. The Delaunay criterion ensures that no vertex lies within the interior of any of the circumcircles of the triangles in the network as shown below:





The result of enforcing the Delaunay criterion is that long thin triangles are avoided as much as possible.

Triangulate

The vertices associated with the active scatter set can be triangulated using the **Triangulate** command from the *Triangles* menu in the Scatter module . Mesh nodes (either the selected nodes, or all nodes) can be triangulated using the **Triangulate** command from the *Elements* menu in the 2D Mesh module .

Optimize Triangulation

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

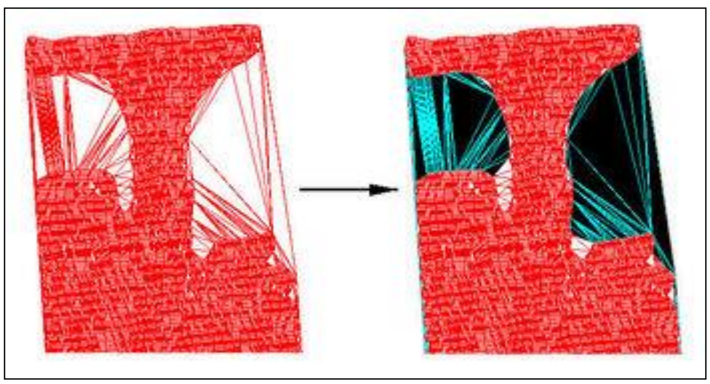
Select Thin Triangles

During the process of triangulation, a mesh of triangular elements is created around existing nodes. This usually creates triangular elements outside the desired scatter set boundary. Many of these exterior triangles are very skinny, and some are virtually invisible. The **Select Thin Triangles** command finds and selects skinny triangular elements which are on the scatter set boundary.

Thin triangles interior to the scatter set will not be selected when this command is performed, since deletion of interior triangles would result in gaps in the mesh. After the thin triangles have been selected, they can be removed by selecting the **Delete** macro.

Select/Delete Long Triangles

This option in the Scatter Module , *Triangles* Menu finds triangles longer than the length specified in the *Scatter Options* dialog. The *Scatter Options* also allows selecting the option to delete or select the long triangles. Selecting/deleting long triangles is useful for deleting triangles that span regions where interpolation is not desired, such as over regions of land (see figure below, the selected triangles are over land).



Process Boundary Triangles

When scatter points are [triangulated, the resulting convex hull often contains triangles outside the desired mesh boundary. The *Process Boundary Triangles* dialog was developed to help remove invalid boundary triangles. To open the *Process Boundary Triangles* dialog:

- In the Scatter Module , make the **Select Triangle** tool active
- Select **Process Boundary Triangles...** from the right-click menu

OR

- In the Scatter Module , select **Process Boundary Triangles...** from the *Triangles* menu

See the article Process Boundary Triangles for more information.

Related Topics

- Scatter Module
- Scatter Module Menus

Scatter Vertices Menu

The *Vertices* menu has the following menu command:

Locked

The points (vertices) in a scatter set can be dragged with the mouse cursor if they are unlocked and the **Select Scatter Points** tool is selected. The **Locked** command in the *Vertices* menu turns on and off the locked status. If scatter points are locked, a check mark is shown next to the menu text. The default status is locked so scatter points are not accidentally moved.

The locked or unlocked status effects all points in the scatter module. One scatter set cannot have locked points and another scatter set have unlocked points. Individual points or groups of points cannot by locked while others points remain unlocked. Therefore, it is best practice to make certain to lock the scatter vertices after moving points in the unlocked status.

If a project in the unlocked status is closed, SMS automatically reverts back to the locked status.

The locked status does not prevent points in a scatter set from being moved using the Edit Window to manually change the precise orientation of a selected point. The locked or unlocked status does not effect the Z value of a selected point. By default, all points are locked when not in plan view .

Related Topics

- Scatter Module
- Scatter Module Menus

3.10.b. Functionalities

Scatter Options

The *Scatter Options* dialog is accessed through the **Scatter Options** command in the *Scatter* menu of the scatter module.

Triangulation Options

This section adjusts the maximum aspect ratio of a thin triangle. The aspect ratio is the ratio of the triangle width to the triangle height. All triangles with an aspect ratio less than what is specified are considered thin.

Long Triangles

This section contains options for deleting or selecting long triangles. The *Max Edge Length* option determines the longest length that can be allowed. Triangles with lengths longer than this will be selected when using the **Select Long Triangles** command.

Individual Points

This section contains the *Retriangulate voids when deleting* option. When scatter points are deleted, the triangles attached to the scatter points (if any) are also deleted. If this option is on, surrounding triangles are retriangulated to fill the void.

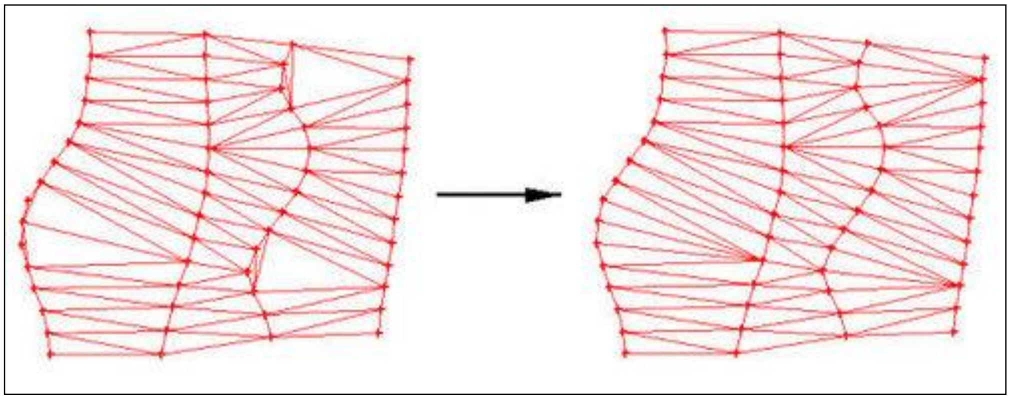
When deleting selected scatter vertices, if the vertex count exceeds ~1000 there will be a noticeable delay if the *Retriagulate voids when deleting* box is checked.

Triangulation Optimization Options

When the optimize triangulation command is invoked, the triangles are optimized in one of two ways:

- *Angle Optimization* – The triangles are swapped to conform to the Delaunay Criterion.

- *Area Optimization* (SHOALS) – The triangles are swapped to align with other triangles. The swapping is done by comparing the area of one triangle to its neighbor. A bias can be set. If the area of the smaller triangle is less than the area of the larger triangle divided by the bias, the triangles are swapped. This is useful for optimizing the triangulation of surveys such as SHOALS surveys.
- *Contour Optimization* – Triangle edges are swapped to minimize sharp angles in scalar contours and define localized topographical features more naturally.



TIN Contour Optimization

Description It is known that a global optimum triangle configuration exists for any given set of scatter points. The qualitative accuracy of that global optimum will vary depending on the density of the available data. Despite its existence, there is no guarantee that a progressive edge-swap process, such as the one used by the contour optimization option, will converge to that global optimum. The likelihood that a global optimum will result varies inversely with the size of the dataset. The contour smoothing algorithm will assuredly converge to a sub-optimum configuration and the smaller the dataset, the more closely the result with approximate the global optimum. As demonstrated by this relationship, the contour smoothing algorithm works very well for discovering local optimums.

Contour Optimization Usage Guidelines

The following are recommended guidelines for getting the most efficacy out of the contour optimization option.

- Break down large datasets into smaller areas for optimization by selecting an area with the triangle selection tool before optimizing. This also has the benefit of decreasing the optimization runtime for the entire dataset.
- Recognize that the more edges a contour deviation spans, the less likely it is that the optimization will correct the problem. It is recommended that large deviations, such as can be easily identified, be minimized by the insertion of appropriately forced breaklines prior to optimization.
- A better optimization can be obtained for large datasets if locally selected optimization areas are chosen so as to overlap each other.
- This progressive edge-swap optimization is sensitive to initial conditions. Often by varying the initial triangulation (Delaunay, Area Bias or hybrid of both) a better optimization can be achieved.

By following these guidelines, the time spent identifying minor changes and swapping edges individually is greatly minimized. Please note that this optimization is iterative and larger data selections will not only require longer runtime but are also more likely to converge to any number of sub-optima dependent on the specific configuration of the triangles. This recommended process of user guided optimization is likely to require some repetition but proves to be much more time efficient.

The following are some of the scenarios under which contour optimization is especially useful.

- Smoothing of channel banks.
- Refining topographical transitions between flat and steep areas.
- Streamlining jagged contours and contour jumps caused by triangulation.
- Connecting thalweg points between cross-sections.

In addition, the following are known limitations of the contour smoothing algorithm.

- Will not define small dams in narrow channels because of the indeterminate nature of the scatter point data.
- Large contour discontinuities and displacements that require the swapping of many edges in a sequential manner, without decreasing the departure angle, before achieving an optimum configuration are unlikely to resolve themselves using this method. It is recommended to insert breaklines to connect the proper points and force them prior to optimizing the area.
- Channel cross sections that are inset into higher resolution scatter sets will result in unacceptable triangulation along the banks if triangulated together. It is recommended to either triangulate and optimize the cross-section scatter set separately and then merge the two together while preserving the cross-section set’s triangulation, or, add and force breaklines connecting the endpoints of the cross sections to better approximate the optimum before attempting to optimize.
- Be aware that large datasets, if optimized in their entirety, are likely to take excessive amounts of time to run. It is recommended to manually select smaller areas of large datasets to run incrementally and work through the dataset in this manner. This method of user guided optimization drastically decreases the runtime required to optimize large datasets and allows bypassing any areas believed to have acceptable triangulation.

Duplicate Vertex Options

When the **Select/Delete Duplicate Points** menu item is selected, points within a tolerance of other points are selected or deleted. Sets the Tolerance and whether to delete or select the points when the command is invoked.

Related Topics

- Scatter Module
- Scatter Module Menus
- Scatter Triangles Menu

Scalar Value Options

The *Scalar Value Options* dialog controls the display options for the scalar values displayed next to each scatter point. The option to display the scalar value of the active scatter function next to each scatter point is turned on and off in the Scatter Module *Display Options* dialog.

The following options are available in the *Scalar Value Options* dialog:

- *Decimal Precision* – The decimal precision used.
- *Alignment* – The location used. The following options are available:
  - On Point
  - Left
  - Center
  - Right
- *Text Options* – Change the color, size, and font for the text.
- *Location Options* – Display the text at:
  - Each point
  - On a grid (works well for dense data). Specify the x and y pixel spacing. Because the spacing is based on pixels, the spacing stays constant during zooming.

Related Topics

- Scatter Module Display Options
- Display Options
- Scatter Module

Use of DEMs in the Scatter Module

The Scatter Point Module is used to visualize and apply various types of data. This data typically comes from surveys, digital maps, previous numerical analysis or digitization on screen. The data is stored as sets or groups of 2D scattered data points with associated values. The most common value is bathymetry and is used to create the geometric representation of the area being modeled.

SMS connects the scattered data points into triangles forming a Triangulated Irregular Network (TIN). TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for a source of bathymetric or other data in a numerical model. TINs can also be used to compute areas, volume, distances, gradients and several other geometric parameters.

SMS applies data from scattered datasets to finite element networks or grids via interpolation. This allows poorly distributed elevation data to be assigned to a well-structured set of elements to create the bathymetry of the entire mesh. A variety of interpolation schemes are supported. Internally, the scattered datasets are triangulated to create surfaces for continuous interpolation. Since the connectivity of the triangulation affects the interpolation, SMS provides tools to allow for the manipulation of this triangulation. The triangulation also allows contouring of the scattered dataset to visualize the data.

Multiple scatter point sets can exist at one time in memory. One of the scatter sets is always designated as the "active" scatter point set. The active scatter set can be changed by changing the *Scatter Set* combo box in the top *Edit Window* . Whenever a new scatter set is created, it becomes the active set.

Related Topics

- Data Acquisition
- TINs

Scatter Breakline Options

When importing a Scatter Set using the *File Import Wizard* , selecting "Breaklines" as a field will open the *Scatter Breakline Options* dialog. Breaklines are useful for maintaining the correct triangulation in a TIN.

Breakline Delimeters

The available breakline delimeter options and samples of the associated file formats are:

- **Names** – Breaklines are identified using a unique name or ID
- **Tags** – Breaklines are identified using "start" and optionally "continue" and "end" values

No vertex can be in more than one breakline, and the breakline must be defined sequentially in the data file.

Named Breaklines

The breakline column of the data file includes the name of the breakline this vertex belongs to. If the breakline column is empty, the associated vertex is not included in any breaklines.

Example Files

Example of a tab delimited file using breakline names:

```
xcoord          ycoord          zcoord      name
215962.9        85203.098       1.483       Breakline1
215957.638      85193.069       1.483       Breakline1
215963.278      85184.35        1.483       Breakline1
215979.111      85179.328       1.483       Breakline1
216056.51       85209.371       1.483       Breakline1
215992.462      85201.477       7.034       Breakline2
216127.386      85264.681       7.034       Breakline2
```

```
216267.187      85327.936      7.034      Breakline2
216371.217      85381.431      7.034      Breakline2
219261.939      90247.944      8.763
219461.211      90220.556      9.167
219678.994      90179.064      9.468
```

Tagged Breaklines

The breakline column of the data file includes a tag or string defining when a breakline starts and stops. There are three types of tags including:

- **Start** – Identifies the start of a new breakline
- **Continue** – Indicates the vertex should be included in the current breakline
- **End** – Identifies the end of the current breakline

These tags may be used in a number of configurations:

- **Start** , **Continue** , **End** – When all three tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter should have a **Continue** tag until the line with the **End** tag. Lines with no entry in the Breakline column between breaklines are read as vertices not belonging to any breakline.
- **Start** , **End** – When the start and end tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter is searched for an **End** tag. All intervening lines are assumed to belong sequentially to a breakline. If two **Start** tags are encountered without an intervening **End** tag, the break line is terminated and another started.
- **Start** , **Continue** – When the start and continue tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter is searched for a **Continue** tag. If any line is encountered without a **Continue** tag, the breakline is terminated and the vertex associated with that line is not included in a breakline.

Example of a tab delimited file using the following breakline tags:

- Start: 1
- Continue: 2
- End: 4
- Not in breakline: 5

```
xcoord          ycoord         zcoord     breakline_tag
215962.9        85203.098      1.483      1
215957.638      85193.069      1.483      2
215963.278      85184.35       1.483      2
215979.111      85179.328      1.483      2
216056.51       85209.371      1.483      4
215992.462      85201.477      7.034      1
216127.386      85264.681      7.034      2
216267.187      85327.936      7.034      2
216371.217      85381.431      7.034      4
219261.939      90247.944      8.763      5
219461.211      90220.556      9.167      5
219678.994      90179.064      9.468      5
```

Related Topics

- File Import Wizard
- Breaklines

Process Boundary Triangles

When scatter points are triangulated , the resulting convex hull often contains triangles outside the desired mesh boundary. The *Process Boundary Triangles* dialog was developed to help remove invalid boundary triangles. To open the *Process Boundary Triangles* dialog:

- In the Scatter Module , make the **Select Triangle**  tool active
- Select **Process Boundary Triangles...** from the mouse right-click menu

OR

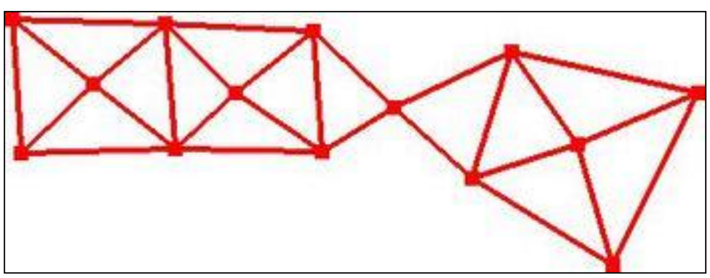
- In the Scatter Module , select **Process Boundary Triangles...** from the *Triangles* menu

Removal Options

- **Aspect Ratio** – A small aspect ratio (sometimes called edge ratio) will result in the removal of more triangles than a large edge ratio. Specifying too small of an aspect ratio will result in the removal of valid triangles. The aspect ratio is calculated by dividing the length of the triangle edge on the boundary by the length of the smallest triangle edge connected to the scatter vertex end points of the triangle edge on the boundary.
- **Select** – Triangles meeting the specified aspect ratio are selected when **OK** is pressed
- **Delete** – Triangles meeting the specified aspect ratio are deleted when **OK** is pressed
- **Preview** – Preview which triangles will be processed based on the specified aspect ratio

Using the Tool / Practical Notes

- Areas on the interior of the triangulation, such as islands, should be manually "seeded" by deleting one of the unneeded triangles in the interior region.
- Triangles will not be removed if their removal would result in:
  - The removal of a scatter vertex from the triangulation
  - The creation of a "bow tie" in the scatter set. A bow tie, a point in the scatter set where the scatter set is "pinched" to a single point, is shown in the image below.



Related Topics

- Select Thin Triangles

Interpolate to Scatter Set

This option interpolates one scattered dataset to another set. Two sets must exist for this option to be enabled. A dialog appears when the scatter set to interpolate from is selected. That scatter set is interpolated to the active scatter set. The **Options** button brings up the *Interpolation Options* dialog, allowing the interpolation type to be set. The interpolation uses an extrapolation value of 0.0.

Related Topics

- Grid Frame Properties

- Scatter Menu
- Scatter Data Menu
- Scatter Triangles Menu

Generate Contour Breaklines

Breaklines in scatter sets can greatly improve the representation of a physical surface. The **Generate Contour Breaklines** tool is used to create breaklines following specified dataset values. This option was designed to work on scatter data that has been digitized to follow features.

The following dialog is brought up by right-clicking on the scatter set and selecting **Generate Contour Breaklines...**

The options in the *Autogenerate Breaklines* dialog are as follows:

- *Min. Scalar Value* – The minimum of the range to be used for autogenerating breaklines (defaulted to the minimum scalar value of the scatter data)
- *Max. Scalar Value* – The maximum of the range to be used for autogenerating breaklines (defaulted to the maximum scalar value of the scatter data)
- *Max. Spacing* – The maximum distance allowed between two adjacent vertices in a breakline
- *Number of Intervals* – The number of dataset intervals from which the breaklines will be created
- *Interval Size* – The size of each interval (this is equal to the entire range; defined by the min. and max. scalar values; divided by the number of intervals)
- *Auto-compute Tolerance* – SMS will automatically compute the tolerance such that all datasets falling within an interval will be included in the breakline generation. The auto-computed tolerance is equal to half of the interval size. This value can be overridden.

SMS creates the breaklines for each contour interval by gathering all vertices in the scatterset that are within the specified tolerance. Starting with the first vertex, SMS will search for the next closest vertex in the scatterset that is also within the specified tolerance. If the closest vertex is within the specified maximum spacing, it becomes the next vertex in the breakline. SMS continues adding vertices to the breakline until it cannot find a vertex within the spacing limit. At this point, SMS will end the breakline and begin to create another breakline using the same method.

Related Topics

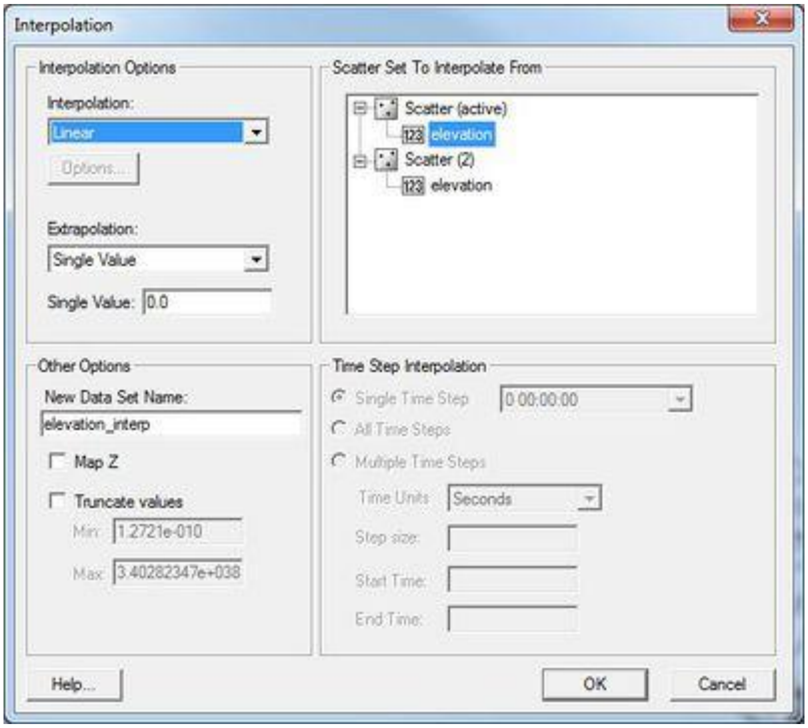
- Breaklines
- Importing Scatter Breaklines

3.10.c. Scatter Interpolation

Scatter Interpolation

Interpolation can be invoked explicitly or implicitly. The implicit invocation is part of the automatic mesh or grid generation. Explicit interpolation occurs when selecting an **Interpolate to _____...** option in the *Scatter* menu in the Scatter Module . These commands require that at least one scatter set exist with at least one function associated. A mesh or grid must also exist in order for the associated interpolation command to be available.

Interpolation Dialog



When selecting any interpolation command, the *Interpolation* dialog appears labeled to indicate what interpolation is being performed. Select the appropriate options and once the **OK** button is selected, the interpolation procedure is performed. Options include:

- *New interpolated dataset name* – The name of the new dataset (or function) created by interpolation.
- *Map Elevation* – For interpolation to mesh nodes, the new dataset is mapped to be the elevation function.
- *Interpolation Method* – Since no interpolation scheme is superior in all cases, SMS supports three interpolation techniques. Many other methods are possible, however, since surface water modeling requires a fairly rich dataset, the more simplistic interpolation methods are more applicable. Select a current method that will be used for all interpolation until another method is selected. The supported methods include:
    1) Linear
    2) Inverse Distance Weighted (IDW)
    3) Natural Neighbor (NN)
    4) Laplacian Interpolation (Cartesian grids only)
- *Default Extrapolation Value* – If the scatter set does not bound the data being interpolated to, the extrapolation value is used (for Linear and NN interpolation only).
- *Existing Dataset Value* – The corresponding value from a specified existing dataset can be used for locations outside of the bounds of the scatter set. The dataset must be from the same object being interpolated to and must be of the same type (i.e. scalar, vector).
- *Truncate Values* – When interpolating a set of values, it is sometimes useful to limit the interpolated values to a specific range. For example, when interpolating contaminant concentrations, a negative value of concentration is meaningless. However, many interpolation schemes will produce negative values even if all of the scatter points have positive data values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by entering a minimum and maximum interpolation value.
- *Scatter Set/Function* – Selects the desired scatter point set and the function to interpolate from.

- *Time Steps* – Selects the option to interpolate a single time step or multiple time steps (if the function is transient).
- *Time Step Interpolation* – If interpolating multiple time steps, select the beginning time, the step size, and the number of time steps to interpolate. It is also possible to interpolate between time steps or to match time steps that fall within the specified time range.

Related Topics

- [Scatter Module]
- [Scatter Menu]

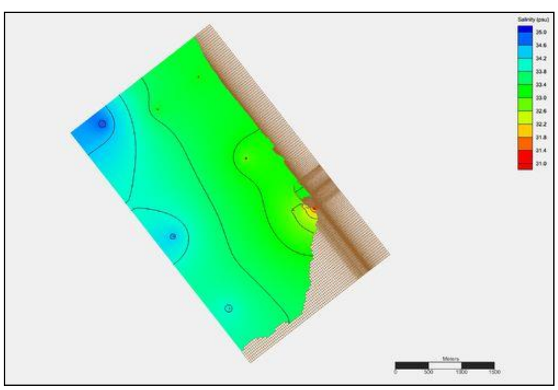
Laplacian Interpolation

Laplacian interpolation is an interpolation/extrapolation approach that tries to give smooth gradients between areas of known values. Laplacian interpolation can only be performed on a cell-centered Cartesian grid.

When performing Laplacian interpolation, the values at each scatter vertex are considered known values and assigned to the Cartesian grid cells that they lie within. We will refer to the known locations/values as seed location/values. The seed values are held constant throughout the process. The algorithm assigns the average seed value to every cell in the grid. Once the initial values have been set, the algorithm repeatedly sweeps the grid smoothing the data values. The result gives the seed values at each of their locations and smooth transitions between the values. The algorithm will also extrapolate outside of the original seed values.

For models that distinguish between wet and dry cells (like CMS-Flow), the Laplacian interpolation will only be performed inside of wet (computational) cells. This gives the often desirable effect that the gradients will follow flow-paths. Cells that are nearby but do not have a water connection, will have a greater difference in value than cells that have a water connection. This effect can be seen in the image below.

The picture below is of the same domain but zoomed out so the full domain can be shown. Notice the smooth transitions between distinct values.



Inverse Distance Weighted Interpolation

One of the most commonly used techniques for interpolation of scatter points is inverse distance weighted (IDW) interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced most by the nearby points and less by the more distant points. The interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases. Several options are available for inverse distance weighted interpolation. The options are selected using the *Inverse Distance Weighted Interpolation Options* dialog. This dialog is accessed through the **Options** button next to the "Inverse distance weighted" item in the *Interpolation* dialog. SMS uses Shepard's Method for IDW:

Shepard's Method

The simplest form of inverse distance weighted interpolation is sometimes called "Shepard's method" (Shepard 1968). The equation used is as follows:

$$F(x,y) = \sum_{i=2}^{n} w_i f_i$$

where *n*< is the number of scatter points in the set, f_i are the prescribed function values at the scatter points (e.g. the dataset values), and w_i are the weight functions assigned to each scatter point. The classical form of the weight function is:

$$w_i = \frac{h_i^{-p}}{\sum_{j=i}^{n} h_j^{-p}}$$

where p is an arbitrary positive real number called the power parameter (typically, $p=2$) and hi is the distance from the scatter point to the interpolation point or

$$h_i = \sqrt{x - x_i)^2 + (y - y_i)^2}$$

where (x,y) are the coordinates of the interpolation point and (x_i, y_i) are the coordinates of each scatter point. The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases. The weight functions are normalized so that the weights sum to unity.

The effect of the weight function is that the surface interpolates each scatter point and is influenced most strongly between scatter points by the points closest to the point being interpolated.

Although the weight function shown above is the classical form of the weight function in inverse distance weighted interpolation, the following equation is used in SMS:

$$w_i = \frac{\left[\frac{R-h_i}{Rh_i}\right]^2}{\sum_{j=i}^{n} \left[\frac{R-h_j}{Rh_j}\right]^2}$$

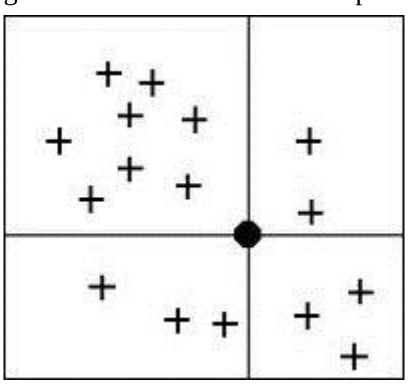
Where h_i is the distance from the interpolation point to scatter point i , R is the distance from the interpolation point to the most distant scatter point, and n is the total number of scatter points. This equation has been found to give superior results to the classical equation (Franke & Nielson, 1980).

The weight function is a function of Euclidean distance and is radially symmetric about each scatter point. As a result, the interpolating surface is somewhat symmetric about each point and tends toward the mean value of the scatter points between the scatter points. Shepard's method has been used extensively because of its simplicity.

Computation of Nodal Function Coefficients

In the *IDW Interpolation Options* dialog, an option is available for using a subset of the scatter points (as opposed to all of the available scatter points) in the computation of the nodal function coefficients and in the computation of the interpolation weights. Using a subset of the scatter points drops distant points from consideration since they are unlikely to have a large influence on the nodal function or on the interpolation weights. In addition, using a subset can speed up the computations since less points are involved.

If the *Use subset of points* option is chosen, the **Subsets** button can be used to bring up the *Subset Definition* dialog. Two options are available for defining which points are included in the subset. In one case, only the nearest *N* points are used. In the other case, only the nearest *N* points in each quadrant are used as shown below. This approach may give better results if the scatter points tend to be clustered.



If a subset of the scatter point set is being used for interpolation, a scheme must be used to find the nearest *N* points. Two methods for finding a subset are provided in the *Subset Definition* dialog: the global method and the local method.

Global Method

With the global method, each of the scatter points in the set are searched for each interpolation point to determine which *N* points are nearest the interpolation point. This technique is fast for small scatter point sets but may be slow for large sets.

Local Method

With the local methods, the scatter points are triangulated to form a temporary TIN before the interpolation process begins. To compute the nearest *N* points, the triangle containing the interpolation point is found and the triangle topology is then used to sweep out from the interpolation point in a systematic fashion until the *N* nearest points are found. The local scheme is typically much faster than the global scheme for large scatter point sets.

Computation of Interpolation Weights

When computing the interpolation weights, three options are available for determining which points are included in the subset of points used to compute the weights and perform the interpolation: subset, all points, and enclosing triangle.

Subset of Points

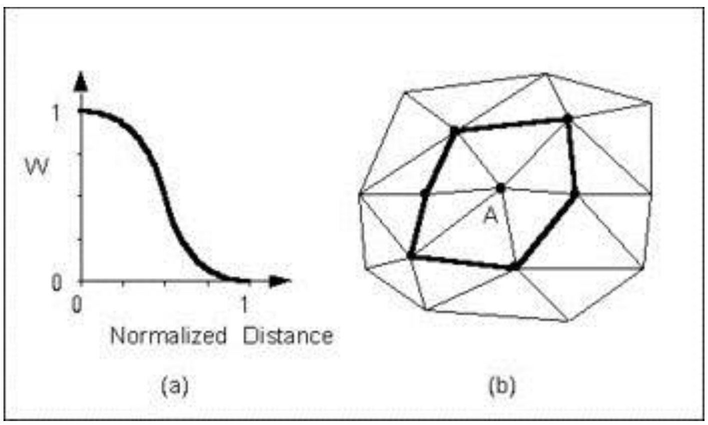
If the *Use subset of points* option is chosen, the *Subset Definition* dialog can be used to define a local subset of points.

All Points

If the *Use all points* option is chosen, a weight is computed for each point and all points are used in the interpolation.

Enclosing Triangle

The Use vertices of enclosing triangle method makes the interpolation process a local scheme by taking advantage of TIN topology (Franke & Nielson, 1980). With this technique, the subset of points used for interpolation consists of the three vertices of the triangle containing the interpolation point. The weight function or blending function assigned to each scatter point is a cubic S-shaped function as shown in part a of the figure below. The fact that the slope of the weight function tends to unity at its limits ensures that the slope of the interpolating surface is continuous across triangle boundaries.

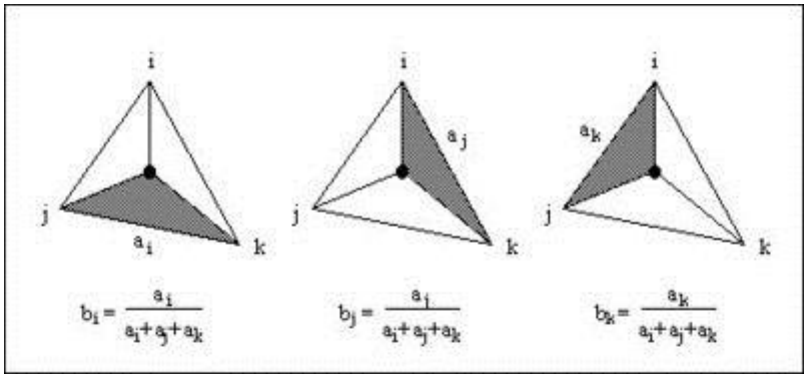


The influence of the weight function extends over the limits of the Delauney point group of the scatter point. The Delauney point group is the "natural neighbors" of the scatter point, and the perimeter of the group is made up of the outer edges of the triangles that are connected to the scatter point as shown in part b. The weight function varies from a weight of unity at the scatter point to zero at the perimeter of the group. For every interpolation point in the interior of a triangle there are three nonzero weight functions (the weight functions of the three vertices of the triangle). For a triangle T with vertices i, j, & k, the weights for each vertex are determined as follows:

$$w_i(x,y) = b_i^2(3-2b_i) + 3\frac{b_i^2 b_j b_k}{b_i b_j + b_i b_k + b_j b_k}$$

$$\left\{ b_j \left[\frac{|e_i|^2 + |e_k|^2 - |e_j|^2}{|e_k|^2} \right] + b_k \left[\frac{|e_i|^2 + |e_j|^2 - |e_k|^2}{|e_j|^2} \right] \right\}$$

where ||ei|| is the length of the edge opposite vertex i, and b_i, b_j, b_k are the area coordinates of the point (x, y) with respect to triangle T. Area coordinates are coordinates that describe the position of a point within the interior of a triangle relative to the vertices of the triangle. The coordinates are based solely on the geometry of the triangle. Area coordinates are sometimes called "barycentric coordinates." The relative magnitude of the coordinates corresponds to area ratios as shown below:



The XY coordinates of the interior point can be written in terms of the XY coordinates of the vertices using the area coordinates as follows:

$$x = b_i x_i + b_j x_j + b_k x_k$$

$$y = b_i y_i + b_j y_j + b_k y_k$$

$$1.0 = b_i + b_j + b_k$$

Solving the above equations for b_i, b_j, and b_k yields:

$$b_i \frac{1}{2A}[(x_j y_k - x_k y_j) + (y_j - y_k)x + (x_k - x_j)y]$$

$$b_j \frac{1}{2A}[(x_k y_i - x_i y_k) + (y_k - y_i)x + (x_i - x_k)y]$$

$$b_k \frac{1}{2A}[(x_i y_j - x_j y_i) + (y_i - y_j)x + (x_j - x_i)y]$$

$$A = \frac{1}{2}(x_i y_j + x_j y_k + x_k y_i - y_i x_j - y_j x_k - y_k x_i)$$

Using the weight functions defined above, the interpolating surface at points inside a triangle is computed as:

$$F(x, y) = w_i(x, y)Q_i(x, y) + w_j(x, y)Q_j(x, y) + w_k(x, y)Q_k(x, y)$$

where w_i, w_j, and w_k are the weight functions and Q_i, Q_j, and Q_k are the nodal functions for the three vertices of the triangle.

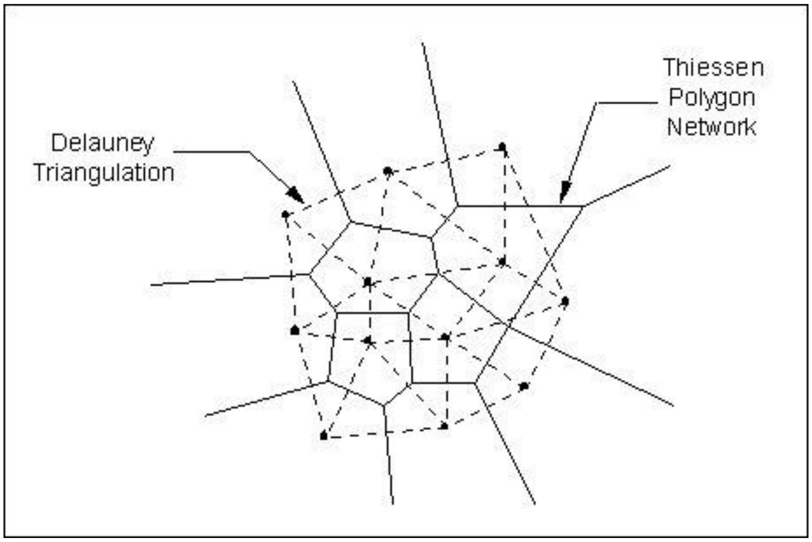
Related Topics

Scatter Interpolation

Natural Neighbor Interpolation

Natural neighbor interpolation is also supported in SMS. Natural neighbor interpolation has many positive features. It can be used for both interpolation and extrapolation and it generally works well with clustered scatter points. Natural neighbor interpolation was first introduced by Sibson (1981). A more detailed description of natural neighbor interpolation in multiple dimensions can be found in Owen (1992).

The basic equation used in natural neighbor interpolation is identical to the one used in IDW interpolation:



SMS uses nodal functions with IDW. The nodal function can be selected using the Natural Neighbor Interpolation Options dialog. The difference between IDW interpolation and natural neighbor interpolation is the method used to compute the weights and the method used to select the subset of scatter points used for interpolation.

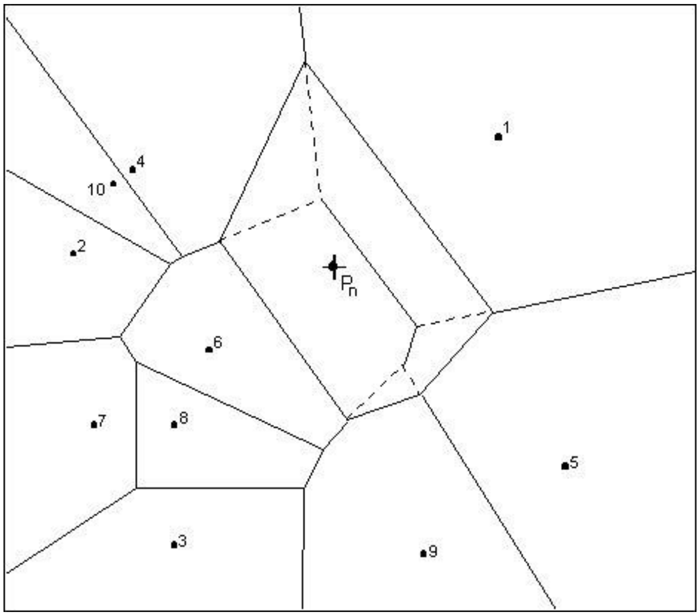
Natural neighbor interpolation is based on the Thiessen polygon network of the scatter point set. The Thiessen polygon network can be constructed from the Delauney triangulation of a scatter point set. A Delauney triangulation is a TIN that has been constructed so that the Delauney criterion has been satisfied.

There is one Thiessen polygon in the network for each scatter point. The polygon encloses the area that is closer to the enclosed scatter point than any other scatter point. The polygons in the interior of the scatter point set are closed polygons and the polygons on the convex hull of the set are open polygons.

Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points. The vertices of the Thiessen polygons correspond to the centroids of the circumcircles of the triangles.

Local Coordinates

The weights used in natural neighbor interpolation are based on the concept of local coordinates. Local coordinates define the "neighborliness" or amount of influence any scatter point will have on the computed value at the interpolation point. This neighborliness is entirely dependent on the area of influence of the Thiessen polygons of the surrounding scatter points.



To define the local coordinates for the interpolation point, P_n, the area of all Thiessen polygons in the network must be known. Temporarily inserting P_n into the TIN causes the TIN and the corresponding Thiessen network to change, resulting in new Thiessen areas for the polygons in the neighborhood of P_n.

The concept of local coordinates is shown graphically in the following figure. Points 1–10 are scatter points and P_n is a point where some value associated with points 1–10 is to be interpolated. The dashed lines show the edges of the Thiessen network before P_n is temporarily inserted into the TIN and the solid lines show the edges of the Thiessen network after is P_n inserted.

Only those scatter points whose Thiessen polygons have been altered by the temporary insertion of P_n are included in the subset of scatter points used to interpolate a value at P_n. In this case, only points 1, 4, 5, 6, & 9 are used. The local coordinate for each of these points with respect to P_n is defined as the area shared by the Thiessen polygon defined by point P_n and the Thiessen polygon defined by each point before point P_n is added. The greater the common area, the larger the resulting local coordinate, and the larger the influence or weight the scatter point has on the interpolated value at P_n.

If we define $K(n)$ as the Thiessen polygon area of P_n and $K_m(n)$ as the difference in the Thiessen polygon area of a neighboring scatter point, P_m, before and after P_n is inserted, then the local coordinate $\lambda_m(n)$ is defined as:

$$\lambda_m(n) = \frac{K_m(n)}{K(n)}$$

The local coordinate $\lambda_m(n)$ varies between zero and unity and is directly used as the weight, $w_m(n)$, in the interpolation equation. If P_n is at precisely the same location as P_m, then the Thiessen polygon areas for P_n and P_m are identical and $\lambda_m(n)$ has a value of unity. In general, the greater the relative distance P_m is from P_n, the smaller its influence on the final interpolated value.

Related Topics

- Scatter Interpolation

Linear Interpolation

If the linear interpolation scheme is selected, the 2D scatter points are first triangulated to form a temporary TIN. If the surface is assumed to vary linearly across each triangle, the TIN describes a piecewise linear surface which interpolates the scatter points. The equation of the plane defined by the three vertices of a triangle is as follows:

$$Ax + By + Cz + D = 0$$

where *A* , *B* , and *C* >, and *D* are computed from the coordinates of the three vertices (x_1, y_1, z_1), (x_2, y_2, z_2), and (x_3, y_3, z_3):

$$A = y_1(z_2 - z_3) + y_2(z_3 - z_1) + y_3(z_1 - z_2)$$
$$B = z_1(x_2 - x_3) + z_2(x_3 - x_1) + z_3(x_1 - x_2)$$
$$C = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$$
$$D = -Ax_1 - By_1 - Cz_1$$

The plane equation can also be written as:

$$z = f(x, y) = -\frac{A}{C}x - \frac{B}{C}y - \frac{D}{C}$$

which is the form of the plane equation used to compute the elevation at any point on the triangle.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the linear interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the *Interpolation Options* dialog.

Related Topics

- Scatter Interpolation

4. General Numeric Models

SMS Models

SMS provides pre- and post- processing for several numeric models. These models are developed and maintained by government or commercial entities rather than the developers of SMS. A comparison chart of the *CIRP Numerical Model Tools and Capabilities* (BOUSS-2D, CMS-Flow, CMS-Wave, GenCade, and PTM) can be found in the 2012 CIRP Brochure .

Hydraulic Models

- ADCIRC (ADvanced CIRCulation Model) – Widely applied coastal circulation and coastal flooding model. Developed commercially.
- Coastal Modeling System CMS-FLOW – Suite of models that simulates a wide variety of coastal processes. Developed and maintained by the USACE.
- FESWMS – Developed in cooperation with the Federal Highway Administration (FHWA).
- Accessed through the generic model interface in cooperation with Aquaveo
    - RiverFlow2D – Commercially developed finite element model to route floods. Provides high resolution flood hydraulics.

    - HYDRO AS-2D – Developed commercially in Germany.
    - TUFLOW FV – Finite volume model developed by the makers of TUFLOW (WBM).
- HEC-RAS – Developed at the Hydrologic Engineering Center for the U.S. Army Corps of Engineers.
- SRH-2D – Developed at the United States Bureau of Reclamation.
- TABS – Suite of models for solving hydrodynamics and transport problems maintained by the USACE including RMA2 and RMA4
- TUFLOW – Finite difference model featuring combined 1D/2D models, very stable wetting drying, and advanced simulation management for Coastal, Riverine or Urban applications.

Wave Models

Harbor Design

- BOUSS-2D – Wave climate, circulation from waves, and sea state using the Boussinesque equations
- CGWAVE – Phase resolving wave reflection and refraction analysis

Wave Generation and Transformation

- STWAVE – Finite difference spectral wave energy model
- CMS-Wave (WABED)

Other Models

- Generic – Rather than a specific model interface, the "Generic" model interface in the mesh module is a collection of interface objects that can be configured to generate specific types of data. The developers at Aquaveo work with some engine developers to utilize this tool. These engines are included in the list above. Other model developers are encouraged to contact Aquaveo for assistance in using these tools. The tools of the generic model interface can be utilized without coordination with Aquaveo. There are partial interfaces with the following:
    - FVCOM
    - HYDRO AS-2D
    - TUFLOW FV
- PTM – Lagrangian Particle Tracking Model which tracks sediment particles based upon input hydrodynamics and wave effects.

Hydraulic Models (Feature Comparison)

Model	Riverine	Tidal Forcing	Wave Forcing	1D	2D	Hydraulic Structures	Sediment Transport	Advection / Dispersion
ADCIRC	no	yes	yes	no	yes	yes	no	no
CMS-Flow	no	yes	yes	no	yes	no	yes	no
FESWMS	yes	no	no	no	yes	yes	yes	no
TABS	yes	yes	no	yes	yes	yes	no	yes
TUFLOW	yes	yes	no	yes	yes	yes	no	no

Model Linkages / Steering

- General Steering
- RMA2 Spindown

- FESWMS Spindown
- CMS-Flow / CMS-Wave Steering

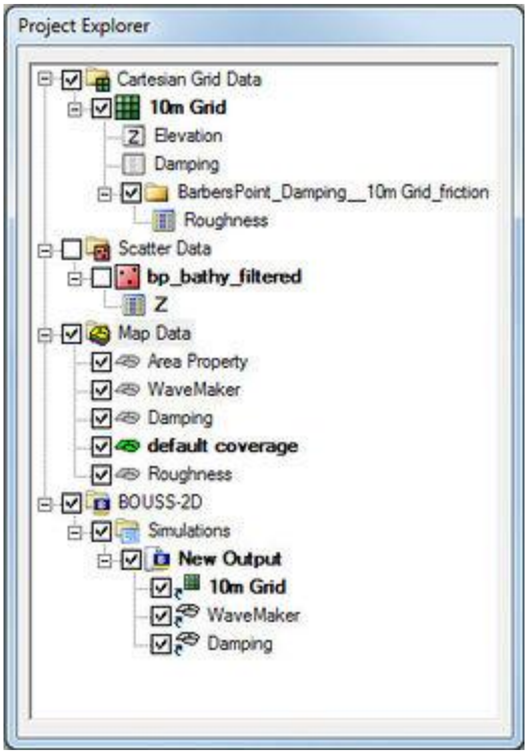
Model versions

Model	**SMS v12.2**	**SMS v12.1**	**SMS v12.0**	**SMS v11.2**	**SMS v11.1**	**SMS v11.0**	**SMS v10.1**
ADCIRC	52.30	51.33	50.99	50.99	50.99	49.82	48.46
ADH	4.5	4.5	4.4	4.4	4.3		N/A
BOUSS-2D							
CGWAVE							2005
CMS-Flow	4.01.52	4.01.52	Not Available	4.02.00	3.75.07	3.75.05	3.75.02
CMS-Wave	3.2	3.2	Not Available	3.2	3.2	3.2	2.5
FESWMS	3.3.2	3.3.2	3.3.2	3.3.2	3.3.2	3.3.2	3.3.2
GenCADE	v1r6	v1r6				N/A	N/A
HEC-RAS	5.0						
Hydro AS-2D	V3 & V4	V3 & V4	V3 & V4	V3 & V4	V3	V3	V2
PTM	2.1.027	2.1.027	2.1.027	2.1.027	2.1.027	2.0.064	2.0.053
SRH-2D	3.0	3.0	3.0	3.0	2.2	2.0	2.0
STWAVE	6.1	6.1	6.1	6.1	6.1	6.0	5.6
TABS-RMA2	4.58	4.58	4.58	4.58	4.58	4.58	4.58
TABS-RMA4	4.56	4.56	4.56	4.56	4.56	4.56	4.56
TUFLOW	2016-03-AD	2016-03-AB	2013-12-AC	2013-12-AC	2012-05-AE	2011-09-AF	2009-07-AC
WAM							N/A

Related Topics

- SMS Main page
- Simulations
- Model Checker

Simulations



Starting in version 11.1, SMS has the capacity to run certain models as a simulation. A simulation includes the domain, boundary conditions, model parameters, event definition, and material set used for a single model run. Multiple simulations can exist within the same project in order to compare alternatives.

Working with Simulations

A simulation can be created by right-clicking on a blank portion of the Project Explorer and choosing the *New Simulation* sub-menu. The *New Simulation* menu will have commands to create a new model simulation. Selecting a model will cause a simulation folder for the chosen model to appear in the Project Explorer.

After a simulation has been created, components may be added to the simulation. Components are often created as map coverages, meshes, grids, or scattersets. The individual map coverages, grids, or scattersets can be linked to the simulation by clicking on the item in the Project Explorer and dragging it under the simulation item in the Project Explorer. This allows multiple simulations to make use of the same datasets.

Components can also be added or removed from a simulation using the right-click *Link To* or *Unlink From* submenus.

Once a simulation exists future simulations are often similar to one already created. Often it is easier to make a copy of the simulation and make changes where appropriate. A simulation can be copied by right-clicking on the simulation and choosing **Duplicate** . All simulations for a model will be listed under the simulation for that model. Under the simulation data item all simulations for all models will be listed—SMS supports using more than one model for a project.

The right-click menu for a simulation includes menu commands specific to each model.

General Simulation Right-Click Menu Commands

Right-clicking on any simulation object in the Project Explorer will bring up a menu for that simulation. Most of the commands are specific for the simulation model type. The simulation right-click menu has the following commands that are used by all simulations.

- **Delete** – Removes the selected simulation. This commands does not delete or modify components linked to the simulation.
- **Duplicate** – Creates a copy of the selected simulation. All components linked to selected simulation will be linked to the duplicate simulation. Adding, or removing the components in the duplicate simulation will not change the original simulation.
- **Rename** – Allows changing the simulation name.

Right-clicking on the simulation folder has the following commands:

- **New Simulation** – This command will create a new simulation of the same model type as the simulation folder.
- **Export All** – Export files for all simulations under the model simulation folder.

Models That Use Simulations

Not all models usable in SMS can be created in the simulation format. Models that currently make use of the simulation format are:

Model	Initial Version	Model Simulation Specifics
Bouss-2D	11.2	See BOUSS-2D Simulations
BOUSS Runup/Overtopping	11.1	
CMS-Flow	12.1	See CMS-Flow Simulation
EFDC	11.1	
HEC-RAS	12.2	See HEC-RAS Simulation
SRH-2D	12.0	See SRH-2D Simulation
TUFLOW	11.1	See TUFLOW Simulation
WAM	11.1	See WAM Simulation Model Control

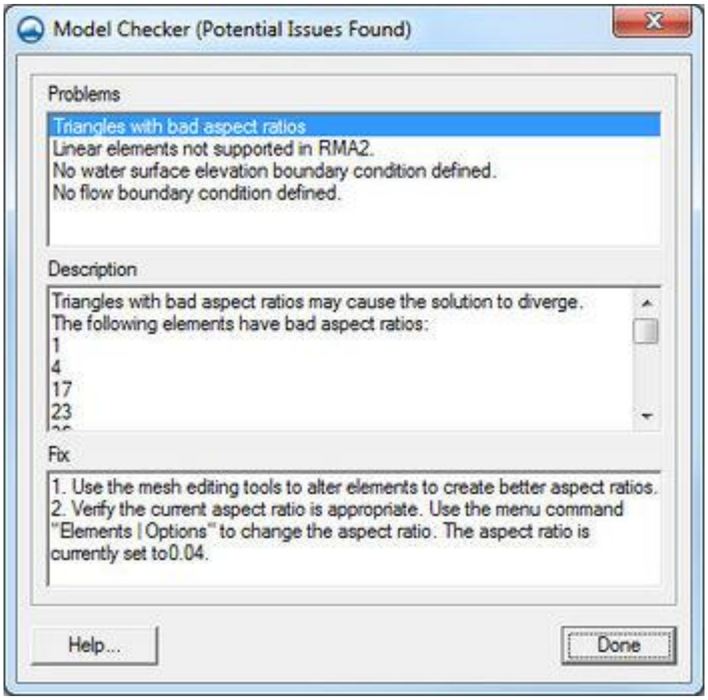
Related Topics

- SMS Models

Model Checker

Once a mesh, grid, or scatter is generated and all of the analysis options and boundary conditions have been specified, the next step is to save the simulation to disk and run the model. However, before saving the simulation and running the model, the model should be checked with the *Model Checker* . Because of the significant amount of data required for a simulation for all the different models, it is often easy to neglect important data or to define inconsistent or incompatible options and parameters. Such errors will either cause the model to crash or to generate an erroneous solution. The purpose of the *Model Checker* is to analyze the input data currently defined for a model simulation and report any obvious errors or potential problems. Running the *Model Checker* successfully does not guarantee that a solution will be correct. It simply serves as an initial check on the input data and can save a considerable amount of time that would otherwise be lost tracking down input errors.

Running the Model Checker



To run the model checker, select the **Model Check** command from the "current model" menu or from the menu accessed by right-clicking on the simulation in the Project Explorer. If there are no errors in the model, a message will appear stating "No model checks violated". If there are potential errors in the model, the *Model Checker* dialog will appear. This generates a list of possible errors.

If the model checker finds any potential errors with the input, then fix the errors and rerun the model checker.

The Model Checker Dialog

If potential issues are found, the *Model Checker* dialog will attempt to help in resolving the issues.

Problems

This field will contain a list of potential problems. The listed problems are specific to each module and model. Each problem listed will need to be addressed and fixed before the model can run successfully. Selecting an item in the list will generate a description of the problem and a suggested solution in the fields below.

Description

This field gives a more detailed explanation of the selected problem in the *Problems* field.

Fix

SMS will attempt to provide a solution for a selected problem in the *Problems* field. Following the suggested steps should resolve the problem. If following the steps does not resolve the issue, contact technical support .

Related Topics

- SMS Models
- Support

4.1. Generic Model

Generic Model

While SMS contains several custom interfaces for specific numerical models, it is not possible to build interfaces in SMS for every model in existance. SMS has a generic interface whereby any two-dimensional finite element or finite difference model can be run using SMS as a pre- and post-processor.

The generic model interface can be added to a paid edition of SMS.

Functionality

The following can be defined for a generic model:

- Global Model Parameters
- Boundary Conditions
- Material Properties

With the model attributes defined, generate a mesh or grid using the geometric tools in the Map, and Mesh or Cartesian Grid Modules. Model parameters can then be set, boundary conditions specified, and material properties applied to the geometric data and saved out to a generic file format.

Using Multiple Meshes

Starting in SMS 11.2, multiple generic mesh model definitions can exist at the same time. When creating a new generic 2D mesh coverage, if multiple generic model definitions exist, a dialog will appear to asking which model definition to use. When performing a map to 2D mesh operation, the definition and values will be copied from the coverage to the new mesh.

The define model menu item will define the definition for the mesh, if the menu item is in the mesh module menu, or for the coverage, if in the feature objects menu. The generic model definition, or template, is stored in a *.2dm file with the SMS project. The *.2dm file will not contain a mesh or parameter values. This template will be used whenever a new mesh or coverage of that type is created.

Graphical Interface

SMS provides a graphical interface that is designed to visualize the projects being created, easily modify project parameters, and view the solutions produced by the Generic Model. See Generic Model Graphical Interface for more information.

The Generic Model Graphical Interface contains tools to create and edit a Generic Model simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to Generic. If a mesh has already been created for a Generic Model simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to Generic. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the generic model *Mesh* menu. See Generic Model Graphical Interface for more information.

Generic Model Menu

The following menu commands are available in the *Mesh* (generic model) menu:

Check Mesh

This command will bring up the *Model Checker* dialog. The dialog will list all potential problems with a description of the problem and a recommend procedure to fix the problem.

Define Model

Opens the *Define Model* dialog.

Global Parameters...

Brings up the *Global Parameters* dialog.

Assign BC...

Opens the *Node Boundary Conditions* , *Nodestring Boundary Conditions* , or *Element Boundary Conditions* dialog depending if a node, nodestring, or element is selected.

Set Active Material Group

Opens the *Active Material Group* dialog. Here indicate which material group is active from the parameter groups created in the *Global Parameters* definition dialog.

Material Properties...

Opens the *Material Properties* dialog.

Run Mesh

Launches the generic mesh model.

Case Studies / Sample Problems

The following tutorials may be helpful for learning to use the Generic Model Interface in SMS:

- General Section
  - *Mesh Editing* [72]
  - *Observation* [73]
- Models Section
  - *Generic Mesh Model* [74]

Related Topics

- BASEMENT
- HYDRO AS-2D
- ELCIRC
- FVCOM
- TUFLOW FV

Generic Model Files

SMS stores all of the Generic Model input parameters in a 2D mesh file (*.2dm) . Since this stores the parameters, the file can be used as a template for multiple projects. It is not necessary for the generic model to include feature objects, though if feature objects have been created they will be saved in the 2D mesh file.

This 2D mesh file includes:

- **The mesh** – Identifies the file as a 2d mesh file.
  - **Node locations** – Defines the ID and location for each node of the mesh.
    - **Element type** – Identifies the type of element. Options are:
    - **Triangle or quadrilateral**
    - **Linear or quadratic**

- **Element connectivity**

- **Global parameters** – Sets the model name, units, state, time units, time steps, and other parameters. Also manages parameter groups and dependencies.
  - **Global parameter assignmnet**
- **Boundary conditions** – Defines boundary conditions display options, values, dependencies, defaults, parameter group assignment, etc.... Options include:
  - **Nodal boundary conditions**
  - **Nodestring boundary conditions**
  - **Element boundary conditions**
- **Material properties** – Defines material values, defaults, dependencies, and parameters.

For more information, see the 2D Mesh File (*.2dm) article.

Related Topics

- HYDRO AS-2D
- SRH-2D

Generic Model Graphical Interface

The Generic Model Graphical Interface includes tools to assist with creating, editing and debugging a Generic Model. The Generic Model interface exists in the Mesh Module .

Define Model

The *Define Model* dialog is used to customize and define the model interface parameters that define various states and characteristics of a model. These model parameters may include items such as those needed to describe flow, channel roughness, and control structures. The parameters, names, and ranges can be created and customized.

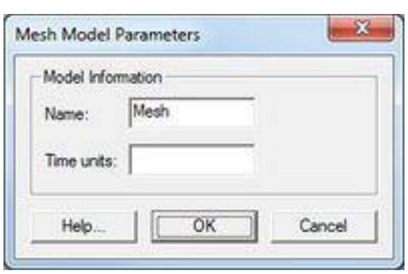
Parameters are organized into groups and are given suitable value ranges depending on the purposes of the model. Proper organization of parameters will increase the abilities of SMS as an interface.

Using the *Define Model* dialog the model interface can be renamed. The *Define Model* dialog can be accessed when the Mesh Module is the active module.

The *Define Model* dialog is used to setup the options that apply to the simulation as a whole.

- Model Parameters
- Global Parameters
- Boundary Conditions
- Material Properties
- Lock Model Definition with Key

Model Parameters



Global Parameters

This dialog creates user-defined parameter groups to be used in the project. Each parameter group must be named. Definitions for the group can be set by clicking on the **Define** button after naming the parameter group. See the Dependencies section for more information.

Boundary Conditions

All numeric models require boundary condition (BC) data. Generic Mesh Model boundary conditions can be defined on nodestrings , nodes , and elements . An entity (nodestring, node, or element) may have multiple boundary conditions set. To add or remove a boundary condition, select a node, nodestring, or element. Then right-click and select **Assign BC** . A dialog with a tree item appears listing all possible boundary conditions. Checked tree items are active boundary conditions. Toggle the tree items as desired to activate/deactivate boundary conditions. The settings in *Display* | **Display Options** will determine how the boundary conditions are displayed.

The model developer can define boundary conditions as constant or as dynamic. This is stored in the BD card as the next to last field. It's also possible to assign multiple boundary conditions to nodes, nodestrings, and elements. To add or remove a boundary condition, select a node, nodestring, or element. Then right-click and select **Assign BC** . A dialog with a tree item appears listing all possible boundary conditions. Checked tree items are assigned. Toggle the tree items as desired to assign/unassign boundary conditions. The settings in *Display* | **Display Options** will determine how the boundary conditions are displayed.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

Material Parameter Groups are the same as the Global Parameter Groups. To add a new Material Parameter Group, add it in the Global Parameter Group by *Gen2DM* | *Define Model* | **Global Parameters** .

Chooses whether to have a single material group or multiple (1 for each group) by *Gen2DM* | *Define Model* | **Material Properties** then select or unselect *Have a seperate material assignment for each parameter group* .

- Multiple
    - Changes the active group by *Gen2DM* | **Set Active Material Group** .
    - Changing the "active material group" effects the display and assignment of materials.
    - In the *Gen2DM* | **Material Properties** dialog, only the active/assigned materials are displayed.
- Single
    - *Set Active Material Group* menu is hidden under the *Gen2DM* menu
    - In the *Gen2DM* | **Material Properties** dialog, all materials are displayed. The tabs across the top correspond to the Groups.

Dependencies

The generic model interface lets a generic model designer create a custom interface by setting up input parameters for their model.

The parameters may be global parameters, bc parameters (applied to node, element, or nodestring), or material parameters.

It is often useful to have certain parameters displayed only in some situations.

For example, the numeric engine may support manning values by depth or a single manning value for all depths. If choosing to use manning values by depth, provide a curve for manning values based upon depth. If choosing to do a single manning value, provide a single manning value.

This manning example could be expanded another level. Suppose the manning value mentioned above was part of the material properties for each material. Also, suppose that the engine supported chezy as well as manning to represent roughness but this had to be applied at a global level. If the global parameter is chezy, the *material properties* dialog should only show the option to enter a chezy value. If the global parameter is manning, the *material properties* dialog will allow choosing whether to provide this using a single value or a curve by depth. The controls would work as described in the preceding paragraph.

The generic model designer can use dependencies to accomplish both situations above. Dependencies show/hide parameters based upon the setting of a parent parameter. The parent parameter must be before the child parameter. If the child parameter is a material or boundary condition parameter, the parent can also be a global parameter.

Dependencies are controlled on the child level when defining the model. To assign a dependency the parent and child parameters must exist and the parent must have its options defined. Dependencies are specified by clicking on the **None** button in the dependency column for the child parameter. Specify the parent desired to be used for this parameter, whether a global parameter or local (whatever level is currently being defined). Then check the boxes next to the parent parameters that will allow the child parameter to be visible. A child option may be visible for several parent options.

Whenever a parent object is invisible all children that are dependent upon the option are also invisible.

Generic 2D Mesh Arc Attributes Dialog

The Generic 2D Mesh *Feature Arc Attributes* dialog is used to set the attributes for feature arcs . Attributes that can be specified for each feature arc include:

- Arc Type
  - *None*
  - *Boundary Conditions* – **Options** button opens the *Mesh Nodestring Boundary Conditions* dialog.

Mesh Nodestring Boundary Conditions Dialog

Generic 2D Mesh Boundary conditions are generally defined on feature arcs in the conceptual model or nodestrings on the 2D mesh. Boundary conditions constrain the water surface elevation and/or flow at the model boundary. The options available will be based on the current generic model in use and are defined in the *Define Model* dialog.

Generic 2D Nodal BC, Nodestring and Element Display Options

The properties of all Gen2DM (Generic 2D Mesh) node boundary conditions, nodestrings, and elements that SMS displays on the screen can be controlled here. This window is accessible only if the current 2D Mesh model is Generic (Mesh module's *Data* | **Switch Current Model…** menu command). Open *Display Options* , turn on either *Nodal BC* , *Nodestrings* , or *Elements* and then select the corresponding **Options** button.

Display options are available for all boundary condition entities defined in the Gen2DM model definition. If no BC entities have been defined then a message will state that fact and display options will not be shown. Each defined BC entity has a checkbox to toggle the display of the item and an **Attribute Options** button to adjust style and color.

Below the list of BC entities is the display options for Inactive entities. Inactive entities are BC entities that are associated with an inactive parameter group (activation is controlled via the *Gen2DM* | **Global Parameters** menu item).

The BC entities can be displayed with labels beside them. *Labels* allows for adjusting the label font, style, size, color and whether they are displayed. When *Labels* is checked for display, the auxiliary *Show BC Values* in labels is available. If checked, BC entities' values will be displayed following the entities' description.

All on checks all BC entity display options.

All off unchecks all BC entity display options.

Overview

The designer/user can define whether the curves in their model are interpolated to the time step duration or not.

Saving the Model

The Generic Model Files are written automatically with the SMS project file or can be saved separately using the *File* | **Save Mesh** or *File* | **Save As** menu commands. See Generic Model Files for more information on the files used for the Generic Model run.

Related Topics

- Mesh Module
- Generic Mesh Model

4.2. Finite Volume Coastal Ocean Model (FVCOM)

FVCOM

FVCOM	
Model Info	
Model type	Prognostic, unstructured-grid, finite-volume, free-surface, 3-D primitive equation coastal ocean circulation model
Developer	Changsheng Chen University of Massachusetts-Dartmouth
Web site	FVCOM web site
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

As stated on the FVCOM website (http://fvcom.smast.umassd.edu/fvcom/):

The Unstructured Grid **F** inite **V** olume **C** oastal **O** cean **M** odel (FVCOM) is a prognostic, unstructured-grid, finite-volume, free-surface, 3-D primitive equation coastal ocean circulation model developed by UMASSD-WHOI joint efforts. The model consists of momentum, continuity, temperature, salinity and density equations and is closed physically and mathematically using turbulence closure submodels. The horizontal grid is comprised of unstructured triangular cells and the irregular bottom is preseented using generalized terrain-following coordinates. The General Ocean Turbulent Model (GOTM) developed by Burchard's research group in Germany (Burchard, 2002) has been added to FVCOM to provide optional vertical turbulent closure schemes. FVCOM is solved numerically by a second-order accurate discrete flux calculation in the integral form of the governing equations over an unstructured triangular grid. This approach combines the best features of finite-element methods (grid flexibility) and finite-difference methods (numerical efficiency and code simplicity) and provides a much better numerical representation of both local and global momentum, mass, salt, heat, and tracer conservation. The ability of FVCOM to accurately solve scalar conservation equations in addition to the topological flexibility provided by unstructured meshes and the simplicity of the coding structure has make FVCOM ideally suited for many coastal and interdisciplinary scientific applications.

FVCOM was originally developed for the estuarine flooding/drying process in estuaries and the tidal-, buoyancy- and wind-driven circulation in the coastal region featured with complex irregular geometry and steep bottom topography. This model has been upgraded to the spherical coordinate system for basin and global applications. A non-hydrostatic version of FVCOM has been coded and is being tested.

Features

The present version of FVCOM includes:

- Choice of Cartesian or spherical coordinate system
- A mass-conservative wet/dry point treatment for the flooding/drying process simulation
- The General Ocean Turbulent Model (GOTM) modules (Burchard et al., 1999; Burchard, 2002) for optional vertical turbulent mixing schemes
- A water quality module to simulate dissolved oxygen and other environmental indicators
- 4-D nudging and Reduced/Ensemble Kalman Filters (implemented in collaboration with P. Rizzoli at MIT) for data assimilation
- Fully-nonlinear ice models (implemented by F. Dupont)
- A 3-D sediment transport module (based on the U.S.G.S. national sediment transport model) for estuarine and near-shore applications
- A flexible biological module (FBM) for food web dynamics study. With various pre-built functions and parameters for these groups, FBM allows either selecting a pre-built biological model (such as NPZ, NPZD, etc.) or building a biological model using the pre-defined pool of biological variables and parameterization functions. FBM includes seven groups:
    - Nutrients
    - Autotrophy
    - Heterotrophy
    - Detritus
    - Dissolved organic matter
    - Bacteria
    - Other

FVCOM was originally coded for sigma-coordinates in the vertical and now has been upgraded to a generalized terrian-following coordinate system with choices of various topographic-following coordinates. FVCOM is written with Fortran 90 with MPI parallelization, and runs efficiently on single and multi-processor machines.

FVCOM is an open source code ocean community model that always welcomes new users. **This program is only permitted for use in non-commercial academic research and education.** Users are required to register to receive the source codes, demo examples, and user manuals as well as some recommended postprocessing tools.

Graphical Interface

FVCOM uses the Generic Model Graphical Interface .

External Links

- FVCOM Model Developer Website
- FVCOM Publications
- FVCOM Forum
- FVCOM Help

Related Topics

- Generic Model Interface

4.3. Particle Tracking Model (PTM)

PTM

PTM	
Model Info	
Model type	Lagrangian particle tracker designed to allow simulating particle transport processes.
Developer	Neil J. MacDonald, Ph.D. Michael H. Davies, M.Sc., Ph.D., P.Eng. Coldwater Consulting Ltd
Web site	PTM web site
Tutorials	Models Section • PTM (Pending)

The Particle Tracking Model (PTM) is a Lagrangian particle tracker designed to allow simulating particle transport processes. PTM is funded through two US Army Corps of Engineers Engineering Research and Development Center (ERDC) research programs, the Coastal Inlets Research Program (CIRP) and the Dredging Operations and Environmental Research (DOER) Program.

The PTM model can be added to a paid edition of SMS.

Functionality

PTM has been developed for application to dredging and coastal projects including dredged material dispersion and fate, sediment pathway and fate, and constituent transport. The model contains algorithms that appropriately represent transport, settling, deposition, mixing, and resuspension processes in nearshore wave/current conditions. It uses waves and currents developed through other models and input directly to PTM as forcing functions.

Using the Model / Practical Notes

- The horizontal and vertical coordinates used for all PTM input files must be in meters. Geographic Coordinates cannot be used since it is a latitude/longitude system defined in decimal degrees.

- If last step trap is checked, the traps will not become active until the last time step.

How do I use PTM?

The SMS tutorials are a good place to start learning to use SMS and associated models. A tutorial is available for PTM.

Graphical Interface

SMS provides a graphical interface that is designed to visualize the projects being created, easily modify project parameters, and view the solutions produced by the PTM model. See PTM Graphical Interface for more information.

The PTM Graphical Interface contains tools to create and edit an PTM simulation. The simulation consists of a geometric definition of the sources, traps, and a set of numerical parameters. The parameters define the hydrodynamic input and options pertinent to the model.

The interface is accessed by selecting the Particle Module and setting the current model to PTM. If a simulation has already been created or an existing simulation read, the particle object will exist in the Project Explorer and selecting that object will make the Particle module active and set the model to PTM. See the Particle Module documentation for guidance on visualizing results.

The interface consists of the Particle Module Menus and tools augmented by the PTM Menu . See PTM Graphical Interface for more information.

PTM Menu

The *PTM* menu becomes active when a PTM model coverage has been created. For more information see PTM Graphical Interface .

Tools

PTM sources and traps are created in the Map Module as feature objects using the Map Module Tools .

PTM Coverages

In the SMS interface, two types of coverages can be applied to a PTM simulation. These include:

- PTM Source/Trap Coverage – Used to define input for a PTM simulation.
- PTM Gage Coverage – Used to view time series of output data computed by the PTM engine.

Theoretical Basis / Mathematical Details

Please refer to the model developer provided documentation listed in the external links section.

Numeric Engine Background

Please refer to the model developer provided documentation listed in the external links section.

General Steps to Build a PTM Model

1) Open SMS
2) Although not required, it is generally easier to visualize the model if reading in the hydrodynamic solution
3) Convert the horizontal and vertical coordinates to meters. PTM cannot be run in Geographic Coordinates since it is a latitude/longitude system defined in decimal degrees.
4) Switch to the Map Module
5) Create a PTM type coverage
6) Create sources and traps using feature points , arcs , and polygons
7) Switch to the Particle Module
8) Use the *PTM* menu to create a new PTM simulation

9) Use the *PTM* menu to open the *PTM Model Control* dialog
   1) Specify input and output parameters in the model control
   2) Click on the **Create input file(s) from data** button to create a native sediment file , and hydrodynamic input files if needed. You may use mesh, water surface elevation, and velocity data from another model (ADCIRC, ADH, RMA2) to create the necessary fort.15 and/or *.h5 file.
10) Save the simulation
11) Use the *PTM* menu to perform a model check
12) Use the *PTM* menu to run PTM

General Post-Processing Steps

1) Use the *display options* to visualize the results
2) Use the *Data Calculator* or *Create Datasets* dialog to create particle datasets
3) Create a Cartesian Grid and use the *Create Grid Datasets* dialog to create Cartesian grid datasets
4) Review the Trap Output files in an ASCII file viewer

External Links

- U.S. Army Corps of Engineers DOER PTM website [75]
- Sep 2006 ERDC/CHL TR-06-20 PTM: Particle Tracking Model Report 1: Model Theory, Implementation, and Example Applications [76]
- Jul 2005 ERDC TN-DOER-D4 Particle Tracking Model (PTM) in the SMS: I. Graphical Interface [77]
- Jul 2005 ERDC TN-DOER-D5 Particle Tracking Model (PTM): II. Overview of Features and Capabilities [78]
- Jul 2005 ERDC TN-DOER-D6 Particle Tracking Model (PTM) in the SMS: III. Tutorial with Examples [79]
- Jul 2008 ERDC/CHL CHETN-IV-71 Particle Tracking Model (PTM) in the SMS 10: IV. Link to Coastal Modeling System [80]
- 2007 The Particle Tracking Model: Description and Processes [81]
- Application of the Particle Tracking Model to Predict the Farfield Fate of:
  - 2007 Sediment Suspended by Nearshore Dredging and Placement, Brunswick, GA [82]
  - 2008 Dredged Suspended Sediment at the Willamette River [83]
- 2009 Assessment of Dredging-Induced Sedimentation on Winter Flounder Spawning habitat [84]

Related Topics

- Particle Module
- Mesh Module

PTM Model Control

The PTM model control is used to create the program control file , which contains general simulation options. Refer to the PTM manual for a more detailed description of how these parameters affect the model results.

Time

The *Time* tab is used to specify time information to be written to the program control file . If a control on the dialog is not enabled, the corresponding keyword will not be written to the program control file . Controls on the dialog are disabled based on the options chosen on the other model control tabs. For example, if no trap file is specified on the *Files* tab, the time controls related to traps will not be available. There is a plot on the dialog of the specified times. This tab has the following inpute fields:

- Simulation

  - Start
  - Stop
  - Duration
  - Time Step
- Particle Output
- Shear, Bedforms, and Mobility Updates
- Mapping Output
- Flow and Elevation Update
- Waves
  - Start
  - Time Step
- Hydrodynamics
- Traps
  - Start
  - Stop
  - Last time step trap

Files

The *Files* tab is used to specify input and output file information to be written to the program control file (*.pcf).

Pressing the **Create input file(s) from data...** button on the *Files* tab will open the *Create PTM External Input Files* dialog. This dialog can import results from ADCIRC, ADH, SRH-2D, CMS-FLOW or other models to create a mesh (fort.14) file, grain size native sediment (*.sediment) file, and/or a XMDF hydrodynamic input flow file (*.h5) to import into a PTM model.

- *Note:* SMS is not capable of importing RMA2 and FESWMS files into PTM at this time.

Native Sediment Grain Size

1) If "Uniform bed" is selected from the "Options" column an **Options** button is displayed under "Filename". Pressing this button allows creating uniform grain sizes for D35, D50 and D90 (where D35 means 35% of the grains are smaller than diameter D). If the project is saved, the :SEDIMENT_FORMAT card is created, with the keyword UNIFORM and the 3 grain sizes being written out.
2) If Sediment file is selected, manually select the file name by pressing the button under "Filename".
3) Pressing the **Create input file(s) from data** and selecting to create a Native sediment grain size file will automatically toggle the native sediments grain size to "sediments file" and place the new file under the "Filename". The user can enter in the D values with the constant mobile bed type or a Nikuradse‘s equivalent sand roughness Ks (surface roughness height which equals D90) for the constant fixed bed type or use a dataset for spatially varied grain sizes.

Computations

The *Computations* tab is used to specify model computation options to be written to the program control file . This tab has the following input fields:

- Computation Methods
  - Advection
  - Centroid
  - Distribution
  - Eulenian

  - Velocity
  - Numerical scheme
- Computational Parameters
  - Bed porosity
  - Temperature
  - Bed density
  - Salinity
  - Minimum depth
- Diffusion Parameters
  - Min. diffusion coefficient
  - Turbulent diffusion scalar
  - Wave diffusion scalar
- Model Calculation Options
  - Currents
  - Morphology
  - Neutrally buoyant particles
  - Bedforms
  - Hiding and exposure
  - Particle-bed interaction
  - Turbulent shear
  - Source and trap Z-value relative to datum
  - Residency (polygon trap required)
  - Wave mass transport

Output

The *Output* tab is used to specify model output options to be written to the program control file . This tab has the following input fields:

- File Output
  - Print default keywords
  - Compress XMDF files
  - Tecplot map data
  - Tecplot particle data
  - Tecplot particle path data
  - tecplot population history
- Mapping Output (on Hydrodynamic Grid)
  - Bed evolution
  - Bed level change
  - Flow conditions
  - Mobility of native sediments
  - Native sediment bedforms
  - Potential sediment transport rate

  - Shear stress
  - Wave parameters
- Particle Output
  - Critical shear
  - Density
  - Fall velocity
  - Grain size
  - Height above the bed
  - Mass
  - Mobility
  - Source
  - State
  - Velocity components

Related Topics

- PTM

PTM Graphical Interface

The SMS interface to PTM includes tools for creating input files as well as post-processing capabilities. The PTM interface includes the following components:

PTM Menu

The following menu commands are available in the *PTM* Menu:

New Simulation

Creates a new PTM simulation and adds it to the Project Explorer .

Model Check

Checks the active PTM simulation for common input errors.

Model Control

Opens the *Model Control* dialog (used to organize input files, specify model parameters, choose output options, etc.).

Run Model

Launches the PTM model using the active PTM simulation input files.

Model Control

The *PTM Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, computation parameters, output options, and other global settings.

Running the Model

The PTM Files are written automatically with the SMS project file or can be saved separately using the *File* | **Save PTM** or *File* | **Save As** menu commands. See PTM Files for more information on the files used for the PTM run.

PTM can be launched from SMS using the *PTM* | **Run Model** menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *PTM* | **Model Check** menu command.

PTM Model Check

The model check will give a warning if it suspects a potential error has been made in creating the input files for PTM and suggest how to resolve the issue. The issues shown in the model checker may or may not cause problems, it is left to the modeler to decide if an issue will affect the results of their simulation.

Map Module

PTM sources and traps are created in the Map Module as feature objects in a PTM type coverage .

Feature Points

- Instant Mass Source – Release at a single point in time (Ex. accidental vessel spill)
- Point Mass Rate Source – Release over a period of time (Ex. leaking pipeline)
- Vertical Line Source – Release with a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line (Ex. bucket dredge)

Feature Arcs

- Horizontal Line Source – Release with a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line
- Horizontal Line Trap – Count particles crossing a line
- Vertical Area Source – Under development

Feature Polygons

- Horizontal Area Source – Release with a uniform distribution over the area and a Gaussian distribution perpendicular to the source
- Horizontal Area Trap – Count particles entering an area

For more information on sources, see PTM Sources . For more information on traps, see PTM Traps .

Polygon Attributes Dialog

Feature polygons created in a PTM type coverage can be used to create either sources or traps .

Related Topics

- PTM

PTM Particle Filters

The particle filters allow evaluating specific particles based upon different criteria. The filters act exclusively upon particle datasets. Multiple filters can be used together to filter multiple datasets.

The filters affect:

- Displayed particles
- Selectable particles (hence selected particle info)
- Compute Grid datasets functionality

Dialog Description

The *Filter Options* dialog is brought up by clicking on the **Filter Options** command in the *Data* menu when the Particle Module is active. The following controls are used to define the particle filters.

Filters

Use the **New** and **Delete** buttons to create / remove filters. Each filter may be enable / disabled by clicking its toggle box. The filters are defined by selecting a Dataset whose values will be compared against the operands / operators pairs defined in the Options control.

Datasets

Each filter operates on a single Particle Module dataset. Examples of datasets in the Particle Module include solutions created by the PTM model or using the data calculator.

Options

- Operators – Comparison operators

<	Less than
<=	Less than or equal to
equal	Equal to
not equal	Not equal to
>	Greater than
>=	Greater than or equal to

- Operands – Values compared against the selected dataset using the corresponding operators
- Conjunction – This toggle box disables the second operator / operand pair or performs the and function on them against the first operator / operand pair.

PTM Traps

A trap is defined as an area into which particles enter and are counted.

Global Trap Options

Global options affect all traps.

- Time limits can be specified on the trap's operation in the *Time* tab of the *PTM Model Control* dialog.
- Traps can be set to only be active on the last time step of the PTM simulation by selecting "Last timestep trap" in the *Time* tab of the *PTM Model Control* dialog.

Individual Trap Options

Individual options are specified on a trap by trap basis.

- The trap may be open (particles are free to leave) or closed.
- Particles can be counted once per simulation (single trap) or every time they re-enter an open trap.
- The trap can have a bottom and top elevation defining the portion of the water column the trap exists within.
- ID – Integer value to identify the trap
- Name – Text to identify the trap

PTM Polygon Attribute Dialog

The options for an individual trap can be set in the *Feature Object Attributes* dialog. The dialog is accessed by right-clicking on the trap polygon and selecting the **Attributes** menu command.

Horizontal Line Traps

Horizontal line traps lie in a vertical plane, defined by the trap end points and a top and bottom elevation.

Horizontal line traps have a direction associated with them. The direction of a trap determines which particles the trap will count.

- Decreasing x-coordinate – Only counts particles passing through the trap moving in the negative x-direction.
- Either direction – Counts particles passing through the trap regardless of direction.
- Increasing x-coordinate – Only counts particles passing through the trap moving in the positive x-direction.

Horizontal Polygon Traps

Horizontal polygon traps lie on a horizontal plane and are extruded in the vertical direction based on the trap's top and bottom elevation.

Related Topics

- PTM Trap File
- PTM Sources

4.3.a. PTM Coverages

PTM Gage Coverage

A "PTM Gage" coverage has been added to allow the creation of virtual gages. A virtual gage can be associated with either a point or a polygon. When a virtual gage is associated with a point, specify a radius that defines the area being monitored by the gage (creating a circle).

The gages can be thought of as open traps which monitor the number parcels and material that enter the specified region during the simulation.

When using the **Particle Module Compute Grid Datasets** command, choose to compute datasets associated with a grid or with a PTM Gage coverage. When the gage coverage option is selected, SMS computes output data associated with each point and/or polygon. This data consists of sets of xy series curves organized by solution and dataset name. Each curve represents a quantity related to the specific gage. These quantities include count, accumulation, concentrations, rate of exposure and all of the options supported by the **Compute Dataset** command including the options to bin concentration, exposure and dosage in the "Z" direction (creating 3D data). The output data is visualized using a PTM Gage plot type in the plot wizard for 2D plots .

The polygons and points in the virtual gage coverage are created with discrete color attributes. Each point has a radius attribute that represents the area over which the computations will apply.

PTM Trap Output as Gages

In addition to creating virtual point and polygon gages, the PTM Gage Coverage can also be used to view data output by PTM in its Trap Output file. This functionality does not include the concentration, dosage and exposure options, but it does allow visualizing data related to closed traps.

Related Topics

- PTM Coverages

4.3.a.1. PTM Sources

PTM Sources

The material which is to be modeled in PTM is released from sources. The amount of material released from each source is specified as a mass, either as an instantaneously released total mass, or as a mass release rate over a given time period. PTM represents this mass by a finite number of particles.

Particles can be introduced via three different types of sources:

- Point Sources
- Line Sources
- Area Sources

There can be any number of any source type used in a simulation, and different source types can be specified in the same simulation.

Point Sources

Instantaneous

If the material to be modeled is to be released at a single point in time, then an instantaneous source should be specified. An example of this type of release is an accidental spill from a vessel. This type of release occurs at a fixed location, and the full release of material occurs at the time given and with the properties specified.

Varying-Release

If the release of material occurs over a period of time, then a varying-release point source should be specified. An example of this type of release is a leak from a pipeline. The characteristics of release point sources can vary with time (e.g., release rate, three dimensional positions, etc.). Varying-release point sources can be started, stopped, re-started, moved, etc., as directed in a source release schedule, which is developed through the SMS interface.

The horizontal and vertical radii of both types of point sources can be specified in the source release schedule. If either radius is greater than zero, then the initial locations of the individual particles are varied so as to produce a two- or three-dimensional Gaussian-distributed cloud.

Line Sources

Line sources must either be vertical or horizontal and are varying-release. Particles released from a line source will have a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line. Line sources are specified by their end points. Line sources may move or change length, position, or discharge properties with time. Linear interpolation in time is used for most properties in a line source, but the characteristics of the release do not vary along the line source (e.g., the release rate of particles can vary with time for a line, but the rate will be the same over the length of that line). To model a line source with varying characteristics along the line, one could use a series of lines positioned end to end, each with different characteristics.

Vertical Line Source Datums

The following vertical line source datums can be used to specify the vertical line source top and bottom elevation:

- Bed (default) – Elevations are specified relative to the bed. The rate is given in kg/m/s.
- Surface – Elevations are specified relative to the water surface. The rate is given in kg/m/s.
- Depth distributed – Elevations are specified as percentages of the water depth. The rate is given in kg/s in this case. An example is if wanting to distribute the source over a fraction of the water depth, set z1=0.33 and z2=0.67 to introduce the source over the middle third of the water column. The depth-distributed source is taken as a percentage from the bed. So if z1 = 0.0 and z2 is 0.5, this suggests a vertical line source that is the lower one half of the water depth.

Area Sources

Area sources lie in a veritical or horizontal plane and are varying-release. Area sources are polygons and are specified by the locations of their vertices. The vertices must be ordered with a counter-clockwise convention. Particles are released from an area source such that there is a uniform distribution over the area and a Gaussian distribution perpendicular to the source. Source properties within an area source are uniform across the polygon.

Estimating generated Parcels

The *Estimate Generate Parcels* dialog allows editing the parcel mass of PTM sources (point, line, or polygon) to estimate the number of parcels created. This can be access by right-clicking on the PTM coverage and selecting **Esitmate generated Parcels** .

Calculated vs. Specified Values

PTM can calculate the Fall Velocity, Critical Shear Initiation, and Critical Shear Deposition. Specifying a value of -1.0 for any of these parameters will result in the model calculating the value. See the model user manual for information on the equations used to calculate these values.

PTM Polygon Attributes Dialog

The options for an individual source can be set in the *Feature Object Attributes* dialog. The dialog is accessed by right-clicking on the source object and selecting the **Attributes** menu command.

Related Topics

- PTM Source File
- PTM Traps
- PTM Coverages

PTM Arc Attributes Dialog

Feature arcs created in a PTM type coverage can be used to create either sources or traps .

Feature Object Attributes dialog for a Source Arc

- *Type* – The selected option will determine which options are available. Options include; "Generic", "Horizontal Line Source", and "Horizontal Line Trap".
- *Source ID* – Used to give multiple source arcs differing IDs so that multiple sources can be tracked.
- *Name* – Field where a description for the source arc can be entered.
- *Show Coordinates in Spreadsheet* – Toggling on this option will populate the spreadsheet below with coordinate columns.

- Date/Time
- *Parcel Mass* – Each parcel represents a specific amount of sediment. Combined with the rate PTM computes the number of parcels being tracked. When a specific quantity of material is being generated by a dredge or other source, this amount allows the tracking of all of that material.
- *Radius* – PTM will distribute the parcels generated for this source randomly along the line. The position will also vary around the line within this specified radius.
- *Rate* – A specific quantity of sediment will be generated based on this rate. The longer the line, the more sediment is added to the flow field.
- *Median Grain Size* – This is the d50 of the sediment being added.
- *Standard Deviation* – This phi unit defines how the grain size varies from one parcel to the next.
- *Density* – This is the density of the sediment. This can be reduced to the density of water to create neutrally buoyant particles.
- *Fall Velocity* – This is the speed of descent of the particles. The shape of the particle influences this quantity. Entering a value of -1 tells PTM to compute this value.
- *Critical Shear Intiation* – Specifies the shear values to initialize motion for this particle.
- *Critical Shear Deposition* – The threshold for this particle to settle.

Feature Object Attributes dialog for a Trap Arc

- *Type* – The selected option will determine which options are available. Options include; "Generic", "Horizontal Line Source", and "Horizontal Line Trap".
- *Trap ID* – Used to give multiple trap arcs differing IDs so that multiple traps can be tracked.
- *Name* – Field where a description for the trap arc can be entered.
- Trap Elevation
  - Bottom
  - Top
- Trap Options
  - Open Trap (particles can leave trap)
  - Single Trap (Count particles once per simulation)
- Trap End Point Locations
  - X1
  - X2
  - Y1
  - Y2

Related Topics

- PTM Sources
- PTM Traps

PTM Feature Point Attributes Dialog

The PTM *feature point attributes* dialog is used to specify point source attributes.

Feature Object Attributes dialog for Node Points

- Type

- Source Identifiers
  - Name
  - ID
- Date/Time
- X
- Y
- Elevation
- Parcel Mass
- Horiz. Radius
- Vert. Radius
- Rate
- Median Grain Size
- Standard Deviation
- Density
- Fall Velocity
- Critical Shear Initiation
- Critical Shear Deposition
- Delete Row

Feature Object Attributes dialog for Vertical Node Points

This dialog is the same as the *Feature Object Attributes* dialog for Node Points with the following additions:

- Vertical Line Sources window
- Vertical Line Source Information
  - Numbers of sources
  - Active
  - Datum

Related Topics

- PTM Sources

4.3.b. PTM Files

PTM Files

Input and Output files for PTM .

Input Files

- Required files
  - Program Control File (*.pcf) – Run time instruction for the model.
  - Mesh file – Bathymetry and boundary information for the model.

PTM Version 1.0 supports ADCIRC compatible three-noded triangular finite element mesh files (fort.14 or .grd). SMS contains tools to convert mesh files to this format.

- Hydrodynamic input file – Time varying water surface elevation and depth-averaged velocities.

  The SMS 9.2 interface to PTM Version 1.0 supports XMDF (*.h5) format hydrodynamic files. SMS contains tools to convert mesh output files to this format.

- Native sediment file – Native sediments over the domain defined by the mesh file.

- Optional files
  - Sediment source file – The sediment sources in the simulation.
  - Trap file – Locations where information about the simulation should be extracted.
  - Wave file(s) – Time varying information about the wave field.
  - Wave breaking file(s) – Time varying information about wave breaking.
  - PTM Boundary Condition file – Describes the wet/dry interface.

Output Files

The output files use the name given after the keyword :OUTPUT_PREFIX in the program control file and are appended with the following endings:

- **Mapping Output File** (*_maps.h5) – Behavior of the flow and native sediments over the whole domain incremented by time.
- **Parcel Data File** (*_particles.h5) – Parcel information incremented by time.
- **Echo file** (*_input.out) – ASCII file echoing the input parameters of the simulation.
- **Neighbor file** (*.neighbors) – Contains mesh connectivity information. Once generated, can be specified as an input file to avoid regenerating for subsequent simulations with a given mesh. If the mesh changes, the neighbor file should be regenerated.
- **Trap Count File** (*_count.out) – Contains the time and ID of particles entering a trap
- **Trap Residency File** (*_residency.out) – Contains the time particles enter and exit a trap and the total time spent in the trap.

Related Topics

- PTM

PTM Control File

The PTM program control file (*.pcf) contains run time instruction for the model.

File Overview

The program control file (PCF) contains all the information necessary for the PTM to perform the simulation requested. The PCF file is a keyword oriented file. Lines in the file should contain either comments, with the line beginning with a "#" symbol, or a keyword, a single word beginning with a colon, ":", and its associated values. Some keywords have no associated values, e.g.:

```
:MOBILITY_MAPPING
```

These are generally on/off instructions to the model. Keywords pointing ot filenames or setting constants have one or two associated values, e.g.:

```
:MESH_FORMAT  ADCIRC    :MESH_FILE  estuary.grd
```

Keywords and values are not case sensitive with the single exception of the simulation output file names (i.e. file names like "Run_C1_Tp_12s_paths.out" are possible).

The PCF file is read until the instruction :

```
:END_DATA
```

is encountered.

The following list summarizes the rules for construction of a PCF:

- Keywords should be preceeded with ":".
- Comments should be preceeded with "#".
- Keywords are not case sensitive.
- Keywords and value(s) are white space delimited (tab or space) and, so, should not contain blanks.
- Keywords may be placed in any order in the file.
- If a keyword appears more than once, then the final incidence is used.
- All value units are S.I.
- All times are relative to the clock defined by the currents file.

Example PCF File

```
#########################################################
#  PTM  Simulation  file  written  by  SMS  9.2.1
#  SMS  Build  Date:  Mar   9  2007
#  Date:  03/09/07
#  Time:  16:36:04
#########################################################
:CURRENTS
#
:ID_OUTPUT
:ELEVATION_OUTPUT
:GRAIN_SIZE_OUTPUT
:MOBILITY_OUTPUT
:STATE_OUTPUT
:PARCEL_MASS_OUTPUT
:OUTPUT_INC  10
:MAPPING_INC  1000
#
:BY_WEIGHT
:GRID_UPDATE  300
#
:MESH_FORMAT  ADCIRC
:MESH_FILE  estuary.grd
:FLOW_FORMAT  XMDF
:FLOW_FILE_XMDF  estuary_tide.h5
:XMDF_VEL_PATH  Datasets/tide  only  velocity
:XMDF_WSE_PATH  Datasets/tide  only  wse
:NEIGHBOR_FILE  estuary.neighbors
:SOURCE_FILE  estuary_Tides.source
:SEDIMENT_FILE  estuary.sediments
:OUTPUT_PREFIX  estuary_Tides
#
:START_RUN  2004  2  11  0  0  0
:STOP_RUN  2004  2  12  7  45  0
:START_FLOW  2004  2  9  0  0  0
#
:TIME_STEP  3.000000
#
:KET  0.25
```

```
:KEV  0.00859
:TEMPERATURE  15
:SALINITY  34
:RHOS  2650
:MIN_DEPTH  0.01
#
:ETMIN  0.02
:EVMIN  0
#
:EMRL_GUID_COV  7d0bcdf2-f218-4046-ba24-218294d74794
:EMRL_PARTICLE_SET_GUID  b7183313-bcba-45d9-8cf7-f6ce96cf1d45
:END_DATA
```

Keywords

Keyword	Value(s)	Action
:waves		Instructs the model that waves are to be runs in the model. Turns on the various wave flags. Default is inactive.
:currents		Instructs the model that currents are to be runs in the model. Turns on the various wave flags. Default is inactive.
:no_parcels		Instructs the model to only perform Eulerian calculations (i.e. no particles are released). Source file is not required if this option is used. Default is inactive.
:ensim		Tells the model that the input bathymetry file uses the CHC EnSim convention of positive upwards. Default is inactive. Default is inactive.
:paths		Instructs the model to output the parcel path files. Default is inactive.
:morphology		Instructs the model to output the predicted bed evolution on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Same result as “:bed_level_mapping “. Default is inactive.
:flow_mapping		Instructs the model to output the flow conditions on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Default is inactive.
:bedform_mapping		Instructs the model to output the predicted native sediment bedforms on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Default is inactive.
:wave_mapping		Instructs the model to output the three predicted wave parameters (H, T, θ) on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Default is inactive.
:mobility_mapping		Instructs the model to output the predicted mobility of the native sediments on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Same result as “:morphology”. Default is inactive.
:bed_level_mapping		Instructs the model to output the predicted bed evolution on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Same result as “:morphology“. Default is inactive.
:bed_level_change_mapping		Instructs the model to output the predicted bed level change on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Default is inactive.

:transport_mapping		Instructs the model to output the predicted potential sediment transport rate of the native sediments on the Eulerian mesh node locations. Frequency of output is given by ‘mapping_inc’. Default is inactive.
:grain_size_output		Instructs the model to output the grain size of the parcels. Frequency of output is given by ‘output_inc’. Default is inactive.
:mobility_output		Instructs the model to output the mobility of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:elevation_output		Instructs the model to output the elevation of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:height_output		Instructs the model to output the height of each parcel above the bed. Frequency of output is given by ‘output_inc’. Default is inactive.
:state_output		Instructs the model to output the state of each parcel (active=1, dormant=0). Frequency of output is given by ‘output_inc’. Default is inactive.
:id_output		Instructs the model to output the identification number of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:source_output		Instructs the model to output the original source of the each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:flow_output		Instructs the model to output the velocity components of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:density_output		Instructs the model to output the density of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:parcel_mass_output		Instructs the model to output the mass of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:time_step	<real>	The time step of the simulation in seconds, e.g.: • time_step 2. • Input value required.
:by_weight		Instructs the model to build up the distributions of parcel grain sizes at each source such that they are Gaussian in terms of weight (i.e. “percentage finer by weight”), rather than by grain size. Default is inactive.
:no_bedforms		Instructs the model to skip bedform calculations on the Eulerian grid. Default is inactive.
:bed_porosity		Input statement for bed porosity. Default value is 0.4.
:eulerian_method	ptm or van_rijn	Selects which method to use to determine the native sediment mobility and shears on the Eulerian grid. The two choices are ptm and van_rijn, e.g.: • eulerian_method van_rijn • The ptm method is much faster. Default is ptm.
:centroid_method	rouse or van_rijn	Selects which method to use to determine the parcel mobility and shears. The two choices are rouse and van_rijn, e.g.: • lagrangian_method rouse • The rouse method uses empirical approximations of integrated

		Rouse curves. The van_rijn method uses a numerical integration of the total sediment transport at the location of the parcel to compute the centroid elevation. The rouse method if much faster. Default is rouse.
:advection_method	1d (formerly 2d – not available in SMS interface) or 2d (formerly q3d - default) or 3d	Selects which method to use for the advection of the parcels. The three choices are 1d, 2d and 3d, e.g.: • advection_method 2d • The 1d method places the parcel at the elevation of the centroid of the local total load distribution. The 2d method allows the parcel to be above or below the total load centroid elevation with the parcel moving vertically towards the centroid. The 3d option allows the parcel to move freely in the vertical in response to a vertical force balance. Default is 2d.
:velocity_method	2d (log) or 2d (uniform) or 2d (two-point) or 3d or 3ds or 3dz	Selects which method to use to compute the velocity profile in the vertical, e.g.: • velocity_method 3dz • The 2d (uniform) option causes the model to use the given velocity throughout the water column. This method is designed primarily for testing. The 2d (log) option causes the model to use a logarithmic velocity profile. The 3d variants force the model to use the given three-dimensional velocity components. These should only be used with three-dimensional input. Default is 2d (log)
:mesh_format	ADCIRC	Gives the format of the mesh file, e.g.: • mesh_format ADCIRC • Options are CMS-2D, CMS-3D, ADCIRC, M2D and CH3D Default value is ADCIRC.
:mesh_file	<character>	Gives the name of the mesh file, e.g.: • mesh_file test.grd • Input value required.
:flow_format	ADCIRC or CMS-2D or CMS-3D or CH3D	Gives the format of the hydrodynamic files: • flow_format ADCIRC • Options are ADCIRC, CMS-2D (for M2D), CMS-3D (for M3D) and CH3D. Default value is ADCIRC.
:flow_files	<character> <character>	Gives names of the hydrodynamic files, e.g.: • flow_files A1.63 A1.64 • Note that the extensions are optional; if a '.' is not found in the filename, the *.63 and *.64 will be appended. Input values required.

:flow_file_uv	<character>	Gives names of the hydrodynamic flow file, e.g.: • flow_file_uv A1.64 • Note that the extensions are optional; if a ‘.’ is not found in the filename, the *.64 will be appended. Input values required. This command allows names with spaces, including directories.
:flow_file_z	<character>	Gives names of the hydrodynamic elevation file, e.g.: • flow_file_z A1.63 • Note that the extensions are optional; if a ‘.’ is not found in the filename, the *.63 will be appended. Input values required. This command allows names with spaces, including directories.
:flow_file_xmdf	<character>	Gives the name of the XMDF format flow file. This is a single file in h5 format.
:flow_file_dst	<character>	Gives name of the density/salinity/temperature file, e.g.: • flow_file_dst A1.45 • This is only used for some ADCIRC-3D model input. If IDEN=0, then no file is required.
:bc_file	<character>	Gives the name of the boundary condition file. Format depends on model type.
:xmdf_wse_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Water Surface Elevation (63)'
:xmdf_vel_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Velocity (64)'
:xmdf_u_path	<character>	Gives the internal data structure path for the U velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:xmdf_v_path	<character>	Gives the internal data structure path for the V velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:xmdf_w_path	<character>	Gives the internal data structure path for the W velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:m2d_control_file	<character>	Gives the name of the M2D control file {*.m2c}. This is only used for M2D version 3.02 and requires the mesh format to be specified as ‘m2d’, rather than ‘cms-2d’
:xmdf_dep_path	<character>	Gives the internal data structure path for the depth in the XMDF format file (M3D only). The default is 'Datasets/Depth'
:xmdf_sal_path	<character>	Gives the internal data structure path for the salinity in the XMDF format file (M3D and CH3D). The default is 'Datasets/Salinity'
:xmdf_temp_path	<character>	Gives the internal data structure path for the temperature in the XMDF format file (M3D and CH3D). The default is 'Datasets/Temperature'
:xmdf_M3D_path	<character>	Gives the internal data structure path for the 3D data in the XMDF format file (M3D only). The default is blank.
:xmdf_d35_path	<character>	Gives the internal data structure path for the D35 values in the XMDF format sediment file. The default is blank.

:xmdf_d50_path	<character>	Gives the internal data structure path for the D50 values in the XMDF format sediment file. The default is blank.
:xmdf_d90_path	<character>	Gives the internal data structure path for the D90 values in the XMDF format sediment file. The default is blank.
:start_flow	yyyy mm dd hh mm ss	Time origin of the ADCIRC flow files, e.g.: • start_flow 1999 12 3 22 30 0 • Each output step of the ADCIRC .63 and .64 files has a timestamp in seconds that is relative to a UTC time selected when the ADCIRC model control file is established. Input values required.
:wave_format	STWAVE or WABED or XMDF	Gives the format of the wave file, e.g.: • wave_format STWAVE • Options are STWAVE, WABED or XMDF. Default value is STWAVE.
:wave_files	<integer>	Gives the number of STWAVE .wav files that are to be used in the present simulation, e.g.: • wave_files 100 • The list of file names should start on the line immediately after the :end_data statement. Input value required if waves are used.
:wave_file_xmdf	<character>	Gives the name of the XMDF input file with wave data, e.g.: • wave_file_xmdf south_waves_example.h5 • This is the main (parent) grid. Input value required.
:wave_file_2_xmdf	<character>	Gives the name of the XMDF input file with wave data, e.g.: • wave_file_xmdf south_waves_example.h5 • This is the nested (child) grid. Input value required.
:wave_frames	<integer>	Gives the number of output frames (steps) that appear in each STWAVE .wav file, e.g.: • wave_frames 1 • In general, STWAVE files will contain a single frame if produced using the SMS steering module and more than one frame if produced using a stand-alone version of STWAVE (e.g. steering module run -> 100 files with one frame each; stand-alone -> 1 file with 100 frames. Input value required if waves are used.
:wave_step	<real>	Gives the time (in seconds) between each output frame (step) in the STWAVE .wav file(s), e.g.: • wave_step 7200. • Input value required if waves are used.
:m2d_step	<real>	Gives the time (in seconds) between each output frame (step) in the XMDF-format M2D file, e.g.: • m2d_step 3600. • New XMDF files have the time units specified, so this may now longer be required.

:wave_grid_angle	<real>	Gives the orientation angle (in degrees, clockwise from y-axis to N) of the STWAVE grid, e.g.: • wave_grid_angle 20. • Input value required if waves are used.
:wave_x_origin	<real>	x-ordinate of the STWAVE grid origin, e.g.: • wave_x_origin 1226722.50 • Input value required if waves are used.
:wave_y_origin	<real>	y-ordinate of the STWAVE grid origin, e.g.: • wave_y_origin 39990.15 • Input value required if waves are used.
:nested		Instructs the model that nested STWAVE grids are to be used. The list of file names should start on the line immediately after the list of parent STWAVE file names. The nested and parent STWAVE grids must have the same number of frames and the same times associated with each frame.
:wave_grid_angle_2	<real>	Gives the orientation angle (in degrees, clockwise from y-axis to N) of the nested STWAVE grid, e.g.: • wave_grid_angle 20. • Input value required if waves are used.
:wave_x_origin_2	<real>	x-ordinate of the nested STWAVE grid origin, e.g.: • wave_x_origin 1226722.50 • Input value required if waves are used.
:wave_y_origin_2	<real>	y-ordinate of the nested STWAVE grid origin, e.g.: • wave_y_origin 39990.15 • Input value required if waves are used.
:start_waves	yyyy mm dd hh mm ss	Time of the first STWAVE frame, e.g.: • start_waves 2004 12 3 0 30 0 • Input values required if waves are used.
:exceed_waves		Instructs model to hold waves steady, if simulation exceeds duration of wave input file
:source_file	<character>	Gives the name of the file containing the sediment source information, e.g.: • source_file test_2.source • Input value required.
:neighbor_file	<character>	Gives the name of the file containing the neighbor source information, e.g.: • neighbor_file test.neighbor • The model will generate a file if one is not supplied. Input value required.
:output_prefix	<character>	Gives the prefix of the files used for output, e.g.: • output_prefix west_test_1

		• The model will append an extension depending on the nature of the output file. Input value required.
:undertow		The model computes wave undertow. Default is inactive. Needs to be made more general.
:trap_file	<character>	Gives the name of the trap file, e.g.: • trap_file trap_A.dat • See trap section for details. If this keyword command is not issued, then no trap is present. Default is no trap.
:xmdf_grid_path	<character>	Path to the main folder in an M2D h5 file. This is the folder with such data as the Origin, Bearing etc. The datasets folder is usually below this one.
:last_step_trap		Surveys all particles on the last time step and places them in a trap, whether they are moving or not.
:start_trap	yyyy mm dd hh mm ss	Time at which the trap, if specified, is operational, e.g.: • start_trap 2004 12 4 16 0 50 • Input values required if trap is used. Default start time is the start of the simulation.
:stop_trap	yyyy mm dd hh mm ss	Time at which the trap, if specified, ceases operation, e.g.: • stop_trap 2004 12 4 19 20 0 • Input values required if trap is used. Default stop time is the end of the simulation.
:sediment_format	ADCIRC or XMDF_PROPERTY or XMDF_DATASET or m2d	Gives the format of the native sediment file, e.g.: • sediment_format ADCIRC • Default value is ADCIRC.
:sediment_file	<character>	Gives the name of the file containing the grain sizes of the native sediments, e.g.: • sediment_file sed-A2.grd
:start_run	yyyy mm dd hh mm ss	Start time of the simulation, e.g.: • start_run 2004 11 4 16 0 0 • Input value required.
:stop_run	yyyy mm dd hh mm ss	Stop time of the simulation, e.g.: • stop_run 2004 11 14 13 30 0 • Must specify either :duration or :stop_run.
:duration	<real>	The duration of the simulation in seconds. Must specify either :duration or :stop_time.
:grid_update	<integer>	The increment in steps between updating of the mobility, shears and bedforms of the Eulerian grid

:flow_update	<Integer>	The increment in steps between updating of the flows and elevations of the Eulerian grid
:ket	<real>	Diffusion scalar. Default value is 0.25.
:temperature	<real>	The water temperature in °C. The default value is 15°C.
:salinity	<real>	The water salinity in ‰. The default value is 34 ‰.
:output_inc	<integer>	The increment in number of steps between output of the Lagrangian parcel data. The default value is 10.
:mapping_inc	<integer>	The increment in number of steps between output of the Eulerian grid data. The default value is 1000.
:min_depth	<real>	The minimum computation depth in m. The default value is 0.01 m.
:debug		Tells the model to turn off certain random generators routines and output stage times
:ensim_parcels		The model outputs a file with particle data in EnSim format (.pcl)
:ensim_maps		The model outputs a file with map data in EnSim format (.t3s)
:tecplot_parcels		The model outputs a file with particle data in Tecplot format (.plt)
:tecplot_maps		The model outputs a file with map data in Tecplot format (.plt)
:no_xmdf_parcels		The model will not output a parcel data file in XMDF format (.h5)
:no_xmdf_maps		The model will not output a map data file in XMDF format (.h5)
:population_record		A file with population history (e.g. particles born, trapped, deposited, etc.) is output as in Tecplot format as *_population.plt
:tau_cr_output		Instructs the model to output the critical shear for the initiation of motion of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:fall_velocity_output		Instructs the model to output the fall velocity of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:xmdf_compressed		Instructs the model to use compressed XMDF files. Default is inactive.
:no_hiding_exposure		Turns off hiding and exposure routines. Default is active.
:no_turbulent_shear		Turns off probabilistic shear calculations. Default is active.
:no_bed_interaction		Turns off bed interaction routines. Default is active.
:no_wave_mass_transport		Turns off wave mass transport calculations. Default is active.
:wave_mass_transport		Turns on wave mass transport calculations. Default is inactive.
:residence_calc		Model performs residency calculations (requires a polygon trap to be active)
:neutrally_buoyant		Model assumes all particles are neutrally buoyant (i.e. particle density set to that of the fluid and fall velocity is set to 0.). Default is inactive.
:bottom_flow_format	ADCIRC or XMDF	Gives the format of the bottom currents file. Options are ‘Default is ‘adcirc’.
:xmdf_bot_path	<character>	Gives the internal data structure path in the XMDF format flow file. The

		default is 'Datasets/Velocity (64)'
:bottom_flow_file	<character>	Gives the name of the bottom currents file.
:bottom_flow_height	<real>	Sets the height of the input bottom currents.
:bottom_mask_file	<character>	Gives the locations of the nodes where bottom currents are provided (1) and not provided (0)
:bottom_start_flow	yyyy mm dd hh mm ss	Time origin of the bottom flow files, e.g.: • bottom_start_flow 1997 10 5 12 45 0 • Each output step of the bottom flow file has a timestamp in seconds that is relative to a UTC time selected when the ADCIRC model control file is established. Input values required.
:wave_breaking		Instructs model to open a *.brk file for each *.wav file opened and to use breaking indices (Bi) estimate increased surf zone mixing.
:kew	<real>	Sets diffusion coefficient for waves. Default is 5. The horizontal eddy viscosity is adjusted by the factor:
:kev	<real>	Sets vertical diffusion coefficient. Default is 0.00859.
:etmin	<real>	Sets minimum vertical eddy viscosity. Default is 0.02.
:evmin	<real>	Sets minimum vertical eddy viscosity. Default is 0.0.
:source_to_datum		Sets the z-value of parcels released by sources to a z-value relative to datum rather than the bed
:numerical_scheme	<Integer>	Controls the order of the numerical scheme. Only valid options are 2 or 4.
:rhos	<real>	Sets the density of the bed sediments. Default is 2650.
:emrl_particle_set_guid	<character>	Used by SMS. Model will pass string to output files.
:emrl*		Model will ignore lines with these first four characters, except :emrl_particle_set_guid.
:end_data		Indicates the end of the keyword part of the file. The only contents of the file beyond this point should be the names of the wave files (if required).

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Sediment File
- Source File
- Trap File

PTM Sediment File

The PTM native sediment file contains the grain size information for the native sediments in terms of its D35, D50, and D90.

File Overview

The native sediments file contains the spatially-varying grain size information for the native sediments in terms of D35, D50 , and D90 . Frictional characteristics of the bed are computed with D90. The D50 value is used in the prediction of bed forms, in the determination of sediment mixing routines that influence reentrainment of deposited particles, and in the hiding and exposure routines that influence the critical shear stress of deposited particles. The D35 value is used in the determination of the suspended sediment transport if the van Rijn approach is selected for the centroid method. Non-erodible areas (e.g., rock outcroppings) can be identified by a negative input grain size. This absolute value of the grain size is treated as an effective roughness height, ks′ and ks″.

The native sediments filename is specified on the *Files* page of the *PTM model control*. A native sediments file can be generated by the SMS interface, if one is required, by pressing the **Create input file(s) from data...** button on the *Files* tab of the *PTM model control*. This will open the *Create PTM External Input Files* dialog.

The native sediment file can be in ASCII or XMDF format.

ASCII File Overview

An ASCII format native sediment file is used when using an ADCIRC (fort.14 or *.grd), or CMS-Flow format mesh file (see PCF File keywords :MESH_FORMAT and :SEDIMENT_FORMAT).

1) The first line is a comment line that is not read by PTM.
2) The second line contains the number of elements and the number of nodes of the mesh.
3) The next section contains a single line for each node, which gives the node number, easting (m), northing (m), D35 (mm), D50 (mm), and D90 (mm).
4) The file extension should be specified as .sediments (for SMS)

Example ASCII Native Sediment Data File

```
Node  ID    X(m)        Y(m)        D35(mm)     D50(mm)     D90(mm)
2704    1401
1          54010.00   60160.00   0.13        0.19        0.3
2          52008.90   60146.50   0.13        0.19        0.3
3          50007.80   60133.00   0.13        0.19        0.3
4          48006.70   60119.40   0.13        0.19        0.3
5          46005.60   60105.90   0.13        0.19        0.3

[Continued...]

1397       -6825.58   -408.412   0.13        0.19        0.3
1398       -6876.47   -481.815   0.13        0.19        0.3
1399       -6950.30   -414.314   0.13        0.19        0.3
1400       -6693.29   -533.139   0.13        0.19        0.3
1401       -6793.13   -526.903   0.13        0.19        0.3
```

In this example, there are two information lines (one comment line and one line with the number of elements and number of nodes) followed by 1,401 lines of data. Each node has sediment with a D35 of 0.13 mm, D50 of 0.19 mm, and D90 of 0.3 mm.

XMDF File Overview

An XMDF format native sediment file is used when using an XMDF format mesh file (see keywords :MESH_FORMAT and :SEDIMENT_FORMAT).

1) D35, D50, and D90 will be the same for every node when given as a property.
2) D35, D50, and D90 can be spatially varied when given as a dataset.

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Source File
- Trap File

PTM Source File

The PTM sediment source file contains the time varying location and parameters assigned to a source.

File Overview

The first line contains the number of Instant Mass Sources, NIMS. This is a single integer value. Any comments can follow on the line, e.g.:

```
1  Instant  Mass  Source(s)
```

This is followed by NIMS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label. In this example, there is one instant mass source, known as “My Instant Mass Source”, that has an id of zero and two instructions:

```
0   2   My  Instant  Mass  Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the point at that time (m), the mass of each parcel (kg), the horizontal radius of the source (m), the vertical radius of the source, the mass of the source (kg), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
2004  10  6  12  0  0  4.76400000000000e+003  2.63800000000000e+003  8  2  1  1  180
2e-007  0.8  2650  -1  -1  -1
2004  10  7  20  0  0  4.76400000000000e+003  2.63800000000000e+003  8  2  1  1  180
2e-007  0.8  2650  -1  -1  -1
```

The next set of data contains information for Point Mass Rate Sources. This begins with the number of Point Mass Rate sources, NPMRS. This is a single integer value. Any comments can follow on the line, e.g.:

```
2  Point  Mass  Rate  Source(s)
```

This is followed by NPMRS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label. e.g.:

```
0   4   My  Mass  Rate  Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the point at that time (m), the horizontal radius of the source (m), the vertical radius of the source, the mass rate of the source (kg/s), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
2004  10  6  10  0  0  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2
0.05  2e-006  0.8  2650  -1  -1  -1
2004  10  6  11  0  0  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2
0.05  2e-006  0.8  2650  -1  -1  -1
2004  10  6  11  0  1  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2  0
2e-006  0.8  2650  -1  -1  -1
2004  10  7  20  0  0  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2  0
2e-006  0.8  2650  -1  -1  -1
```

The next set of data contains information for Line Sources. This begines with the number of Line Sources, NLS. This is a single integer value. Any comments can follow on the line, e.g.:

```
1  Line  Source(s)
```

This is followed by NLS blocks of data. Each block of data contains:

- An identification line giving the line source id number, the number of insturctions, NIS, and a label, e.g.:

```
1   4   My  Horizontal  Line  Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the first end point of the line at that time (m), the x, y, and z location of the second end point of the line at that time (m), the horizontal radius of the source (m), the vertical radius of the source, the mass rate of the source (kg/s/m), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
2004  10  6  18  0  0  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  2.2e-005  2.5e-008  0.8
2650  -1  -1  -1
2004  10  6  19  0  0  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  2.2e-005  2.5e-008  0.8
2650  -1  -1  -1
2004  10  6  19  0  1  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  0         2.5e-008  0.8
2650  -1  -1  -1
2004  10  7  20  0  0  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  0         2.5e-008  0.8
2650  -1  -1  -1
```

The next set of data contains information for Polygon Sources. The points must be ordered using a standard counter-clockwise convention. This begins with the number of Polygon Sources, NPS. This is a single integer value. Comments can follow on the line, e.g.:

```
1  Polygon  Source(s)
```

This is followed by NPS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label, e.g.:

```
1  2  My  Polygon  Source
```

- NIS blocks of instructions. Each block of instructions contains:

A line with the time to issue the instruction (year, month, day, hour, minute, second). For each point in the polygon (1 point per line), the x, y, and z location of the point (m). The final line of the block of instructions contains the horizontal radius of the source (m), the vertical radius of the source (m), the mass rate of the source (kg/s/m^2), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
2000   1     1  8  0  0
 2451  4195  2
 2453  4560  2
 2183  4544  2
 2000  4342  2
 2261  4129  2
1      0     0  .00001  0.0003  0.4  2650  -1  -1  -1
```

The following list summarizes the rules for construction of a sediment source file:

- Data files must contain an entry stating the number of each type of sediment source, even if one or more sediment source types are not used.
- All sediment sources are time varying and are accompanied by a list of instructions.

- Linear interpolation between instructions is used to obtain source characteristics at a given time. All characteristics are interpolated. A constant source can be defined by two instructions with all data, except the start and stop times constant.
- The final instruction must have a time later than the stop time of the model.

Example Source File

```
1  Instant  Mass  Source(s)
 3   2   My  Instant  Mass  Source
  2004  10  6  12  0  0  4.76400000000000e+003  2.63800000000000e+003  8  2  1  1
180  2e-007  0.8  2650  -1  -1  -1
  2005  10  6  20  0  0  4.76400000000000e+003  2.63800000000000e+003  8  2  1  1
180  2e-007  0.8  2650  -1  -1  -1
2  Point  Mass  Rate  Source(s)
 2   4   My  Moving  Point  Mass  Rate  Source
  2004  10  6  16  0  0  4.09100000000000e+003  3.37500000000000e+003  8  2  3  3
0.05  5e-008  0.8  2650  -1  -1  -1
  2004  10  6  17  0  0  4.18600000000000e+003  1.90700000000000e+003  8  2  3  3
0.05  5e-008  0.8  2650  -1  -1  -1
  2004  10  6  17  1  0  4.09100000000000e+003  3.37500000000000e+003  8  2  3  3  0
5e-008  0.8  2650  -1  -1  -1
  2005  10  6  20  0  0  4.09100000000000e+003  3.37500000000000e+003  8  2  3  3  0
5e-008  0.8  2650  -1  -1  -1
 1   4   My  Point  Mass  Rate  Source
  2004  10  6  10  0  0  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2
0.05  2e-006  0.8  2650  -1  -1  -1
  2004  10  6  11  0  0  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2
0.05  2e-006  0.8  2650  -1  -1  -1
  2004  10  6  11  1  0  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2  0
2e-006  0.8  2650  -1  -1  -1
  2005  10  6  20  0  0  4.74500000000000e+003  3.76200000000000e+003  10  2  2  2  0
2e-006  0.8  2650  -1  -1  -1
1  Line  Source(s)
 4   4   My  Horizontal  Line  Source
  2004  10  6  18  0  0  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  2.2e-005  2.5e-008  0.8
2650  -1  -1  -1
  2004  10  6  19  0  0  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  2.2e-005  2.5e-008  0.8
2650  -1  -1  -1
  2004  10  6  19  1  0  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  0  2.5e-008  0.8  2650  -1
-1  -1
  2005  10  6  20  0  0  3.57500000000000e+003  3.84500000000000e+003  10
3.57500000000000e+003  1.57000000000000e+003  10  2  1  1  0  2.5e-008  0.8  2650  -1
-1  -1
1  Polygon  Source(s)
5  12  2  My  Polygon  Source
2004  10  6  12  0  0
 4.25800000000000e+003  1.88400000000000e+003  0
 4.13900000000000e+003  2.04000000000000e+003  0
 4.14400000000000e+003  2.19100000000000e+003  0
 4.19600000000000e+003  2.28900000000000e+003  0
 4.37700000000000e+003  2.34100000000000e+003  0
 4.53300000000000e+003  2.31000000000000e+003  0
 4.59500000000000e+003  2.27900000000000e+003  0
 4.64700000000000e+003  2.17500000000000e+003  0
```

```
 4.64700000000000e+003  2.08200000000000e+003  0
 4.58500000000000e+003  1.96800000000000e+003  0
 4.50700000000000e+003  1.91600000000000e+003  0
 4.37700000000000e+003  1.90000000000000e+003  0
2  1  1  0.05  0.0001  0.8  2650  -1  -1  -1
2005  10  6  20  0  0
 4.25800000000000e+003  1.88400000000000e+003  0
 4.13900000000000e+003  2.04000000000000e+003  0
 4.14400000000000e+003  2.19100000000000e+003  0
 4.19600000000000e+003  2.28900000000000e+003  0
 4.37700000000000e+003  2.34100000000000e+003  0
 4.53300000000000e+003  2.31000000000000e+003  0
 4.59500000000000e+003  2.27900000000000e+003  0
 4.64700000000000e+003  2.17500000000000e+003  0
 4.64700000000000e+003  2.08200000000000e+003  0
 4.58500000000000e+003  1.96800000000000e+003  0
 4.50700000000000e+003  1.91600000000000e+003  0
 4.37700000000000e+003  1.90000000000000e+003  0
2  1  1  0.05  0.0001  0.8  2650  -1  -1  -1
***************************************************************************************
*******************
* Guide:
***************************************************************************************
*******************
Number of Instant Mass Source(s)
  time(6), x_location, y_location, z_location, Pmass(kg), Hradius(m),
Vradius(m), mass(kg), grain_size(m), standard_dev(Phi-units), density(kg/m^3),
fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute), TAU_critical_deposition(N/m^2 (-1
= compute)
Number of Point Mass Rate Source(s)
  time(6), x_location, y_location, z_location, Pmass(kg), Hradius(m),
Vradius(m), rate(kg/s), grain_size(m), standard_dev(Phi-units), density(kg/m^3),
fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute), TAU_critical_deposition(N/m^2 (-1
= compute)
Number of Line Source(s)
  time(6), x1_loc, y1_loc, z1_loc, x2_loc, y2_loc, z2_loc, Pmass(kg),
Hradius(m), Vradius(m), rate(kg/s/m), grain_size(m), phi_sd(Phi-units),
density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute), TAU_critical_deposition(N/m^2 (-1
= compute)
Number of Polygon Source(s)
  time(6)
  For each point (1 point per line): x_location, y_location, z_location
  Pmass(kg), Hradius(m), Vradius(m), rate(kg/s/m^2), grain_size(m),
standard_dev(Phi-units), density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute), TAU_critical_deposition(N/m^2 (-1
= compute)
***************************************************************************************
*******************
* Source file written by SMS 10.0.0 Development
* SMS Build Date: Jun 20 2007
* Date saved: 06/20/07
* Time saved: 09:43:14
***************************************************************************************
*******************
```

Vertical Line Source Datums

The source file format has been changed to allow the specification of vertical line source datums:

Old Format

```
#  Line  Source(s)
 LineSourceID   #Instructions    SourceName
```

New Format

```
#  Line  Source(s)
 LineSourceID   #Instructions    SourceName   SourceDatum
```

The values available for the SourceDatum field are:

- beddatum (default)
- surfacedatum
- depthdistributed

The Sourcedatum has a default value of beddatum. If no datum is given in the file, then it is assumed to use the bed as the datum. Therefore, PTM will still process source files in the old format correctly.

Relative to the bed

This is the default and is what was already in PTM. Given the following input from the source file:

```
Time          X1          Y1          Z1          X2          Y2          Z2          PMass
Hrad          Vrad          Rate          Additional  Info
```

Relative to the water surface

If the SourceDatum is surfacedatum then the code takes Z1 and Z2 as relative to the water surface. The rate is given in kg/m/s.

Depth-distributed

If the SourceDatum is depth distributed then the code takes the Z1 and Z2 as percentages of the water depth. The rate is given in kg/s in this case. An example is if wanting to distribute the source over a fraction of the water depth, set z1=0.33 and z2=0.67 to introduce the source over the middle third of the water column. The depth-distributed source is taken as a percentage from the bed. So if z1 = 0 and z2 is 0.5, this suggests a segment that is the lower one half of the water depth.

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Sediment File
- Trap File

PTM Wave File

The PTM wave input files contain the time-varying phase-averaged wave information for the simulation in terms of significant wave height, Hs, peak period, Tp, and direction.

File Overview

The wave input files must contain information for each node in the mesh input file. Wave input files are usually time-varying. The wave input file can be in ASCII or XMDF format.

ASCII File Overview

ASCII format wave input files are used when using STWAVE or CMS-Wave (WABED) results.

Example ASCII Wave File

Examples of ASCII Wave files can be viewed in the CMS-Wave Wave Breaking File and CMS-Wave Wave Output File articles.

XMDF File Overview

Related Topics

- PTM Files

PTM Trap File

A PTM trap file defines areas into which parcels enter and are counted. The traps can be open or closed traps. When a parcel enters an open trap, it is free to leave the trap. When a parcel enters a closed trap, it is not allowed to leave the trap. If the trap is a single trap, each parcel is only counted once per simulation.

File Overview

The first line contains the number of line traps, NLT. This is a single integer value. Any comments can follow on the line, e.g.:

```
1  Line  Trap(s)
```

This is followed by NLT blocks of data. Each block of data contains:

- An identification line giving the trap id number, and a label. In this example, there is one line trap, known as “My Line Trap”, that has an id of two:

```
2  My  Line  Trap
```

- A single line giving the trap direction, trap bottom elevation, trap top elevation, open trap flag, and single count trap flag:

```
0   3   15   0   1
```

- Two lines giving the x, y location of the line trap endpoints:

```
3.99000000000000e+001  1.07600000000000e+002
4.92000000000000e+001  1.78200000000000e+002
```

The next set of data contains information for Polygon Traps. The points must be ordered using a standard counter-clockwise convention. This begins with the number of Polygon Traps, NPT. This is a single integer value. Comments can follow on the line, e.g.:

```
1  Polygon  Trap(s)
```

This is followed by NPS blocks of data. Each block of data contains:

- An identification line giving the trap id number, and a label, e.g.:

```
1  My  Polygon  Trap
```

- A single line giving the number of points in the trap, NP, trap bottom elevation, trap top elevation, open trap flag, and single count trap flag:

```
8  2  5  1  0
```

- For each point in the polygon (1 point per line), the point id, x, y location of the point:

```
1  1.78600000000000e+001  2.39300000000000e+001
2  3.54200000000000e+001  1.46700000000000e+001
3  7.29400000000000e+001  2.04200000000000e+001
4  7.91600000000000e+001  5.42600000000000e+001
5  6.11200000000000e+001  7.88400000000000e+001
6  2.93600000000000e+001  8.14000000000000e+001
```

```
7  9.08000000000000e+000  6.55900000000000e+001
8  3.97000000000000e+000  5.18600000000000e+001
```

Trap File Flag Values

Flag	Value(s)
Trap direction	-1 (decreasing x-coordinate), 0 (any direction), or 1 (increasing x-coordinate)
Open trap	0 (closed trap) or 1 (open trap)
Single count trap	0 (Count parcels every time they enter a trap) or 1 (Count parcels once per simulation)

If the keyword :LAST_STEP_TRAP is included in the Program Control File (PCF File) , the traps will only be active on the last time step of the PTM simulation.

Example Trap File

```
1  Line  Trap(s)
 0  My  Line  Trap
  0   2   12   1   0
   -5.28000000000000e+000  7.08600000000000e+001
   4.67600000000000e+001  8.57100000000000e+001
1  Polygon  Trap(s)
 1  My   Polygon  Trap   6  5  20  0  1
   1  3.55800000000000e+001  1.56300000000000e+001
   2  6.54300000000000e+001  1.45100000000000e+001
   3  7.64500000000000e+001  2.90400000000000e+001
   4  5.92100000000000e+001  5.42600000000000e+001
   5  3.36700000000000e+001  5.18600000000000e+001
   6  2.18500000000000e+001  3.06300000000000e+001
*************************************************************************************
********************
* Guide:
*************************************************************************************
********************
Number  of  Line  Traps(s)
For  each  Line  Trap:
 Trap  ID,  Trap  Name
  Trap  Direction,  Trap  Bottom,  Trap  Top,  Open  Trap,  Single  Trap
  First  node  x1_location,  y1_location
  Second  node  x2_location,  y2_location
Number  of  Polygon  Trap(s)
For  each  Polygon  Trap:
 Trap  ID,  Trap  Name
  Number  of  points  in  trap,  Trap  Bottom,  Trap  Top,  Open  Trap,  Single
Count  Trap
   For  each  point:  (1  point  per  line):  x_location,  y_location
*************************************************************************************
********************
*  Trap  Keyword  Values:
*************************************************************************************
********************
Line  Trap  Direction:
 -1  (decreasing  x-coordinate),  0  (any  direction),  or  1  (increasing  x-
coordinate)
 Default  value  is  0  (any  direction)
Open  Trap:
```

```
 0  (closed  trap)  or  1  (open  trap)
 Default  value  is  1  (open  trap)
Single  Count  Trap:
 0  (Count  parcels  every  time  they  enter  a  trap)  or  1  (Count  parcels  once
per  simulation)
 Default  value  is  0  (Count  parcels  every  time  they  enter  a  trap)
*************************************************************************************
********************
*  Trap  file  written  by  SMS  10.1.0  Development  Release
*  SMS  Build  Date:  Sep   9  2008
*  Date  saved:  09/09/08
*  Time  saved:  14:05:03
*************************************************************************************
********************
```

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Sediment File
- Source File

PTM Trap Output

Trap output files are generated by PTM when one or more traps have been specified (this can be done by linking to an existing *.trap file or SMS will create this file from a PTM coverage containing trap objects.

A PTM trap file is a tabular ASCII file which contains information about when parcels entered specific traps in a PTM simulation. The first row of the file must be a header row defining the contents of the file. Each row after the header defines a single entry event of a parcel into a trap

File Overview

The first line of a standard trap output files contains the header defining the standard five columns of output, e.g.:

```
STEP      DATE           TIME         PARTICLE      TRAP
```

The trap output file may also include additional columns which give extra information about the parcel, e.g.:

```
STEP      DATE           TIME         PARTICLE      TRAP         MASS
MEDIAN_SIZE         SOURCE
```

The header line is followed by entry lines. Each line defines an entry event of a parcel into a trap:

```
STEP      DATE         TIME           PARTICLE  TRAP    MASS    MEDIAN_SIZE   SOURCE
1178      2001/01/01   02:38:10.000   53        1       2       0.1           2
1215      2001/01/01   02:41:15.000   47        3       3       0.05          1
1233      2001/01/01   02:42:45.000   99        1       3       0.2           2
.
.
```

When SMS is asked to read a trap output file, the file is identified by the extension "*_count.out". If the extension has been modified, SMS can be instructed to read this file using the file import wizard.

When SMS reads the standard trap output file, a time series curve for each trap represented in the file is created counting the number of parcels which entered the trap and logging the time (in hours from the first parcel) that the parcels entered. This is a "Count" time series curve. These time series curves are monotonically increasing.

If one or more "Value" column is specified when reading the file, SMS also creates one curve for each trap showing the accumulated value from this column. In the example above, the "MASS" column could be mapped as a value and the cumulative mass of each parcel would be summed as it enters the trap.

If one or more "Filter" column is specified, SMS asks for filter ranges associated with this quantity. For example, to only count the parcels with a mean grain size below 0.15 mm, the "MEDIAN_SIZE" column above would be mapped as a "Filter" and a range of (0.00-0.15) specified in the *Filter Range* dialog that appears after the import wizard specification is complete. A parcel will be counted as in the range if it's value is greater than the minimum and less than or equal to the maximum). Ranges that are specified with the same minimum and maximum values are ignored. The values are always interpreted as doubles even when they represent an integer quantity such as "SOURCE". In this case, the ranges could be specified as (0.5-1.5) and (1.5-2.0) to filter out parcels from a specific source. SMS will create additional curves (both for count and cumulative value) for each trap for the parcels that fall into each filter range.

Therefore, the total number of curves created when reading a trap output file would be:

```
Num_Curves  =  (1  +  Num_Value_Columns)  *  (1  +  Number_Filter_Ranges)  *
Number_Traps
```

These curves can be viewed using the *Plot Wizard* and selecting *PTM Gage plots* .

Related Topics

- PTM

PTM Boundary Condition File

This file follows the ADCIRC fort.14 format. The only difference is that the header line will identify the file as a PTM_BC file with a version number. Here is an example:

```
PTM_BC  CMS-PTM  v2.1.025
```

When SMS reads the keyword (PTM_BC) in the header, it will then convert the file into a MIF/MID file, and read it into the GIS module. Blue is used to indicate open boundaries, and brown for closed boundaries. The reason for reading the file into the PTM interface and not the ADCIRC is in case the ADCIRC interface is not enabled or if there is an existing mesh.

Related Topics

- PTM Files

4.4. TUFLOW FV

TUFLOW FV

TUFLOW FV	
Model Info	
Model type	Two-dimensional (2D) flexible mesh finite volume flood, tide and water quality simulation software.
Developer	WBM BMT WBM (Australia)
Web site	www.tuflow.com
Tutorials	SMS Learning Center

TUFLOW FV (which stands for Two-dimensional Unsteady FLOW Finite Volume) is a flexible mesh finite volume numerical model that simulates hydrodynamic, sediment transport and water quality processes in oceans, coastal waters, estuaries and rivers. The model may be used for coastal and nearshore environments including beaches and coastlines as well as offshore environments such as estuaries, river entrances and deltas, and floodplains. Uses include modeling river flood flow, tsunami inundation (the finite-volume scheme is well suited for a tsunami's mixed sub/super-critical flow regimes), beach erosion, ocean pollution, and estuary flow.

Unlike the fixed square grids of TUFLOW Classic, the flexible triangular or quadrilateral mesh of TUFLOW FV allows users to modify mesh resolution spatially, seamlessly increasing the model resolution in areas of interest. This modelling approach reduces the number of computation cells needed in a model reducing run times. Additionally, TUFLOW FV can be run in parallel on multiple processors, threads, or computers.

The TUFLOW FV model can be added to a paid edition of SMS.

TUFLOW FV ENGINE

The TUFLOW computational engine computes 2D hydraulic solutions. The engine uses a macro style text-file input which allows the user to flexibly and efficiently control model configurations and simulations.

For more information see the TUFLOW FV webpage .

SMS Interface

The TUFLOW FV engine is interfaced in SMS through the generic model interface .

TUFLOW FV Menu

If the TUFLOW FV model paremeters have been correctly loaded into SMS, the *TUFLOW FV* menu is available when the Mesh module is active. The menu has the following commands:

- **Check Mesh** – Performs a general model check and will bring up the *Model Checker* dialog if errors are found.
- **Define Model** – Opens the model definitions. This is only accessible to the TUFLOW FV model developers.
- **Global Parameters** – Brings up the *TUFLOW FV Global Parameters* dialog where parameters for the model run are specified.
- **Assign BC** – Brings up the *TUFLOW FV Nodestring Boundary Conditions* dialog. Available when a boundary nodestring is selected.
- **Material Properties** – Brings up the *TUFLOW FV Material Properties* dialog.
- **Run TUFLOW FV** – Launches the TUFLOW FV model run.

Using the Model / Practical Notes

A TUFLOW licence is not required to run a TUFLOW FV model.

External Links

- TUFLOW FV Manual
- TUFLOW FV Wiki

5. Coastal Numeric Models

5.1. ADCIRC (Advanced Circulation)

Model

ADCIRC

ADCIRC	
Model Info	
Model type	Finite element hydrodynamic model for coastal oceans, inlets, rivers and floodplains.
Developer	Rick Luettich Joannes Westerink Randall Kolar Cline Dawson
Web site	http://www.adcirc.org
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • ADCIRC Several Sample problems can be found on the ADCIRC model developer's webpage

The ADCIRC (Advanced Circulation) model is a finite element hydrodynamic model for coastal oceans, inlets, rivers and floodplains. The initial developers of the code were Rick Luettich (University of North Carolina at Chapel Hill) and Joannes Westerink (University of Notre Dame). Other principal developers include Randall Kolar (University of Oklahoma at Norman) and Cline Dawson (University of Texas at Austin). Various other groups are involved in development and support around the country.

The ADCIRC model can be added to a paid edition of SMS.

Graphical Interface

SMS provides a graphical interface that is designed to visualize the projects being created, easily modify project parameters, and view the solutions produced by the ADCIRC model (for example, SMS 12.1 comes with ADCIRC version 50). See ADCIRC Graphical Interface for more information.

The ADCIRC Graphical Interface contains tools to create and edit an ADCIRC simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to ADCIRC. If a mesh has already been created for a ADCIRC simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to ADCIRC. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D mesh module menus and tools augmented by the *ADCIRC* menu . See ADCIRC Graphical Interface for more information.

ADCIRC Files

The list of files (both input and output) that may be associated with an ADCIRC simulation is quite long. The web site lists all of these with details describing format and purpose. A brief summary of the most common file types is included here:

Input files

- Required
  - fort.14: Grid file – Saved as *proj_name* .grd by SMS and copied to fort.14 for use in an ADCIRC run.
  - fort.15: Control file – Saved as *proj_name* .ctl by SMS and copied to fort.15 for use in an ADCIRC run.
- Optional
  - fort.13: Nodal or Spatial attributes
  - fort.19: Specified water surfaces (non-periodic elevation)
  - fort.20: Specified flow rates (non-periodic flow/flux such as a river)
  - fort.22, fort.2**: Meteorologic conditions (winds and atmospheric pressure) – Several formats depending on the option being used
  - fort.23: Wave radiation stress forcing

Output files

- Diagnostic
  - fort.6 – Screen output
  - fort.16 – General information
  - fort.18 – Parallel file
  - fort.33 – ITPACKV 2D Solver convergence issue information
- Global
  - fort.63 or fort.63.nc – Water surface at each node
  - fort.64 or fort.64.nc – Velocity components at each node
  - fort.53 – Elevation Harmonic constituents at each node
  - fort.54 – Velocity Harmonic constituents at each node
  - fort.73 – Atmospheric pressure at each node
  - fort.74 – Wind stress or velocity at each node
  - fort.80 – Parallel run file
- At observation stations
  - fort.61 – Water surface at stations
  - fort.62 – Velocity components at stations
  - fort.51 – Elevation Harmonic constituents at stations
  - fort.52 – Velocity Harmonic constituents at stations
  - fort.71 – Atmospheric pressure at stations
  - fort.72 – Wind stress or velocity at stations

Global Output Format

Generally, ADCIRC has the ability to output global data in three formats. These include:

- Standard ASCII – This format loops through the time steps including a value for each node (both node ID and solution value). These files are commonly very large (multiple GB) and can take a significant amount of time to load (sometimes as long as half an hour) because SMS has to process each time step of each dataset and build information about the solution for faster access. When instructing SMS to read a file of this type, SMS recognizes that the ASCII format is not efficient and converts the data to XMDF format in an "h5" file. The name of the XMDF file that will be created can be specified. Multiple standard ascii files can be combined into a single "h5" file. The new "h5" file hase the following advantages:

    - The "h5" file is binary and compressed so it is much smaller than the standard ASCII file.
    - SMS can read the "h5" file almost instantaneously because all of the time step information is already compiled and a single time step is retrieved rather than processing the entire dataset.
- Sparse ASCII – This format loops through the time steps includes a default value for the time step and a number of nodes that don't have this default. Most commonly, the default would be -9999 indicating dry nodes. The file then includes the *exceptions* consisting of node ID and solution values for nodes that are not the default value. These lines are identical to the value lines in the standard ASCII format. (This format is supported in SMS starting at version 11.2) SMS converts sparse ASCII files to XMDF files just as it does the standard ASCII files.
- NetCDF – This format is a binary library format using the NetCDF library. The data can be viewed using an HDF viewer. (This format is supported in SMS starting at version 11.2)

(*Note:* ADCIRC documentation references a global binary format as an option. These options correspond to NOUT** values of 2 and -2 generally. There is no evidence that this option is functional in the current version of ADCIRC. It has been removed from the SMS interface.)

Functionality

ADCIRC is a system of computer programs for solving time dependent, free surface circulation and transport problems in two and three dimensions. These programs utilize the finite element method in space allowing the use of highly flexible, unstructured grids. Typical ADCIRC applications have included: (i) modeling tides and wind driven circulation, (ii) analysis of hurricane storm surge and flooding, (iii) dredging feasibility and material disposal studies, (iv) larval transport studies, (v) near shore marine operations.

ADCIRC (which models wetting and drying) has also been used to model the propagation and inundation of tsunami waves as shown in this paper. ADCIRC was also coupled with SWAN outside of SMS to model tsunami inundation in this paper.

For more information about the ADCIRC model visit www.adcirc.org .

Running ADCIRC in Parallel (PADCIRC)

SMS 11.2 and later allows running ADCIRC in parallel (PADCIRC) on a single machine to take advantage of multiple cores. This can greatly speed up the runs. Use the following steps to set up a PADCIRC model:

1) Download the MPI (Message Passing Interface) executable found on the SMS downloads page in the ADCIRC Basic Utilities zip folder.
2) To avoid run issues, install the MPI as an administrator and run with administrative privileges.
3) In SMS, select *Edit* | **Preferences** to bring up the *Preferences* dialog.
4) Select the *File Locations* tab
5) In the *Other Files* section, set the file path for the *MPIEXEC* executable.
6) Open the *ADCIRC Model Control* dialog.
7) Select the *General* tab in the model control.
8) In the *Processors* section put the desired number of processors into the *Computational* field.
    1) To find out how many processors (CPU's or cores) the machine can use, right-click on the windows task bar and select **Task Manager** to bring up the *Windows Task Manager* .
    2) In the task manager, select the *Performance* tab and count the number of boxes under the *CPU Usage History* section or check the number next to *Cores* . This will show how many physical processors the computer has. It may also list the number of logical (or virtual) processors if hyper-threading technology is enabled (it may have twice as many CPU's show up as actual physical cores in the hardware).
    3) Note that running PADCIRC using hyperthreading (specifying more logical processors than physical cores) has not been shown to significantly reduce run time.

9) In SMS, run the ADCIRC model and PADCIRC.exe will run instead.

Saving ADCIRC

When using *File* | **Save As...** the following files get saved in the SMS file.

- *.mat referenced to new save location
- *.map referenced to new save location
- *.grd referenced to new save location
- *.ctl referenced to new save location
- *.h5 referenced to new save location
- *.dat referenced to new save location

Using the Model / Practical Notes

It's important to note that ADCIRC is configured to accept only one calendar year's data, so it is not possible to combine meteorological data from two different calendar years into a single file and then run it (e.g. Dec 2015 and Jan 2016 data could not be combined into a single ADCIRC model).

There is an ADCIRC listserv that may be useful to keep up-to-date about the latest releases of ADCIRC and to post any questions about ADCIRC. It is adcirc@listserv.unc.edu . If wanting to join, please email Crystal Fulcher .

Related Topics

- SMS Models
- LTEA – Linear Truncation Error Analysis
- ADCIRC Database

External Links

- ADCIRC Home page
- 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 [85]
- Jun 2001 ERDC/CHL CHETN-IV-32 Leaky Internal-Barrier Normal-Flow Boundaries in the ADCIRC Coastal Hydrodynamics Code [86]
- 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 [87]
- Dec 1999 Coastal Engineering Technical Note IV-21 Surface-Water Modeling System Tidal Constituents Toolbox for ADCIRC [88]
- ADCIRC wiki hosted by Seahorse Coastal Consulting
- Glacier Bay Test Case by Dave F. Hill
- Assessment of ADCIRC's Wetting and Drying Algorithm

ADCIRC Database

The ADCIRC database includes a grid file and a harmonics file that uses an *.exe extractor to put information into SMS. In order to work with the ADCIRC Database, it's necessary to download the files separately. The needed files are:

1) Atlantic Database and Grid Files – 74.9 MB. Covers all waters west of the 60 deg W Meridian and east of the North American continent.
    - adcircnwattides.exe (utility)
    - ec2001.grd (adcirc grid file)
    - ec2001.tdb (adcirc harmonic output file)

- tides.in (sample file)
- tides_ec2001.f (source code)

2) Pacific Database and Grid Files – 86.2 MB. Covers waters along the Northern Mexican Coast, the US West Coast, the Canadian West Coast and the Alaskan South Coast.

- adcircnepactides.exe (utility)
- enpac2003.grd (adcirc grid file)
- enpac2003.tdb (adcirc harmonic output file)
- tides.in (sample file)
- tides_enpac2003.f (source code)

These files should be unzipped and placed in C:\Program Files\SMS X.X\models (or a location of choice). It is then necessary to tell SMS where these files are located by going to *Edit* | **Preferences** and selecting the *File Locations* tab in the dialog. In the *File Locations* tab, the model executables are set under "NE Pacific Tidal Databases" and "NW Atlantic Tidal Databases" in the *Model Executables* section.

Related Topics

- ADCIRC
- ADCIRC Tidal/Harmonics

ADCIRC Files

Here are tables of all the availible input and output files with the ADCIRC model. All formats might not be supported by SMS. Currently SMS supports ASCII and NetCDF file formats.

- For more information on the formating of these files click the link on the file name to go to the ADCIRC online documentation (ADCIRC.org)

Input Files

Name	Description	Required/Conditional
fort.14	Grid and Boundary Information File	Required
fort.15	Model Parameter and Periodic Boundary Condition File	Required
fort.10	Passive Scalar Transport Input File	Conditional
fort.11	Density Initial Condition Input File	Conditional
fort.13	Nodal Attributes File	Conditional
fort.19	Non-periodic Elevation Boundary Condition File	Conditional
fort.20	Non-periodic, Normal Flow Boundary Condition File	Conditional
fort.22	Single File Meteorological Forcing Input	Conditional
fort.200,...	Multiple File Meterological Forcing Input	Conditional
fort.23	Wave Radiation Stress Forcing File	Conditional
fort.24	Self Attraction/Earth Load Tide Forcing File	Conditional
fort.67 or fort.68	2DDI Hot Start Files	Conditional

Output Files

Name	Description
fort.6	Screen Output

fort.16	General Diagnostic Output
fort.33	Iterative Solver ITPACKV 2D Diagnostic Output
fort.41	3D Density, Temperature and/or Salinity at Specified Recording Stations
fort.42	3D Velocity at Specified Recording Stations
fort.43	3D Turbulence at Specified Recording Stations
fort.44	3D Density, Temperature and/or Salinity at All Nodes in the Model Grid
fort.45	3D Velocity at All Nodes in the Model Grid
fort.46	3D Turbulence at All Nodes in the Model Grid
fort.51	Elevation Harmonic Constituents at Specified Elevation Recording Stations
fort.52	Depth-averaged Velocity Harmonic Constituents at Specified Velocity Recording Stations
fort.53	Elevation Harmonic Constituents at All Nodes in the Model Grid
fort.54	Depth-averaged Velocity Harmonic Constituents at All Nodes in the Model Grid
fort.55	Harmonic Constituent Diagnostic Output
fort.61	Elevation Time Series at Specified Elevation Recording Stations
fort.62	Depth-averaged Velocity Time Series at Specified Velocity Recording Stations
fort.63	Elevation Time Series at All Nodes in the Model Grid
fort.64	Depth-averaged Velocity Time Series at All Nodes in the Model Grid
fort.67, fort.68	Hot Start Output
fort.71	Atmospheric Pressure Time Series at Specified Meteorological Recording Stations
fort.72	Wind Velocity Time Series at Specified Meteorological Recording Stations
fort.73	Atmospheric Pressure Time Series at All Nodes in the Model Grid
fort.74	Wind Stress or Velocity Time Series at All Nodes in the Model Grid
fort.81	Depth-averaged Scalar Concentration Time Series at Specified Concentration Recording Stations
fort.83	Depth-averaged Scalar Concentration Time Series at All Nodes in the Model Grid
fort.91	Depth-averaged Density Fields at Specified Recording Stations
fort.93	Depth-averaged Density Fields at All Nodes in the Model Grid

Related Topics

- Run ADCIRC

ADCIRC Graphical Interface

The ADCIRC Graphical Interface includes tools to assist with creating, editing and debugging an ADCIRC model. The ADCIRC interface exists in the 2D Mesh Module .

Model Control

The *ADCIRC Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In ADCIRC boundary conditions are defined on nodestrings . The default boundary condition is a closed boundary (no flow). See ADCIRC BC Nodestrings for more information.

Running the Model

The ADCIRC files are written automatically with the SMS project file or can be saved separately using the *File* | **Save ADCIRC** or *File* | **Save As** menu commands. See ADCIRC Files for more information on the files used for the ADCIRC run.

ADCIRC can be launched from SMS using the *ADCIRC* | **Run ADCIRC** menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *ADCIRC* | **Model Check** menu command.

ADCIRC Menu

See ADCIRC Menu for more information.

Related Topics

- Boundary Types
- Boundary Conditions – tidal forcing
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Spatial Attributes
- Steering

ADCIRC Menu

The *ADCIRC* menu becomes active in the mesh module when a mesh has been created from an ADCIRC map coverage.

The following menu commands are available in the *ADCIRC* menu:

Assign BC

(Boundary Condition) Opens the *ADCIRC Arc/Nodstring Attributes* dialog.

Spatial Attributes

Opens the *Spacial Attributes* dialog.

Create Observation Station

Opens the *Recording Station* dialog.

Mesh Generation Toolbox

Brings up the *Mesh Generation Toolbox* where a mesh generation method can be selected for SMS to run. Currently, SMS supports the Localized Truncation Error Analysis (LTEA) method.

Model Check

Starts the model checker .

Model Control

Opens the *ADCIRC Model Control* dialog.

Run ADCIRC

Launches the ADCIRC engine and generates output files. See below for more information.

Obsolete Commands

The following commands are no longer included in current versions of SMS.

Define Nodal Attributes

Assign Nodal Attributes

Run ADCIRC

Presently, ADCIRC uses a specific naming convention for its input and output files. Therefore, before ADCIRC can start, the basic input files must be present in the working directory, which SMS takes care of automatically.

To run ADCIRC:

1) Select *ADCIRC* | **Model Control** .
2) Select the *General* Tab.
3) Name the project in the *Project title:* field and enter a name in the *Run ID:* field as well.
4) Press **OK** .
5) Select *ADCIRC* | **Run ADCIRC** .
6) If the name of the ADCIRC executable does not appear, click the folder icon, locate the ADCIRC executable, and click **OK** .

The ADCIRC model wrapper appears and gives status for the time steps while the model runs. On a typical desktop machine, this will take around 15 minutes.

Once the ADCIRC run has completed, there will be several new files created. See ADCIRC Files for more information about the files generated.

Related Topics

- ADCIRC Coverage
- ADCIRC Mesh
- Steering

ADCIRC Mesh

ADCIRC computations are performed on a finite element mesh. ADCIRC meshes can only contain linear triangular elements. For more information about the mesh structure and using meshes see Mesh Module .

ADCIRC meshes are often generated using scalar density paving which helps define meshes that transition well between different sizes of elements (see Mesh Generation). Size functions used for scalar density paving can be defined using Linear Truncation Error Analysis (LTEA).

Example ADCIRC Mesh File (fort.14 or *.grd)

```
My  Mesh  File
200   125
      1     0.0000000000  5000.0000000000    10.0000000000
      2     0.0000000000  4500.0000000000    10.0000000000
      3     0.0000000000  4000.0000000000    10.0000000000
      4     0.0000000000  3500.0000000000    10.0000000000
      5     0.0000000000  3000.0000000000    10.0000000000
      6     0.0000000000  2500.0000000000    10.0000000000
      7     0.0000000000  2000.0000000000    10.0000000000
      8     0.0000000000  1500.0000000000    10.0000000000
      9     0.0000000000  1000.0000000000    10.0000000000
```

```
   10     0.0000000000   500.0000000000    10.0000000000
   11     0.0000000000     0.0000000000    10.0000000000
   12   387.2875168500  4679.5600328850    10.0000000000
   13   500.0000000000  5000.0000000000    10.0000000000
   14   448.0608478950  4243.2347889200    10.0000000000
   15   439.9566817750  3740.3946708250    10.0000000000
   16   438.0183235050  3237.7140888100    10.0000000000
   17   434.2179653200  2728.5012612700    10.0000000000
   18   434.4501409550  2223.5360522450    10.0000000000
   19   445.6728191600  1709.4083456900    10.0000000000
   20   441.7355135250  1217.0963461800    10.0000000000
   21   446.7690604800   665.5453978200    10.0000000000
   22   500.0000000000     0.0000000000    10.0000000000
   23   879.4691569000  4527.8977321650    10.0000000000
   24  1000.0000000000  5000.0000000000    10.0000000000
   25   915.9217524550  3970.1981044250    10.0000000000
   26   908.0623079800  3485.8982158350    10.0000000000
   27   865.4508483900  2975.6001854250    10.0000000000
   28   870.3584492700  2470.6287334900    10.0000000000
   29   865.3646748800  1963.5880264200    10.0000000000
   30   958.2489022100  1408.2747297650    10.0000000000
   31   846.2862793400   940.9107967100    10.0000000000
   32   894.5851194050   468.8869156450    10.0000000000
   33  1000.0000000000     0.0000000000    10.0000000000
   34  1407.4389323650  4568.9272855350    10.0000000000
   35  1319.0864158900  4152.9138196250    10.0000000000
   36  1500.0000000000  5000.0000000000    10.0000000000
   37  1519.2308151400  3630.3323211050    10.0000000000
   38  1289.5949017700  3195.6810638750    10.0000000000
   39  1309.3380986300  2727.7292707550    10.0000000000
   40  1305.6561222300  2246.1068999650    10.0000000000
   41  1287.1975618200  1772.0435687850    10.0000000000
   42  1581.6880220050  1382.8502340350    10.0000000000
   43  1352.9489540350   880.7930319200    10.0000000000
   44  1338.3794076000   386.3892121500    10.0000000000
   45  1500.0000000000     0.0000000000    10.0000000000
   46  1912.7374514350  4563.7881224100    10.0000000000
   47  1799.9414465350  4111.9865086650    10.0000000000
   48  2000.0000000000  5000.0000000000    10.0000000000
   49  2095.6095101550  3684.4249560900    10.0000000000
   50  2029.7448911600  3256.1867318100    10.0000000000
   51  1724.4261905750  3002.3231849500    10.0000000000
   52  1757.6153478550  2485.5439440750    10.0000000000
   53  1700.5538408750  1971.3218032750    10.0000000000
   54  2022.3261809400  1718.2985960850    10.0000000000
   55  2115.1877830250  1308.3558784300    10.0000000000
   56  1908.2173673400   884.8414132300    10.0000000000
   57  1796.4311279300   444.4909875000    10.0000000000
   58  2000.0000000000     0.0000000000    10.0000000000
   59  2421.8299088200  4565.2347923850    10.0000000000
   60  2296.2815207500  4136.2010145200    10.0000000000
   61  2500.0000000000  5000.0000000000    10.0000000000
   62  2548.1854914450  3736.1664761600    10.0000000000
   63  2245.7592831150  2715.5365617900    10.0000000000
   64  2156.5989885850  2163.7822188800    10.0000000000
```

```
   65  2576.1588612050  1255.8568827850    10.0000000000
   66  2424.8132302500   862.3361492400    10.0000000000
   67  2296.5804477550   435.0891291250    10.0000000000
   68  2500.0000000000     0.0000000000    10.0000000000
   69  2928.9231473700  4562.5355493550    10.0000000000
   70  2813.0832006250  4131.9646648200    10.0000000000
   71  3000.0000000000  5000.0000000000    10.0000000000
   72  3005.2616900500  3650.6086203300    10.0000000000
   73  3079.6575104700  3155.0929235250    10.0000000000
   74  3026.1311773200  1347.8384132750    10.0000000000
   75  3074.0212166200  1837.0105551000    10.0000000000
   76  2941.1113740800   868.2621980900    10.0000000000
   77  2803.0270930300   435.0218954200    10.0000000000
   78  3000.0000000000     0.0000000000    10.0000000000
   79  3428.1706765000  4553.5572992900    10.0000000000
   80  3334.2421005900  4097.5833709050    10.0000000000
   81  3500.0000000000  5000.0000000000    10.0000000000
   82  3585.5001362100  3580.4710896600    10.0000000000
   83  3644.1599668700  3020.1398601150    10.0000000000
   84  3268.7966206450  2718.6831817150    10.0000000000
   85  3591.9118685300  1432.2295580150    10.0000000000
   86  3444.7797743450   901.6282802800    10.0000000000
   87  3639.1938445900  1996.7994288000    10.0000000000
   88  3191.2693888900  2293.1529447050    10.0000000000
   89  3312.8201976500   437.8216047050    10.0000000000
   90  3500.0000000000     0.0000000000    10.0000000000
   91  3875.4018613600  4597.6915792100    10.0000000000
   92  3897.1163185450  4126.3881442500    10.0000000000
   93  4000.0000000000  5000.0000000000    10.0000000000
   94  4132.8748876900  3728.5069540050    10.0000000000
   95  4108.4300319250  3272.5630823750    10.0000000000
   96  4124.3068688700  2760.4946533350    10.0000000000
   97  3697.0596781500  2499.0463538250    10.0000000000
   98  4103.2022573000  1742.3164195700    10.0000000000
   99  4136.5962459150  1283.2373595850    10.0000000000
  100  3961.1675874700   903.0618231000    10.0000000000
  101  3813.1576567050   450.2947123950    10.0000000000
  102  4125.8315088050  2248.5711365450    10.0000000000
  103  4000.0000000000     0.0000000000    10.0000000000
  104  4367.7984401650  4498.4664948950    10.0000000000
  105  4521.7565602000  4046.3826573000    10.0000000000
  106  4500.0000000000  5000.0000000000    10.0000000000
  107  4598.8223028500  3503.3617712150    10.0000000000
  108  4561.3391885400  2955.6940550700    10.0000000000
  109  4563.9298111000  2441.5637730600    10.0000000000
  110  4561.8541878800  1933.3440965250    10.0000000000
  111  4569.5749317300  1429.5434022850    10.0000000000
  112  4527.5800133650   930.2255863050    10.0000000000
  113  4350.9621391150   487.6105782500    10.0000000000
  114  4500.0000000000     0.0000000000    10.0000000000
  115  5000.0000000000  4500.0000000000    10.0000000000
  116  5000.0000000000  4000.0000000000    10.0000000000
  117  5000.0000000000  5000.0000000000    10.0000000000
  118  5000.0000000000  3500.0000000000    10.0000000000
  119  5000.0000000000  3000.0000000000    10.0000000000
```

```
   120  5000.0000000000  2500.0000000000    10.0000000000
   121  5000.0000000000  2000.0000000000    10.0000000000
   122  5000.0000000000  1500.0000000000    10.0000000000
   123  5000.0000000000  1000.0000000000    10.0000000000
   124  5000.0000000000   500.0000000000    10.0000000000
   125  5000.0000000000     0.0000000000    10.0000000000
  1     3      1     12     13
  2     3     12      1      2
  3     3     12      2     14
  4     3      2      3     14
  5     3     15     14      3
  6     3      3      4     15
  7     3      4     16     15
  8     3      4      5     16
  9     3     17     16      5
 10     3      5      6     17
 11     3     18     17      6
 12     3      6      7     18
 13     3     19     18      7
 14     3     19      7      8
 15     3      8     20     19
 16     3     20      8      9
 17     3      9     21     20
 18     3      9     10     21
 19     3     22     21     10
 20     3     10     11     22
 21     3     13     12     23
 22     3     23     12     14
 23     3     13     23     24
 24     3     14     25     23
 25     3     14     15     25
 26     3     25     15     26
 27     3     15     16     26
 28     3     26     16     27
 29     3     17     27     16
 30     3     17     28     27
 31     3     17     18     28
 32     3     28     18     29
 33     3     29     18     19
 34     3     19     30     29
 35     3     30     19     20
 36     3     20     31     30
 37     3     20     21     31
 38     3     31     21     32
 39     3     22     32     21
 40     3     32     22     33
 41     3     24     23     34
 42     3     23     35     34
 43     3     23     25     35
 44     3     36     24     34
 45     3     35     25     37
 46     3     25     26     37
 47     3     37     26     38
 48     3     27     38     26
 49     3     38     27     39
```

50	3	39	27	28
51	3	28	40	39
52	3	40	28	29
53	3	40	29	41
54	3	29	30	41
55	3	41	30	42
56	3	42	30	43
57	3	31	43	30
58	3	31	32	43
59	3	43	32	44
60	3	33	44	32
61	3	33	45	44
62	3	36	34	46
63	3	46	34	47
64	3	34	35	47
65	3	35	37	47
66	3	48	36	46
67	3	47	37	49
68	3	49	37	50
69	3	50	37	51
70	3	37	38	51
71	3	38	39	51
72	3	51	39	52
73	3	39	40	52
74	3	40	53	52
75	3	53	40	41
76	3	41	42	53
77	3	42	54	53
78	3	54	42	55
79	3	42	56	55
80	3	43	56	42
81	3	57	56	43
82	3	57	43	44
83	3	44	45	57
84	3	58	57	45
85	3	59	48	46
86	3	46	60	59
87	3	47	60	46
88	3	60	47	49
89	3	59	61	48
90	3	49	62	60
91	3	62	49	50
92	3	51	63	50
93	3	63	51	52
94	3	52	64	63
95	3	52	53	64
96	3	64	53	54
97	3	65	54	55
98	3	55	66	65
99	3	56	66	55
100	3	56	67	66
101	3	56	57	67
102	3	58	67	57
103	3	67	58	68
104	3	61	59	69

```
105     3    70    69    59
106     3    59    60    70
107     3    60    62    70
108     3    69    71    61
109     3    70    62    72
110     3    72    62    73
111     3    74    75    65
112     3    65    76    74
113     3    76    65    66
114     3    76    66    77
115     3    67    77    66
116     3    77    67    68
117     3    77    68    78
118     3    79    71    69
119     3    69    80    79
120     3    70    80    69
121     3    70    72    80
122     3    81    71    79
123     3    80    72    82
124     3    72    73    82
125     3    82    73    83
126     3    73    84    83
127     3    75    74    85
128     3    85    74    86
129     3    76    86    74
130     3    88    75    87
131     3    87    75    85
132     3    76    89    86
133     3    76    77    89
134     3    78    89    77
135     3    78    90    89
136     3    79    91    81
137     3    79    92    91
138     3    92    79    80
139     3    80    82    92
140     3    91    93    81
141     3    94    92    82
142     3    94    82    95
143     3    95    82    83
144     3    95    83    96
145     3    83    97    96
146     3    83    84    97
147     3    84    88    97
148     3    87    85    98
149     3    98    85    99
150     3    85   100    99
151     3   100    85    86
152     3   100    86   101
153     3    86    89   101
154     3    97    88    87
155     3    97    87   102
156     3   102    87    98
157     3   101    89    90
158     3    90   103   101
159     3   104    93    91
```

```
 160     3    104     91     92
 161     3    104     92    105
 162     3     92     94    105
 163     3    106     93    104
 164     3     94    107    105
 165     3    107     94     95
 166     3     95    108    107
 167     3     96    108     95
 168     3     96    109    108
 169     3     96    102    109
 170     3     97    102     96
 171     3    110    102     98
 172     3    110     98    111
 173     3     98     99    111
 174     3    111     99    112
 175     3    100    112     99
 176     3    100    113    112
 177     3    100    101    113
 178     3    113    101    103
 179     3    102    110    109
 180     3    103    114    113
 181     3    104    115    106
 182     3    115    104    105
 183     3    105    116    115
 184     3    116    105    107
 185     3    117    106    115
 186     3    116    107    118
 187     3    118    107    119
 188     3    119    107    108
 189     3    119    108    120
 190     3    120    108    109
 191     3    109    121    120
 192     3    109    110    121
 193     3    122    121    110
 194     3    110    111    122
 195     3    123    122    111
 196     3    111    112    123
 197     3    123    112    124
 198     3    124    112    113
 199     3    114    124    113
 200     3    124    114    125
1  =  Number  of  open  boundaries
11  =  Total  number  of  open  boundary  nodes
11  =  Number  of  nodes  for  open  boundary  1
1
2
3
4
5
6
7
8
9
10
11
```

```
4 = Number of land boundaries
44 = Total number of land boundary nodes
11 1 = Number of nodes for land boundary 1
50
62
73
84
88
75
65
54
64
63
50
11 22 = Number of nodes for land boundary 2
125
124
123
122
121
120
119
118
116
115
117
11 0 = Number of nodes for land boundary 3
11
22
33
45
58
68
78
90
103
114
125
11 0 = Number of nodes for land boundary 4
117
106
93
81
71
61
48
36
24
13
1
```

Related Links

- ADCIRC
- Coverage

- Linear Truncation Error Analysis (LTEA)
- Steering

ADCIRC Model Control

The *ADCIRC Model Control* dialog is where important project parameters are chosen and defined. The six different tabs are listed below.

- *General* – Contains such options as model type, cold/hot start, terms, maximum number of iterations, bottom stress/friction, etc.
- *Timing* – Defines simulation run time, start time, and time step options
- *Files* – Specifies which input and output files are desired for the project
- *Tidal/Harmonics* – Defines tidal constituents and harmonic analysis options
- *Wind* – Enables specific wind types and their options
- *Sediment Options* – Enables sediment transport options

General Tab

Here is a list of items found on the *General* tab:

- *Simulation Titles* – Includes the Project Title and Run ID.
- *Model Type* – 2D DI, 3D VS, 3D DSS. The two dimensional depth integrated is the only option that is recommended for commercial applications at this time.
- *Initial Values* – Cold Start, Hot Start 1, Hot Start2. This option allows choosing a file to initialize the values of water level and velocity for a simulation. The files must have been saved from a previous run using the same finite element grid.
- *Coriolis Option* – Constant, Variable. The type of Coriolis forcing used does not impact run time for ADCIRC at all. Variable should be used when the coordinate system is geographic. In this case, ADCIRC will load a Coriolis coefficient for each node based on the latitude of the node. If wanting to run ADCIRC in rectilinear coordinates, specify a constant Coriolis option because the model does not support reprojection from rectilinear to geographic to determine individual latitudes for each node.
- *Minimum Angle For Tangential Flow*
- *Terms* – Finite Amplitude Terms, Advective Terms, Time Derivative Terms
    - *Wetting/Drying* – Activating this option will cause an **Options** button to appear. Clicking this button will bring up the *Wetting/Drying Parameters* dialog.
- *Solver Type* – Lumped, Direct Banded, Iterative JCG – The Iterative JCG is the default and recommended solver. The other options are in the interface for backward compatibility and analytic tests.
- *Absolute Convergence Criteria*
- *Maximum Number of Iterations per Time Step*
- *Generalized Properties* – Wave Continuity, Lateral Viscosity
- *Bottom Stress/Friction* – Constant Linear, Constant Quadratic, Constant Hybrid, Varying Linear, Varying Quadratic, Friction Coefficient – The "Constant Linear" option should only be used for analytical cases when verifying the code. Linear friction causes overdamping in deep water. For "Constant Quadratic" the value of TAU0 should be set based on the principle depth. A value of TAU0 = 0.005 is suggested for deeper water (greater than 10 m depth). For shallow water, TAU0 = 0.02 is recommended. If the domain includes both shallow and deep water, consider the varying quadratic option.

Timing Tab

Important parameters to be aware of:

- *Ramp Function Value* – This value is a period of time, starting at the beginning of a simulation, that allows the full forcing of the tidal constituents to be gradually applied to the model. This prevents any issues or negative effects from occurring that could harm the results of a simulation. [1]
- *Reference time* – Specifies that the extracted tidal constituents are earlier or later than the start time of the simulation (see ADCIRC documentation Reftime).
- *Start Day* – The start time of a project
- *Time Step* – Time interval between successive measurements
- *Run Time* – Total length of time a simulation is run
- *End Time* – Day when simulation should finish

1 Militello, A., and Zundel, A. K. (1999). "Surface-water modeling system tidal constituents toolbox for ADCIRC," Coastal Engineering Technical Note CETN IV-21, U.S. Army Engineer Research and Development Center, Vicksburg, MS.

Files Tab

This tab contains two spreadsheets. The first of which is concerned with the input information ADCIRC can use before a simulation is run. Turn on or off any of the enabled input files by checking the checkbox in the *Input* column. Some of the more common input files that are used are:

- **Initial Water Level** – Provides initial water elevation levels
- **Hotstart 1 (fort.67)** – See ADCIRC Hot Start Files (fort.67 & fort.68)
- **Hotstart 2 (fort.68)** – See ADCIRC Hot Start Files (fort.67 & fort.68)

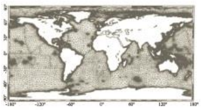
The second spreadsheet on this tab lists all of the possible output/solution files that ADCIRC can produce. This spreadsheet works similarly to the *Input Files* spreadsheet. Turn on or off any of the output files listed by clicking in a checkbox in the *Output* column. Some of the files also require a start time, end time, or frequency to be specified for results to be produced. The start time and end time do not have to be the same as the start and end time of the entire simulation, but they do need to be somewhere within that time frame.

Tidal/Harmonics Tab

Tidal constituents are variations in tides that are created by different frequencies of astronomical forcing. They arise due to the gravitational influences of the Moon and Sun on the Earth, the tilt of the rotational axis of the Earth, the elliptical shape of the Moon's orbit around the Earth, the shape of the Earth's orbit around the Sun, and other such factors. Many constituents have been defined and are classified based on their cycle lengths. Most of the tidal constituents used in the ADCIRC interface of SMS are either diurnal (one cycle per day) or semidiurnal (two cycles per day) in nature. Harmonic constituents are those variations that have periods of less than half a day. For a table of commonly used constituents see Principal Tidal Constituents .

SMS uses the following databases to provide the tidal constituent data to an ADCIRC simulation:

- LeProvost – The LeProvost database is a set of *.legi files which provide amplitude and phase information. The LeProvost database can be downloaded at: sms.aquaveo.com/leprovost.zip .

  An image of the LeProvost tidal database domain coverage can be found here –  [2]

- ADCIRC – The ADCIRC database includes a grid file and a harmonics file that uses an *.exe extractor to put information into SMS. The ADCIRC database can be downloaded at: sms.aquaveo.com/adcirctides.zip . More information about the ADCIRC database can be obtained at the ADCIRC web page: ADCIRC Tides Databases . The ADCIRC database is only valid for projections in North America.

The path to each database on a computer must be specified in the *Preferences* dialog by going to *Edit* | **Preferences** then selecting the *File Locations* tab. The two tidal constituent databases (LeProvost and ADCIRC) are only accessible if the SMS project has a global (not local) display projection.

In the *New Constituent* dialog, the radio buttons that allow selecting either of these databases are either enabled or grayed out depending on the display projection setting.

2 Le Provost, C., Genco, M. L., and Lyard, F. (1995). "Modeling and predicting tides over the World Ocean," Quantitative Skill Assessment for Coastal Ocean Models, Coastal and Estuaring Studies, Vol. 47, pp 175-201.

Wind Tab

ADCIRC can model and compute wind velocities and stresses. The ADCIRC model is able to use wind data from a variety of different file types. The SMS interface is not able to read and create all of the available wind data file types. There are three types that SMS is able to read and create. Even though SMS does not read and create these other file types, it does tell ADCIRC that there is data there, where it is, and how to access it for a simulation to run correctly.

One of the parameters found in the fort.22 file (Single File Meteorological Forcing Input File) will tell which of the wind data file types SMS can read and create. The first parameter specified in the fort.22 file is called the Node Wind Stress (NWS) value. This value represents what type of wind data is in the file. It can range from 0 to 100+. SMS can read and create fort.22 files that are of the NWS value of 1, 2, or 5. All other values of the NWS value are only shown and directed to the ADCIRC model by SMS so that a simulation will run correctly.

Sediment Options Tab

This section of the SMS interface is still under development and is not fully functional. It will be updated sometime in the future.

Related Links

- ADCIRC
- Boundary Conditions
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Spatial Attributes
- Steering

ADCIRC Spatial Attributes

ADCIRC Nodal/Spatial Attributes

Spatial attributes (or nodal attributes) are parameters that can be applied to the entire domain of the simulation in the form of a dataset with values assigned to the nodes of the mesh. ADCIRC supports several such spatially varying input parameters and SMS (starting with version 11.0) includes functionality to assist defining, managing and editing these parameters. With each release of ADCIRC, the list of supported nodal attributes changes. There are several such attributes that are under development and are used by individuals in the ADCIRC community, but are not available in the release version. Therefore, the interface in SMS is a combination of specifically defined attributes to correspond with the standard parameters in ADCIRC and *custom* attributes to allow for customization and support of attributes not yet included in either the ADCIRC release or the standard list in the SMS interface.

Define these attributes in a the *Spatial Attributes* dialog under the *ADCIRC* menu. There are a number of predefined attributes that can be used or custom attributes can be defined. Here is a list of the predefined spatial attributes and a brief meaning of each.

In version 50 of ADCIRC, the following predefined nodal attributes are supported:

- Primitive weighting in continuity equation – Tau0
- Surface submergence state – StartDry
- Quadratic friction coefficient at sea floor – Fric
- Surface directional effective roughness length – z0Land
- Surface canopy coefficient – VCanopy
- Bridge pilings friction parameters – BK, BAlpha, BDelX, POAN
- Manning's n at sea floor – ManningsN

- Chezy friction coefficient at sea floor – ChezyFric
- Sea surface height above geoid – GeoidOffset
- Horizontal eddy viscosity – ESLM
- Wave_refraction_in_swan – SwanWaveRefrac (Supported in list of *predefined* attributes in SMS 11.2)
- Bottom_roughness_length – Z0b_var (Supported in list of *predefined* attributes in SMS 11.2)
- Average horizontal eddy viscosity in sea water wrt depth – ESLM
- Elemental_slope_limiter – elemental_slope_limiter_grad_max (Supported in list of *predefined* attributes in SMS 11.2)
- Advection_state – AdvectionState (Supported in list of *predefined* attributes in SMS 11.2)

Please visit the ADCIRC web site to learn more about the meaning of each of the spatial attributes listed above.

How to Use Spatial Nodal Attributes in a Simulation

Spatial nodal attributes are managed in the SMS using editable datasets. The creation and management of attributes is managed through the *Spatial Attributes* dialog. Access the *Spatial Attributes* dialog by selecting the *ADCIRC* | **Spatial Attributes** menu command.

Create a new spatial attribute by selecting the attribute name in the *New Attribute* combo-box. This will bring up the *Spatial Attributes Initial Values* dialog (discussed below).

The top left of the dialog has a list of the currently defined spatial attributes. The currently selected spatial attribute can be modified by the controls on the right side of the dialog or by editing specific dataset values by selecting the specific node(s) and entering a value. Each attribute has an associate comment. The comment is ignored by the model and is generally used for units information. The value below the comment is intended to represent the default value, or the value used the most in the dataset. Values at nodes that do not use the default value are written by the SMS to the fort.13 file as exceptions. The reset button will bring up the *Spatial Attributes Initial Values* dialog (discussed below).

Delete the current attribute by clicking on the **Delete** button. For each spatial attribute, SMS creates one or more datasets in the project explorer. However, these datasets cannot be deleted through the project explorer. They must be managed through the *Spatial Attributes* dialog.

Editing Spatial Attributes

To edit a spatial attribute, open the *Spatial Attributes* dialog. Select the spatial attribute that needs to be edited and select the **Reset** button.

Initial Values Dialog

The *Initial Values* dialog is used to setup the initial values for the dataset(s) used for the spatial attribute.

Every spatial attribute has the option of starting from a constant value or use the values from an existing dataset. In addition, some spatial attributes have an option to populate the spatial attribute based upon other data inside of SMS, using a rule, or using a utility similar to those provided on the ADCIRC websit to generate.

Populate Options

A spatial attribute consists of one or more datasets that can include unique values at each of the nodes in the mesh. This is potentially a lot of data and will generally not be specified manually. Two options exist for general population of the dataset. These include:

- Constant – Specifies a single value for all nodes. it is possible to later select one or more regions of nodes and assign a different value to these nodes using the scalar edit box at the top of the SMS interface.
- Dataset – Selects an existing dataset that is copied to the spatial attribute. Use any of the standard methods for defining the souce dataset including interpolation and the data calculator. Individual edits can still be made to nodes or regions of nodes by manually selecting them.

In addition to these two default methods of populating data sets, SMS supports some custom population approaches for specific attributes. These are described below.

Primitive weighting

The SMS includes a tool for populating the primitive weighting coefficient for each node based on the node spacing and the node depth. Specify a threshold for node spacing (Critical average node spacing) and depth (Critical depth). The defaults for these are 1750 m and 10 m respectively. Also specify a values for tau for three conditions including default (0.03), deep (0.005) and shallow (0.02).

If the average distance between a node and its neighbors is less than the critical value then tau0 for that node is set at the tau default. If the average distance between a node and its neighbors is greater than or equal to a critical spacing then tau0 for that node is assigned based on the depth using the deep or shallow value specified.

Quadratic, Manning N, and Chezy friction coefficients

The spatial attributes which reflect bottom friction "Chezy_friction_coefficient_at_sea_floor," "Mannings_n_at_sea_floor," and "quadratic_friction_coefficient_at_sea_floor" can be populated from NLCD land use data. Obtain NLCD geo-tiffs from the Multi-Resolution Land Characteristics Consortium website (www.mrlc.gov). Use the seamless server to download the data in the area of interest. To populate using this method, download the NLCD geo-tiff for the region of interest, load it into the SMS (creating an image), and convert it to a raster (right-click on the image). (Refer to the information of rasters for guidance).

When populating any of these spatial attributes, provide a roughness value for each land-use type and a default roughness value that will be used in areas of the raster that are NULL or outside the bounds of the raster.

The SMS will extract values from all visible raster objects (turned on in the project explorer). For each node in the mesh, the "area of influence" is computed for the node. The area of influence is a polygon that represents the area of the mesh around a node. This is computed as the area encompassing the node and half the distance to each of his neighboring nodes. All of the raster values within the area of influence are extracted from the visible raster objects. A composite roughness value is computed taking a weighted average of all the raster values. If the area of the extracted values is less then the area of influence around the node (because we passed the extent of the raster objects or encountered null values), the default value is used to compute the composite n using the area not represented in the values.

Surface directional effective roughness

The surface directional effective roughness length composes the contributing vegetation types associated with each of the 12 directions (30 degrees each) around each node in a mesh and assigns the 12 characteristic or composite reduction values to the datasets for the node.

The SMS includes an option to populate these values using NLCD land use data raster objects. These must be loaded before populating this attribute. Specify a *Total distance* to be included in the computation of the directional roughness as well as a *Weighted distance* and method of interpolation. Also specify a roughness value for each land use type.

This is in accordance with the utility provided on the ADCIRC web site for generation of this attribute. The SMS invokes this utility to compute an effective roughness for each cell for each direction.

Surface canopy coefficient

The surface canopy attribute provides a method for ADCIRC to reduce wind stress at specific locations in the mesh to represent the protection afforded by local vegetation. The SMS includes an option to populate these values based on NLCD land use raster objects (which must be loaded prior to generating this parameter).

Specify whether to compute the reduction based only on the nodal location, or based on the area of influence around a node (half the distance to adjacent nodes). If the area of influence method is used, the use can also specify a percentage cover that must exist for a reduction to be applied. Finally, specify a roughness reduction to be applied for each land use (vegetative cover type).

Custom Attributes

The name of a spatial/nodal attribute is key for its use in an ADCIRC simulation. ADCIRC reads the name from the fort.15 file and the corresponding values from the fort.13 file. If the name does not match a supported attribute in that version of ADCIRC that is being used, it is ignored. For this reason, the SMS interface includes the specific names supported by ADCIRC at the time of development. This minimizes the possibility of errors when entering the name of a nodal attribute.

There are two common situations which would require entering a *Custom* attribute. The first is to define an attribute for an ADCIRC run for an attribute that is not yet included in the SMS interface. The second is to define an attribute for a developmental feature in a custom version of ADCIRC.

In either case, select the *Custom...* option from the *New Attribute* combo box and the *Spatial Attribute – Custom* dialog appears. This dialog allows specifying a name for the attribute, which **must** match the expected string in the corresponding version of ADCIRC. Also specify a comment, and the dataset(s) name(s) associated with this attribute. By clicking **OK** , the *Initial Values* dialog appears which defines how to populate each dataset.

Related Links

- ADCIRC
- Boundary Conditions
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control

- Steering

LTEA

Linear Truncation Error Analysis (LTEA) Toolbox incorporated into SMS uses the LTEA algorithm as the heart of a utility which creates finite element meshes of varying resolution for ADCIRC analysis. The algorithm was initially presented by Dr. Scott C. Hagen as part of his doctoral research at Notre Dame, and development has continued on the methodology at the University of Central Florida. It performs analysis on an existing ADCIRC mesh and its solution to help quantify the error associated with the mesh. Normally, this ADCIRC solution is taken from a "linear ADCIRC" run. This type of run is used to make the process faster and to simplify the LTEA algorithm applied to the unstructured mesh. A second phase of the LTEA process uses the error values at each node to create a relative size function covering the domain called DelX. This refines the mesh where the element error values would be greatest such as near shorelines or around islands.

The LTEA Toolbox

SMS includes a graphical interface that allows using the LTEA theory to guide the generation of a finite element mesh. The tool requires two inputs which must be loaded into SMS. These include:

- A bathymetry scatter set.
- An ADCIRC coverage having at least one polygon with the boundary conditions assigned.

If wanting to start the mesh generation process from an existing ADCIRC mesh, convert the mesh to both a scatter set and ADCIRC conceptual model, Right-clicking on the mesh in the project explorer gives access to commands to perform the basic conversion. The conceptual model must be defined on the arcs created in this way.

The toolbox is accessed through the **Mesh Generation Toolbox...** from the *ADCIRC* menu in the Mesh Module. From the dialog that appears, select the *LTEA* option and click **Run** . The steps that follow include:

- Step 1 – Linear Run Mesh generation. In this step specify the scatter set and conceptual model to be used for mesh generation. This step generates a basic mesh from the conceptual model to perform a linear run. This mesh may be saved for future reference. If a mesh already exists that is suitable for the linear run, the option to generate a linear run mesh should be turned off.

- Step 2 – Linear ADCIRC Run. In this step, the toolbox runs ADCIRC in linear mode with the M2 tidal constituent and performs an harmonic analysis on the result. If a run of ADCIRC and harmonic analysis has already been performed, the option to use existing solution data becomes available. In this case specify the datasets that contain the output of the harmonic analysis. If this is not the initial pass through the mesh generation toolbox, or if wanting to generate a "Size guideline function" from anaother source, it may be provided. If this is the case, the linear run as well as the next step "LTEA calculations" are bypassed.
- Step 3 – LTEA Analysis. The LTEA algorithm processes the harmonic analysis output and determines the relative error due to node spacing throughout the domain. It then computes a size guideline for generating a new mesh. The guideline is a dataset (a value for each node of the linear run mesh) named "DelX". Instruct LTEA to only perform calcualtions on the interior (no partial molecule) or to approximate the LTEA calculations right up to the boundary. (Note: extra controls exist in the dialog for tools that are under development.)
- Step 4 – Generate Final Mesh. At this point, SMS has almost all the information required for mesh generation. Specify the target size of desired mesh as a number of nodes and a tolerable deviation from that target. The acceptable size transitions are also specified here (see the Smooth Dataset article for a description of this).

The tool can be used repetitively to generate various meshes of the same area with varying resolution. In these cases, the first three steps should be bypassed by entering the input data in step 1 and the "DelX" function in step 3 and then proceeding to step 4.

LTEA Tool Overview

A set of standard buttons can be found at the bottom of the

. They are as follows:

- **Help...** – Launches the *Help* dialogue
- **Continue >** – Moves to next step of the *LTEA Toolbox* dialog.
- **< Previous** – Allows returning to a an earlier step in the *LTEA Toolbox* dialog.
- **Stop and Run** – Causes SMS to close the toolbox and run the LTEA analysis. The analysis generates several datasets used as size functions in the mesh generation process.
- **Run** – Activates the model engine at the completion of all steps.
- **Cancel** – Closes the *LTEA Toolbox* dialog without saving and information entered.

Step 1:Linear Rush Mesh

- Boundary:
- Bathymeytry:
- Create Linear Run Mesh
    - Override Boundary Spacing
        - Coastline Spacing
        - Deep Water Spacing
- Use Current Mesh
- Save Linear Run Mesh

Step 2: IMS-ADCIRC Linear Run

- Run IMS-ADCRIC Linear Run
    - Wave Continuity/Friction Coefficient:
    - Minimum Angle for Tangential Flow:
    - Absolute Convergence Criteria:
    - Ramp Time

  - Run Time:
  - Time Step:
- Provide Harmonic Solutions
  - Eta Amptitude:
  - Eta Phase:
  - Velocity Ampitude:
  - Velocity Phase:
- Provide DelX
  - DelX:

Step 3: LTEA Analysis

- Run Type:
- Value for dc:
- Minimum spacing from:
- Node Number:
- Use Partial molecule
  - Molecule size:

Step 4:Generate Final Mesh

- Target number of nodes:
- Element area change limit:
- Redistribute boundaries
- Truncate element sizes
  - Minimum Size
  - Maximum Size

Case Studies / Sample Problems

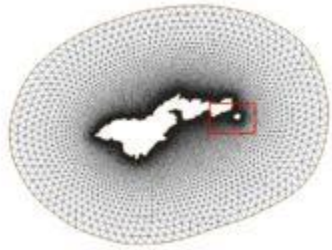
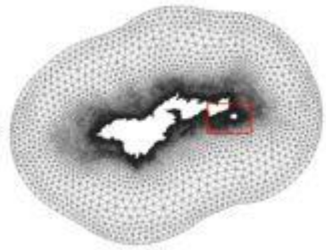
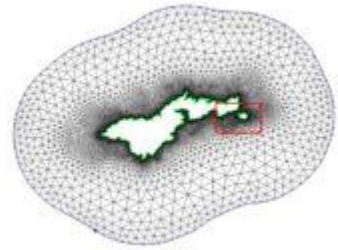
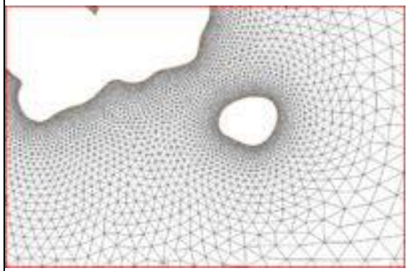
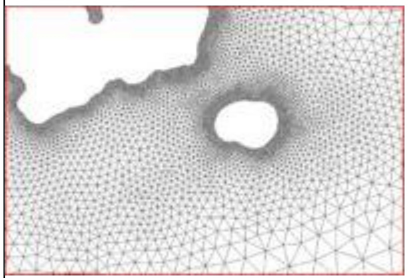
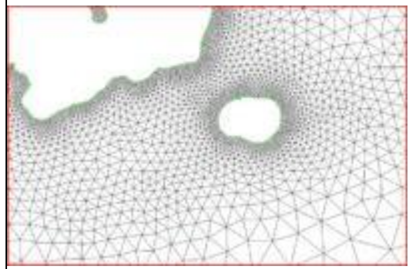
Tutorials

The following tutorials may be helpful for learning to use LTEA in SMS:

- Models Section
  - ADCIRC LTEA – Uses LTEA to mesh Shinnecock bay and the area around it along Long Island, NY

American Samoa

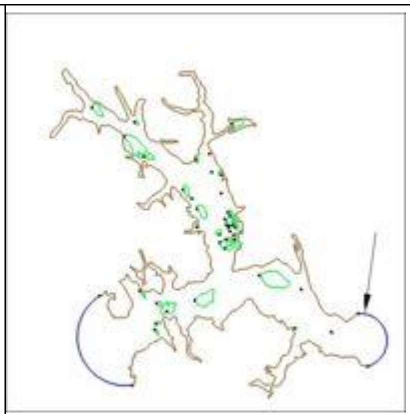
The following images illustrate the results of the LTEA toolbox applied to a domain around American Samoa. The first pair of images illustrate a mesh generated for the domain using the paving method. Density at the coastline was controlled by redistributing the vertices on the arcs representing the coastline and the density varied to a larger ocean boundary density. This mesh consists of 22,576 nodes (43,055 elements). The other images illustrate the varying resolution generated by LTEA to result in constant error with target mesh sizes of 24,000 nodes and 12,000 nodes respectively. The LTEA toolbox created meshes with 24,078 nodes (45,929 elements) and 12,029 nodes (22,543 elements).

These images illustrate the redistribution of density to increase the density in areas that require additional detail for solution variations, or to reduce the number of nodes in the mesh.

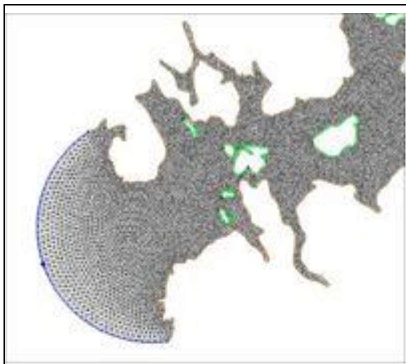
Glacier Bay Alaska

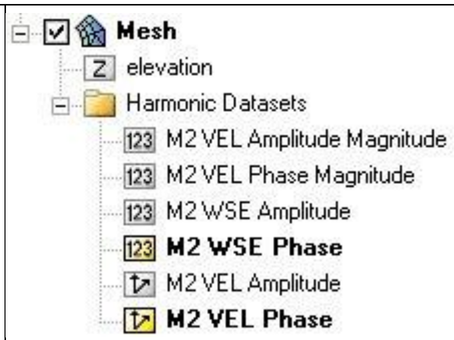
The case of Glacier Bay Alaska , by Dave F. Hill's research group, poses another problem for the LTEA toolbox. This case includes two ocean boundaries. The figures below show three meshes generated for this case and illustrate the large variation in node density that can be produced by the procedure.

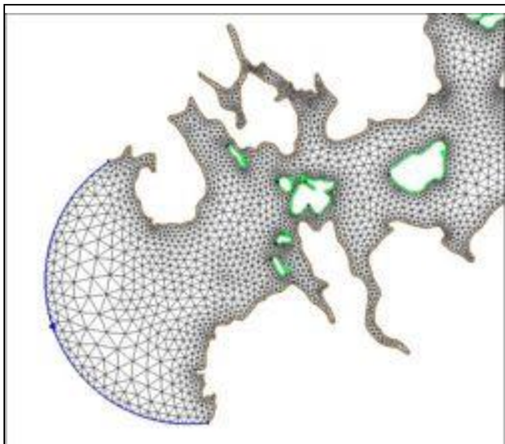
This case includes two ocean boundaries. Currently, the LTEA toolbox makes a sometimes erroneous assumption that only one ocean boundary exists. To work around this problem in the current version of SMS, the following steps are required:

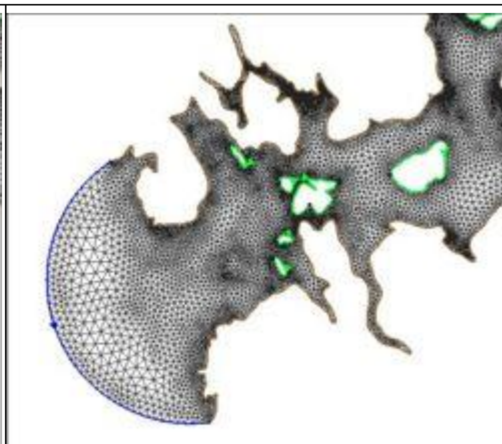
- Change the inland ocean boundary to land
- Run the first step of the LTEA toolbox to generate the "Linear Run Mesh" and then "Stop and Run" to that point.

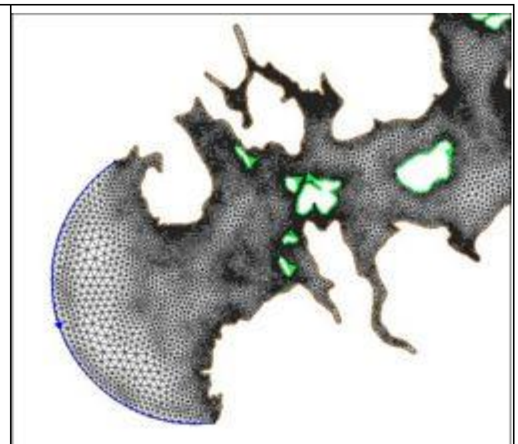




- Outside of the toolbox, change the second ocean boundary to ocean on the linear run mesh and run a linear run of ADCIRC with harmonic analysis turned on. This will generate the datasets for LTEA calcuations.
- Relaunch the toolbox and select the datasets from the linear run to guide the mesh generation process.







Related Topics

- ADCIRC
- Steering

External Links

- Coastal Hydroscience Analysis, Modeling & Predictive Simulations Laboratory (CHAMPS Lab)
- Sep 2006 Automatic, unstructured mesh generation for 2D, shelf-based tidal models
- Sep 2006 Automatic, unstructured mesh generation for tidal calculations in a large domain
- Sep 2006 Resolution Issues in Numerical Models of Oceanic and Coastal Circulation
- 2001 Two-dimensional, unstructured mesh generation for tidal models
- 2000 One-dimensional finite element grids based on a localized truncation error analysis
- Glacier Bay Test Case by Dave F. Hill

5.1.a. ADCIRC Boundary Conditions

ADCIRC Boundary Conditions

Much of ADCIRC's versatility as a model is due to the large number of different boundary types and boundary conditions available in the model. For detailed description of the ADCIRC boundary conditions, see the documentation provided by the ADCIRC development group at http://www.adcirc.org.

ADCIRC Boundary Conditions Nodestrings

Boundary conditions are assigned by setting attributes to *Feature Arcs* representing boundary arcs in the domain used in the sifmulation.

The *ADCIRC Arc/Nodestring Attributes* dialog defines the type of boundary conditions for each feature arc/nodestring.

A feature arc/nodestring can be defined as mainland, island, normal flow, mainland barrier, island barrier, tidal constituents, etc.

To set the boundary types, choose the **Select Feature Arc** tool from the Toolbox and double-click the desired arc to open the *ADCIRC Arc/Nodestring Attributes* dialog; then assign the desired boundary conditions to the arc.

ADCIRC Boundary Types

Boundary types are assigned to feature arcs in the conceptual mode or nodestrings on the ADCIRC mesh. Correct boundary type assignments are very important to run a successful ADCIRC project. The following boundary types are available in SMS for ADCIRC:

- *Unassigned* – Default, no boundary condition assigned
- *Mainland* – This type of boundary represents a mainland boundary with no normal flow condition and free tangential slip.
- *Island* – This type of boundary represents an island boundary with no normal flow condition and free tangential slip. This can be selected for a closed feature arc.
- *Normal Flow* – This type of boundary represents a river inflow or open ocean boundary with a specified normal flow condition and free tangential slip. Discharges are specified either for harmonic discharge forcing or for time series discharge forcing.
- *Normal Wave Radiation* – This type of boundary represents an open boundary where waves are allowed to propagate freely out of the domain.
- *Mainland Barrier* – This type of boundary represents a mainland boundary comprised of a dike or levee. Non-zero normal flow is computed using a supercritical, free surface weir formula if the barrier is overtopped. Zero normal flow is assumed if the barrier is not overtopped.
- *Island Barrier* – This type of boundary represents a dike or levee that lies inside the computational domain. Non-zero normal flow is compute using either subcritical or supercritical, free surface weir formula (based on the water level on both sides of the barrier) if the barrier is overtopped. Zero normal flow is assumed if the barrier is not overtopped. This boundary condition requires two nodestrings with an equal number of nodes. See ADCIRC Weirs and Island Barriers for more information.
- *Weir* – This type of boundary is similar to an Island Barrier with the addition of cross barrier flow. Cross barrier flow simulates flow through the barrier simulating pipes or culverts from one side to the other. Flow rate and direction are based on barrier height, surface water elevation on both sides of the barrier, barrier coefficient and the appropriate barrier flow formula. In addition cross barrier pipe flow rate and direction are based on pipe crown height, surface water elevation on both sides of the barrier, pipe friction coefficient, pipe diameter and the appropriate pipe flow formula. This boundary condition requires two nodestrings with the same number of nodes. See ADCIRC Weirs and Island Barriers for more information.

- *Zero Normal Velocity Gradient* – This type of boundary forces flow through the specified nodestring reflective of flow at a fictitious point inside the domain. This is referred to as a weakly reflective boundary in some numerical engines. The fictitious point lies on the inward directed normal to the boundary a distance equal to the distance from the boundary node to its farthest 'neighbor. This should ensure that the fictitious point does not fall into an element that contains the boundary node. The velocity at the fictitious point is determined by interpolation.
- *Ocean* – This type of boundary represents an open interface for flow with a specified water surface elevation. Elevations are specified either as tidal constituents for harmonic forcing or as time series or water level.

ADCIRC Boundary Conditions Options

Depending upon the specified boundary type, the following boundary conditions are available:

Boundary Condition	**Boundary Type**
Essential w/ Tangential Slip	Mainland Island Normal Flow Mainland Barrier
Essential w/o Tangential Slip	Mainland Island Normal Flow Mainland Barrier
Natural (w/ Tangential Slip)	Mainland Island Normal Flow Mainland Barrier
Tidal Constituents	Ocean
Curve	Ocean
Extract from Dataset	Ocean

Recording Stations

The SMS interface allows creating recording stations at specified nodal locations. At these locations, the ADCIRC model will output specified quantities at a user specified time interface. This allows for comparison of time series with observed buoy data. For example, the global output interval may be 30 minutes or 1 hour, while the recording station output could be at a higher frequency such as 6 minutes.

The following recording stations can be assigned to an ADCIRC mesh node:

2D Station Types

2D station options include:

- *Elevation* – ADCIRC will output a times series of computed water surface elevation at this location.
- *Velocity* – ADCIRC will output a times series of computed velocity magnitude at this location.
- *Concentration* – ADCIRC will output a times series of constituent concentration at this location.
- *Meteorological* – ADCIRC will output a times series of wind and pressure variables at this location.

3D Station Types

These options are only available when running in 3D mode.

- *Density/Temperature/Salinity*

- *Velocity*
- *Turbulence*

Description of Station

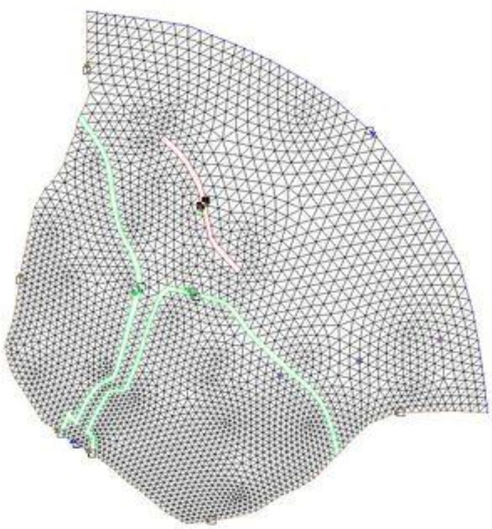
Stations can be assigned a name to make station identification easier. A single point can be a recording station for multiple types of data (i.e. a node can record both elevation and velocity).

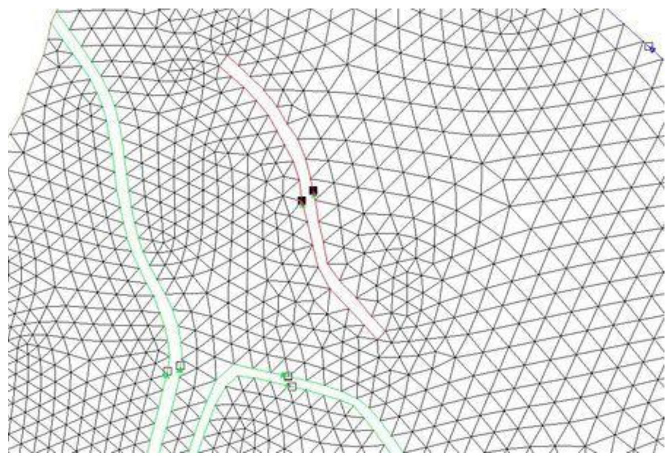
Related Links

- ADCIRC
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Spatial Attributes
- Steering

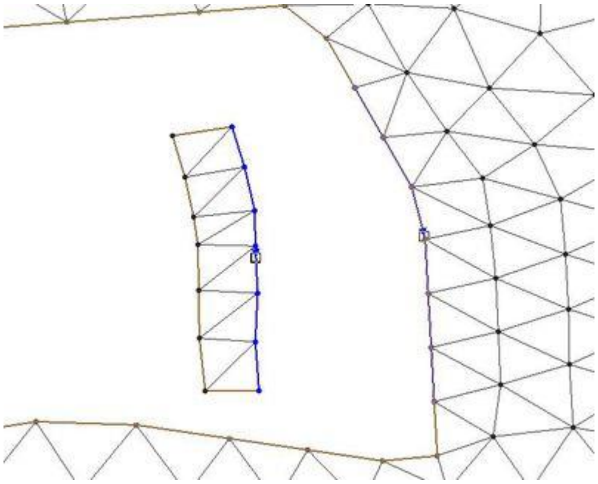
ADCIRC Weirs and Island Barriers

An ADCIRC weir is a boundary type that are assigned to two nodestrings on an ADCIRC mesh. A weir comprises of two nodestrings next to each other with an equal amount of nodes on each nodestrings. In order to add a weir in ADCIRC, there must be two adjacent nodestrings available as shown in figures 1 and 2 below.





The number of nodes on each nodestrings must be of an equal amount or a weir cannot be assigned. See figure 3 below. Each node and it's corresponding node on the parallel nodestring will form a node pair.



Weir Options

Once two nodestrings with same number of nodes exist, they can be assigned as a weir by selecting both weirs and selecting **Assign BC** . The *Nodestring attributes* will open. By selecting *Weir* and then clicking on the **Parameters** button, the *Weir Options* dialog will open. The dialog shows the different Node pairs and elevation for each node pair.

Elevation

Elevation of each node pair. The elevation for each node pair can be entered manually or can be interpolated from a dataset

Super

Weir flow coefficients for super critical flow. A typical value for this is 1.0.

Sub

Weir flow coefficients for sub critical flow. A typical value for this is 1.0.

- Pipe
- Elevation = elevation of the pipe
- Diameter = diameter of pipe
- Coefficient = $f (L / D)$

where

$$f \cdot \frac{L}{D}$$

– L is the length of the pipe

– D is the hydraulic diameter of the pipe (for a pipe of circular section, this equals to the internal diameter of the pipe)

– f is a dimensionless coefficient called the Darcy friction factor . It can be found from a Moody diagram or by solving the Colebrook equation.

Interpolate Elevations Button

Clicking on the **Interpolate Elevations** button opens the *XY Series Editor* . The *XY Series Editor* can be used to generate and edit curves defined by a list of x and y coordinates. The curve can be created and edited by directly editing the xy coordinates using a spreadsheet list of the coordinates. An entire list of curves can be generated and edited with the editor and curves can be imported from and exported to text files for future use. It is also possible to paste the xy data directly to the spreadsheet.

Extract Elevations

Data entered in as a weir or island barrier boundary condition for ADCIRC define the crest elevation between the nodes. This elevation data is independent from the elevation of the nodes on the grid (usually it is assigned from a separate source).

SMS includes tools to incorporate these elevations into a Cartesian grid that will be used in a different model and/or a coverage. These tools are accessed by selecting the **Extract Weir Elevations...** command in the *Nodestrings* menu. This command is not available unless there is an ADCIRC mesh loaded into SMS.

The dialog includes three separate sections.

- The first section includes a toggle box to specify which weirs/barriers are to be used in the operation. If the toggle is not selected, all weir/barrier boundary conditions in the simulation are used. If it is toggled, only the weirs/barriers that are selected (or partially selected) will be used. If a single nodestring from a weir/barrier is selected, it is treated the same as if both were selected.
- The second section maps the specified weirs to arcs. SMS will create one arc for each weir/barrier, bisecting the area between the nodestrings that make up the boundary condition. The elevation of the nodes/vertices created on the arc will match the specified crest elevations for the weir/barrier. This can also determine if these arcs should be added to a new or existing coverage.
- The third section modifies the elevation of the cells in one or more Cartesian grids, if they are loaded into SMS, to correspond to the weir/barrier elevation. Adding the elevations to the grid causes the land surface to actually represent the shape of the weir/barrier. Caution should be exercised since assigning the crest elevation to the grid can cause steep slopes between the weir/barrier cells and adjacent cells.

After selecting **OK** in the dialog, SMS will compute elevations at the midpoint of each node pair in the BC. This is done by first interpolating between the node elevations of each pair, and then subtracting the weir/barrier elevation (which were specified in the weir or island barrier parameters) from that value.

If the coverage option is used, SMS will create arcs that represent the weirs or barriers on the specified coverage. The computed elevations will be assigned along the arc. If one or more grids are used, SMS will set the elevation values for all grid cells that lie underneath the midpoints to match the computed values.

Remove Weir

Selecting two nodestrings that have been assigned a weir boundary condition allows using the **Remove Weir** command in the right-click menu. Using this command will bring up the *Remove Weir* dialog. This dialog gives the following options for removing the weir:

- *Pave over* – Removes the nodestrings that had been assigned the weir boundary condition.
- *Merge nodestrings* – Results in new nodes down the center of where the two nodestrings used to make the weir boundary conditions.
- *Runumber* – Generally required for model execution. This should be done later if not done with this step.

Add Weir

Selecting a single nodestring can be used to create a weir if it is known the desired width of the weir. Right-clicking a single boundary condition nodestring and selecting the **Add Weir** command will bring up the *Add Weir* dialog. After entering the weir *Width* and selecting **OK** , the weir is created.

Related Topics

- ADCIRC

5.2. BOUSS-2D – A Boussinesq Wave Model for Coastal Regions and Harbors

BOUSS-2D

BOUSS-2D	
Model Info	
Model type	Boussinesq Wave Model for Coastal Regions and Harbors.
Developer	Okey George Nawogu, Ph.D. Zeki Demirbilek, Ph.D.
Web site	BOUSS-2D web site
Tutorials	General Section

	• Data Visualization • Observation Models Section • BOUSS-2D Several sets of sample problems and case studies are available. These include: • Aquaveo sample problems • Model Validataion cases from the BOUSS-2D technical report

BOUSS-2D is a comprehensive model for simulating the propagation and tranformation of waves in coastal regions and harbors based on a time-domain solution of Boussinesq-type equations. It is based on Boussinesq-type equations derived by Okey Nwogu and has been under development since 1993. The equations are depth-integrated for the conservation of mass and momentum for nonlinear waves propagating in shallow and intermediate water depths.

The BOUSS-2D model can be added to a paid edition of SMS.

Functionality

BOUSS-2D computes nearshore wave fields including mean wave heights, mean current direction, mean water level breaking and transient representation of water levels, currents, and wave breaking.

BOUSS-2D is a comprehensive numerical model for simulating the propagation and transformation of waves in coastal regions and harbors based on a time-domain solution of Boussinesq-type equations. The governing equations are uniformly valid from deep to shallow water and can simulate most of the phenomena of interest in the nearshore zone and harbor basins including:

- Reflection/diffraction near structures
- Energy dissipation due to wave breaking and bottom friction
- Cross-spectral energy transfer due to nonlinear wave-wave interactions
- Breaking-induced longshore and rip currents
- Wave-current interaction
- Wave interaction with porous structures

The governing equations in BOUSS-2D are solved in the time domain with a finite-difference method. Input waves may be periodic (regular) or non-periodic (irregular), and both unidirectional or multi-directional sea states may be simulated. Waves propagating out of the computation domain are either absorbed in damping layers or allowed to leave the domain freely. The SI engineering units are used in BOUSS-2D calculations.

Output Options

See *Output Options* in the BOUSS-2D Simulations article.

Saving BOUSS-2D

When completing a *File* | **Save As...** command, the following files get saved in the *.sms

- *.mat referenced to new save location
- *.map referenced to new save location
- Damping files saved to temp folder
- *.par referenced to new save location
- *.sol referenced to original save location unless rerun
- *.h5 referenced to new save location

Using the Model / Practical Notes

BOUSS-2D can be applied to a wide variety of coastal and ocean engineering problems, including complex wave transformation over small coastal regions (1-5 km), wave agitation and harbor resonance studies, wave breaking over submerged obstacles, breaking-induced nearshore circulation patterns, wave-current interaction near tidal inlets, infra-gravity wave generation by groups of short waves, and wave transformation around artificial islands.

As with many numerical models, BOUSS-2D can terminate or crash due to numerical instabilities. These are usually caused by problems related to the grid, the boundary conditions, or model parameters. The following lists describe common causes of instability and methods to correct them.

Instability due to the grid/geometry

- Model stability requires a low Courant number throughout the domain. SMS computes an approximate maximum time step to maintain a Courant number below 0.5. In some cases, it is desired to lower the time step even more. Additionally, some may want to truncate the computational domain to areas with depth above a specified minimum. Another option is to increase resolution by using smaller computational cells. Either of these options increase run time, so before applying them, look at the other causes of instability.
- Abrupt changes in elevation from one cell to another in the computational domain could result in instabilities. It may be helpful to smooth the grid. (A smoothing command is available by right clicking on the grid object in the project explorer in the SMS interface.)
- Computation nodes surrounded on three or four sides by land may be created during the grid creation process. These "isolated" cells may become unstable and generally don't have an impact on the wave climate. They can be converted to land cells.

Instability due to the boundary conditions

- Generally, avoid placing damping or porosity layers along structures and shorelines.
- Wave makers are more stable on the edges of the domain. Therefore, generally speaking, the wave maker should be placed on the boundary of the domain in constant (or nearly constant) depth water. (The SMS interface offers to extend the grid and transition to constant depth if a wave maker is created in a location with more that 20% variation in depth.) This is especially true in real world applications where reflected waves are of no concern. Also, when simulating large waves, the greater stability of external wavemakers may be required.
- Wave makers should be placed far enough from shore to avoid interaction between the wave maker and reflecting waves. This is because the external boundary behind the wave maker is treated as a vertical wall.
- Exceptions, or applications in which internal wavemakers (i.e. wavemakers placed inside the domain) are recommended include:
    - In applications with significant reflections from structures inside the computational domain. When reflected are caused by coastlines, structures, or bathymetry (reflected wave sources), the simulated seastate will become less uniform spatially, and the simulation may not reach a steady-state condition. The resulting wave field in such simulations will generally consist of nodes and anti-nodes that resemble a standing wave pattern, where waves appear to be bouncing back and forth inside the domain. If reflected waves cannot escape through boundaries of the modeling domain (or are constrained to exit the domain), a steady-state condition technically cannot be reached irrespective of the length of simulation. When reflected waves intercept external wavemakers, the extremes (lows and highs) in the calculated wavefield may keep building and can eventually lead to model instabilities.
    - Internal wavemakers should be used for finite domains and especially for limited area physical modeling studies, and with the above specified guidance.
- If wavemakers are placed on the interior of the domain, they should cross the entire domain to avoid potential "end effects", and have a damping layer placed behind (on the seaward side of) the internal wavemakers to absorb reflected waves. There should also be a gap (at least one non-damped cell) between the internal wavemaker and the damping layer located offshore.

- In the absence of laboratory or field data to calibrate damping and porous layers for an application, consider multiple simulations with a range of damping widths and/or coefficients. This graph from BOUSS-2D's technical report illustrates the variation of effective reflectivity given various damping coefficients and damping layer widths. To use this graph:
  - Compute L (the wavelength for the incident wave).
  - Select a w/L ratio. Use this ratio to compute w (damping width).
  - Select an expected reflection percentage. Follow a horizontal line for this percentage on plot to intersect the graph for selected w/L ratio. Read associated damping coefficient from plot.
  - Note that the reflection coefficient is very sensitive to a change in damping coefficient when the coefficient is small (< 0.3) and much less sensitive when the coefficient is larger.
  - This process may require the damping parameters be changed when different wave conditions are considered.
  - It should be observed that this plot is for normally incident waves. Different reflection coefficients would be obtained for obliquely incident waves.
- Damping layers should be 5-10 cells wide.

Instability due to model parameters

- The model includes a Smagorinsky term to account for subgrid turbulence. If the turbulence is known, it is expected this term can be left at the default (0.0), however, it may be increased to increase stability. (This should be done with caution. Remember, don't suppress the wiggles, they are trying to say something.)

Test Cases

- **Test 1** – Simple test demonstrating the use of an internal wavemaker.

External Links

- CHL BOUSS-2D website [89]
- May 2007 ERDC/CHL CHETN-I-73 Infra-Gravity Wave Input Toolbox (IGWT): User’s Guide [90]
- May 2005 ERDC/CHL CHETN-I-70 BOUSS-2D Wave Model in SMS: 2. Tutorial with Examples [91]
- Mar 2005 ERDC/CHL CHETN-I-69 BOUSS-2D Wave Model in the SMS: 1. Graphical Interface [92]
- Sep 2001 ERDC/CHL TR-1-25 BOUSS-2D: A Boussinesq Wave Model for Coastal Regions and Harbors [93]

- Aug 2011 Tsunami Modeling Example Study - A Joint Hydraulic/Structural Methodology for the Rehabilitation of the Crescent City Marina [94]
- Apr 2014 Tsunami Modeling Article [95]

Related Topics

- BOUSS-2D Graphical Interface
- Cartesian Grid Module
- BOUSS-2D Files
- BOUSS-2D Model Control Dialog
- BOUSS-2D Calculators
- CGWAVE
- SMS Models page
- Spectral Energy

BOUSS-2D Using the Model

BOUSS-2D Grid

When using the model, the project will need a BOUSS-2D Cartesian grid. If a grid has already been created or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active.

To create a bouss-2d grid:

1) Right-click on the map data tree item and select new coverage.
2) Select **BOUSS-2D** .
3) Give the coverage a name then click **Ok** .
4) Click out a grid frame using the **Grid Frame** tool.
5) Follow the instructions found at Creating 2D Cartesian Grids .

Boundary Condition data

All numeric models require boundary condition data. In BOUSS-2D , boundary conditions are defined along arcs for porosity/damping, and on nodes for wave makers.

Damping / Porosity coverage

Damping and porosity coverages contain arcs that represent damping/porosity. The location of the arcs will be the location of the damping/porosity when running a simulation. Assigned to the arcs is the attributes of width and value. Multiple arcs can be created in a single coverage.

To create a damping/ porosity coverage:

1) Right-click on the map data tree item and select new coverage
2) Select **BOUSS-2D Damping or Porosity**
3) Give the coverage a name then click **Ok**
4) Select the newly created coverage
5) Click on the **Create Feature Arc** tool
6) Click out an arc over the grid where wanting the damping/porosity
7) Click on the **Select Feature Arc** tool
8) Click on an arc, then right-click and select **Attributes**

Wave Maker coverage

Wave maker coverages contain points that represent a wave maker. The location of the point will be the location of the wave maker when running a simulation. Assigned to the points are the wave maker settings including type and direction.

To create a wave maker coverage:

1) Right-click on the map data tree item and select new coverage
2) Select **BOUSS-2D wave maker**
3) Give the coverage a name then click **Ok**
4) Select the newly created coverage
5) Click on the **Create Feature Point** tool
6) Click out a point over the grid where wanting the wave maker
7) Click on the **Select Feature Point** tool
8) Click on an point, then right-click and select **Attributes**

Creating a Constant Depth Wave Maker Platform

The BOUSS-2D model generates waves along the specified wave makers (as described above). In order to increase model stability, and allow using an internal wave maker, it is necessary that the depths in the grid under a wave maker be relatively constant. For this reason, wave makers should be positioned far enough off shore that an entire row/column of the grid is at a relatively constant water depth. If the depth varies more than 10%, SMS will issue a warning when outputting the model.

To correct this situation, the depth under the wave maker can be adjusted to a constant value, creating a flat area. (This meets the requirements for stability for the BOUSS-2D model. We refer to this flat area as the *wave platform* .) The steps to create a flat area include:

1) select the cells of the area to be made flat (directly under and adjacent to the wave maker). Use the select cell, select row or select column tools to do this.
2) specify the elevation (remember that BOUSS-2D uses elevations, so depths are negative numbers) in the *Z* edit field to change the elevation of the selected area.

Example

- Assume the grid origin (I=1, J=1)is offshore, and wanting to place the wave maker at the I=50 column.
- Assume the average elevation of all cells between I=1 and I=50 is -30 m.

Steps

1) Select all the cells in columns 1 through 50
2) Specify an elevation of -30 m in the *Z* edit field

Notes

- SMS computes the average depth of all selected cells. This is displayed in the *Z* edit field when multiple cells are selected.
- The average depth could be chosen by selecting the row or column where the wave maker will be placed. However,
- Selecting a few rows/columns using the shift key to add to the selection in the region of the wave maker displays the average elevation for a larger region. This average may blend into the natural bathymetry with less the irregularity.
- Individual rows/columns can be selected with the SMS grid tools. Larger regions can be selected using the select cell tool and holding down the control key and dragging a box. The *alt* key can be used to just add new cells to the selection list.
- The constant elevation area should extend for several columns/rows behind and in front of the wave maker. The rule of thumb is to select all the rows/columns within half-wavelength from the wave maker.

Creating a Transition Zone from Wave Maker Platform to Natural Bathymetry

If a constant elevation area has been created as described in the previous section, rapid changes in elevations past the wave maker can cause model instabilities. To avoid this, it is recommended to create a smoothed transition zone shoreward of the wave maker by smoothing the bathymetry. Otherwise, the BOUSS-2D model may still encounter instabilities do to rapid changes in bathymetry. To avoid this potential problems, use SMS to smooth the bathymetry to transition smoothly from the wave maker into the natural bathymetry. (Note: this operation should only be applied in area close to the wave maker, and no smoothing should be made to the principal study area. The effect of bathymetry smoothing on model results depends on the magnitude and extent of smoothing applied.

To create a smooth transition, use the following steps:

- Right-click on the grid and convert the grid to a scatter set.
- Switch to the scatter module and make sure the newly created scatter set is active.
- Issue the *Data* | **Dataset Toolbox...** command to bring up the toolbox.
- Select the **Smooth datasets** command.
- Select the dataset which represents the bathymetry values.
- Select the option to limit the *Maximum slope* and enter the transition value. (i.e. 0.1 implies a maximum slope of 1 on 10)
- Select the option to anchoe the *Minimum value* . This assumes that the wave platform is the minimum value and will ensure that it stays flat as defined in the section above.
- Enter a new dataset name such as *ramped_bathy* to store the smoothed bathymetry dataset.
- Click the **Compute** button to generate the new dataset and then the **Done** button to exit the toolbox.
- Switch back to the grid module and select the **Select Cell** tool.
- Drag a box (control key) around the wave platform and the adjacent area where the grid transition is to be applied. This will select the cells that will represent this transition.
- Right-click on any of the selected cells and select the **Interpolate Bathymetry...** command.
- Select the *ramp_bathy* function (using the name used in the step above) to select the new bathymetry source for the selected cells and click the **OK** button.

Note: The above steps replace the natural bathymetry in the grid only in the transition zone. If wanting to save this, duplicate the dataset before applying this smoothing operation. Also note that the bathymetry smoothing of the dataset is applied to the entire region. The coastline and any structures along the coastline would also be smoothed. That is why the application of the new bathymetry is limited to the selected cells in the vicinity of wave maker in the final steps of this procedure.

Using Roughness Coefficients

Roughness coefficients can be defined in a roughness coverage then converted to the active grid.

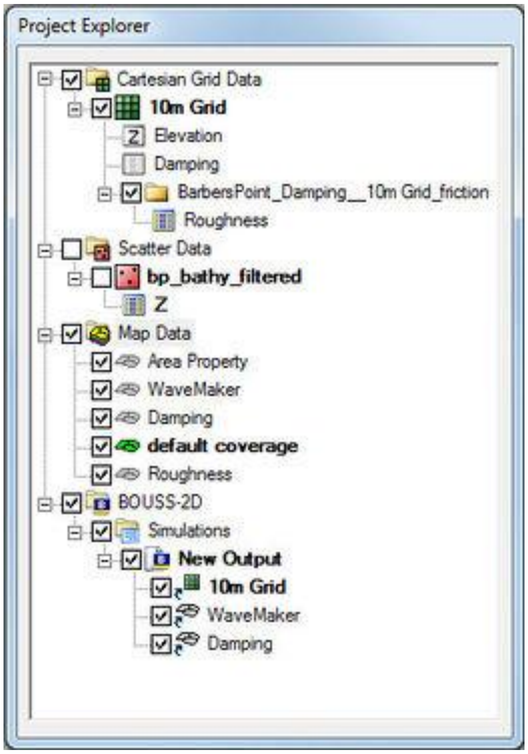
The BOUSS-2D roughness coverage allows building polygons that can be assigned a roughness coefficient. Right-click on a feature polygon and selecting the **Attributes** command with bring up a *Roughness* dialog.

After setting the roughness coefficient for the polygons, the roughness coverage can be converted to the active grid. The grid that is currently selected in the Project Explorer will receive the roughness dataset. Right-clicking on the roughness coverage in the Project Explorer and selecting the *Convert* | **Map → Active Grid** will bring up the *Map → Active Grid* dialog. The *Map → Active Grid* dialog allows setting a default roughness coefficient value and assigning a dataset name for the roughness coefficient dataset being assigned to the active grid.

Related Topics

- BOUSS-2D

BOUSS-2D Simulations



Simulations are available in BOUSS-2D starting in SMS 11.2. A simulation should contain a BOUSS-2D grid, a wave maker coverage, and optional damping and porosity coverages. Right-clicking on the simulation will display the options and dialogs.

To create a new simulation:

1) Right-click on empty space in the project explorer and select *New Simulation* | **BOUSS-2D**
2) Drag and drop bouss2d grid, wave maker/damping/porosity coverages under the newly created **Simulation** tree item.

A new tree item will appear.

Simulation Components

A BOUSS-2D simulation uses the following components in the model run:

- Cartesian Grid (created from the BOUSS-2D coverage)
- Damping Coverage
- Porosity Coverage
- Wave Maker Coverage

Each component can be added to the simulation by selecting the component and dragging it under the simulation item in the Project Explorer.

Menu Commands

Right-clicking on the simulation tree item will display a list of menu commands and dialogs that are available. These items are:

- **Generating Arcs along land boundary** – Opens a dialog that will create arcs in a damping or porosity coverage from an existing grid or scatterset.
- **Generating Arcs along open boundary –** Opens a dialog that will create arcs in a damping or porosity coverage from an existing grid or scatterset.
- **Calculators**
- **Probe Manager**
- **Model Control**
- **Model Check** – The model check will warn of potential errors that should be considered for fixing before running the model.
- **Export BOUSS-2D** – When the BOUSS-2D files are exported a new directory under the project called “BOUSS-2D” is created. In this directory, the grids will be written to. Also a new directory under the “BOUSS-2D” will be created and it will be the name of the simulation. In this directory, the *.par file(link to Parameter file) can be found.
- **Run BOUSS-2D**
- **Save Project, Export, and Run BOUSS-2D**

The simulation right-click menu also has general simulation commands .

See BOUSS-2D Graphical Interface for more information.

Output Options

BOUSS-2D can be instructed to create a variety of output files. These may include spatially varied data consisting of a value for each cell in the grid, transient data defining time series at a location, or a combination of these two options (multiple time steps of data that includes a value at each cell). The output options, along with the keyword included in the parameter file to enable these options are shown below.

- Steady-state/single value spatially varied datasets
  - Significant wave height (":HS_FILE")
  - Mean currents (":MEAN_UV_FILE")
  - Mean wave direction (":THETA_FILE")
- Transient spatially varied datasets. Each output includes data from a specified start time, to a specified end time at a specified time step.
  - Water surface elevations (":SAVE_ETA_ANIMATION")
  - Transient currents (":SAVE_UV_ANIMATION")
- Time series output at specified cells (probes). BOUSS-2D saves each type of data (for multiple locations) in a single "*.ts1" file.
  - Water surface elevations (":TS_ETA_FILE")
  - Currents (":TS_U_FILE", ":TS_V_FILE") – This saves the current at a specified elevation up from the bed. Multiple elevations can be monitored.
  - Pressure (":TS_PRESSURE_FILE") – This saves the pressure at a specified elevation up from the bed. Multiple elevations can be monitored.
  - Flow rate (":TS_Q_FILE" – This saves the flow crossing a location and can be used for overtopping.

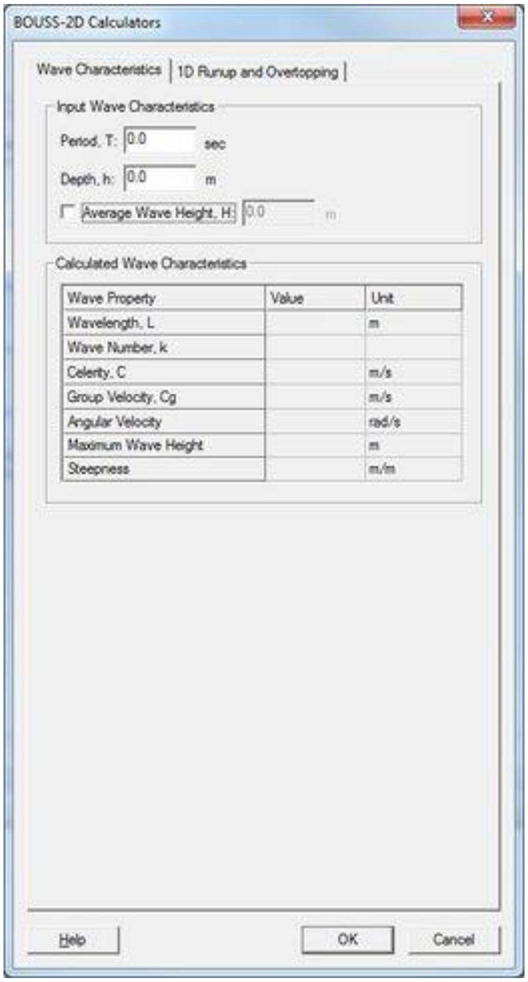
The spatially varied data may be output in to either BOUSS-2D native files or a eXtensible Data Format File (XMDF). If BOUSS-2D format is specified, the model creates "*.grd" files for each of the single value spatially varied outputs and binary data files for the transient data. The ":SOLUTION_FILE_OPTION" in the par file instructs the model to save the data in BOUSS-2D format (if set to 0), XMDF format (if set to 1) or both formats (if set to 2). When the XMDF option is specified, the ":XMDF_SOLUTION_FILE" record must also be in the parameters file along with the name of the XMDF file to store the datasets in.

The output options are located in the *Model Control* .

Related Topics

- BOUSS-2D

BOUSS-2D Calculators



Wave Characteristics Calculator

This calculator allows setting the following values:

- Input Wave Characteristics
  - Period, T
  - Depth, h
  - Average Wave Height, H
- Calculated Wave Characteristics
  - Wavelength, L
  - Wave Number, k
  - Celerity, C

- Group Velocity, Cg
- Angular Velocity
- Maximum Wave Height
- Steepness

Run-up and Overtopping Calculator

To assist in the design of coastal structures, the interface includes a one-dimensional wave run-up and overtopping calculator. This utility runs a 1D simulation with BOUSS-1D based on user specified parameters. To access the *1D Run-up and Overtopping* calculator select the **Calculators** menu item in the *BOUSS2D* menu to bring up the *BOUSS-2D Calculators* dialog. The *1D Run-up and Overtopping* calculator is a tab in the *BOUSS-2D Calculators* dialog. The *BOUSS-2D Calculators* dialog is always in the BOUSS-2D interface.

Input to the *1D Run-up and Overtopping* calculator is organized into a spreadsheet. The first row of the input parameters spreadsheet is fixed and will contain the column titles given in the table below. The remaining rows contain the input parameters as shown in the dialog. The names and units columns are read-only. The value column is editable.

Parameter	Value	Units
Wave Type	Choose between “Regular” and “Irregular”. Titles of the wave height & period change depending on wave type. Default is “Regular”	-
Wave Height(H)(regular) or Significant Wave Height (Hs) (irregular)	Must specify	(m)
Wave Period(T) (regular) or Peak Period (Tp) (irregular	Must specify	(sec)
Depth at Toe of Breakwater (ds)	Must specify	(m)
Crest Elevation above Still Water (Zc)	Must specify	(m)
Side Slope (m)	Must specify	-
Offshore Slope (p)	Must specify	-
Chezy Roughness Coefficient	Must specify	m1/2/sec

Output of the calculator is displayed on the bottom portion of the dialog after the calculate button is clicked. The output is organized into a spreadsheet. When the dialog first comes up, the values in the output parameters spreadsheet are blank. The output parameters calculated are:

1) Run-up, R. This value corresponds to Rmax for regular waves and R2% for irregular waves. The units of run-up are in meters
2) Overtopping, Q. A single overtopping value will be computed. The units for overtopping are (m3/s)/m.

External Links

- Example of the BOUSS-1D Run-Up and Overtopping Calculator May 2005 PDF (pg.14-16) "BOUSS-2D Wave Model in SMS: Tutorial with Examples"
- Evaluation of BOUSS-1D Jan 2009 PDF (pg.10-55) "Wave Transformation Over Reefs: Evaluation of One-Dimensional Numerical Models"

Related Topics

- BOUSS-2D Graphical Interface

BOUSS-2D Files

BOUSS-2D makes use of input files during the model run and has options for outputting files.

Input Files

BOUSS-2D makes use of the following files during a model run.

- Required files
    - Parameter File (*.par)
    - Bathymetry File (*_bathy.grd)
- Optional files
    - Damping File (*_damping.grd)
    - Current File (*_current.grd)
    - Porosity File (*_porosity.grd) (Still under development – be advised to use with caution)
    - Variable Roughness File (*_friction.grd) (Still under development – be advised to use with caution)

Output Files

BOUSS-2D can output both spatial and locational files. Output file options are specified in the *BOUSS-2D Model Control* dialog.

- Spatial files – Values at every active computation point, These can be saved in 5 separate native BOUSS-2D formatted files (three ASCII and two binary) or in a single binary XMDF file (*.h5 – This is an HDF5 file). The native format files include:
    - Transient datasets
        - Water Surface File (*.eta) (This also includes breaking index at each cell.)
        - Current File (*.uv)
    - Mean values
        - Significant Wave Height File (*_hs.grd)
        - Mean Current File (*_mean_uv.grd)
        - Mean Water Level File (*_mwl.grd)
- Locational files – These files contain information for specifaclly selected cells (probes)
    - Water Level at Probes File (*_ts_eta.ts1)
    - U Component of current at Probes File (*_ts_u.ts1)
    - V Component of current at Probes File (*_ts_v.ts1)
    - Pressure at Probes File (*_ts_p.ts1)

Related Topics

- BOUSS-2D
- BOUSS-2D Parameter File

BOUSS-2D Graphical Interface

The BOUSS-2D graphical interface includes tools to assit with creating, editing, and debugging a BOUSS-2D model. The BOUSS-2D interface exists in the Cartesian Grid Module .

BOUSS-2D Menu

The following menu commands are available in the BOUSS-2D simulation right-click menu:

Generating Arcs along land boundary

Opens a dialog that will create arcs in a damping or porosity coverage from an existing grid or scatterset.

Generating Arcs along open boundary

Opens a dialog that will create arcs in a damping or porosity coverage from an existing grid or scatterset.

Calculators

Brings up a pop up menu to access the *Wave Conditions Calculator* (see appendix A) as well as the Run-up/Overtopping Estimator.

Probe Manager

Brings up the *Probe Manager* to control time series output from the model.

Model Control…

Brings up the *Model Control* dialog to specificy model parameters.

Model Check …

Launches the Model Check to search for common problems.

Export BOUSS-2D

When the BOUSS-2D files are exported a new directory under the project called “BOUSS-2D” is created. In this directory, the grids will be written to. Also a new directory under the “BOUSS-2D” will be created and it will be the name of the simulation. In this directory, the *.par file(link to Parameter file) can be found.

Run BOUSS-2D

Brings up a dialog that allows checking what executable of BOUSS-2D should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

Save Project, Export, and Run BOUSS-2D

Performs the processes of saving the project, exporting BOUSS-2D files, and launching the BOUSS-2D model run.

Obsolete Commands

The following commands are no long in use in current versions of SMS but may appear in older versions.

Spectral Energy

Brings up the *Spectral Energy* dialog to define/view wave energy spectra. Generally, BOUSS-2D will generate wave conditions internally, but a spectrum may be input. This command also allows visualizing wave spectra that are generated inside of the model.

Assign BC

Used to assign damping, porosity, or wave maker conditions along a selected cell string(s) . Using this command will open the *BOUSS-2D Boundary Conditions* dialog.

Assign Cell Attributes

Selected cells can be defined as land or water

Polygon Attributes

The *Polygon Attributes* dialog in BOUSS-2D is used to set the attributes for feature polygons before converting to a grid. Attributes that can be specified for each polygon include:

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

Model Control

The *BOUSS-2D Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In BOUSS-2D , boundary conditions are defined on cell strings. The default boundary condition is a closed boundary (no flow).

BC Cell Strings

When the BOUSS-2D grid is created, SMS creates cell strings around the computational boundaries of the domain. A cell string is a list of contiguous node locations in the grid. The image to the right shows a grid with four cell strings that were automatically created. The cell string on the left includes the cells along the open ocean. This one has been assigned to be a wave-maker. The one on the right defines the interface between ocean and land along the coastline, and the top and bottom define the portion of the grid that are open to the ocean on those sides.

Cell strings can also be created manually to specify the location of structures, wave-makers, and areas where damping and/or porosity layers may be necessary.

Boundary conditions are specified along cell strings in the BOUSS-2D *Boundary Conditions* dialog, which is accessed by selecting one or more cell strings using the select cell string tool, and then selecting the **Assign BC** menu item from the *BOUSS-2D* menu. Normally, select a single cell string and assign a boundary condition. If a boundary condition has already exists for the selected cell string, the attributes are displayed. The different options for a cell string include:

1) *Unassigned BC* – When a cell string is created in BOUSS-2D its default boundary condition type is Unassigned. Unassigned cell strings do not influence the model. In fact, unassigned cell strings are not saved as part of the BOUSS-2D input files.
2) *Damping BC* – Waves propagating out of the computational domain are absorbed in damping regions (or damping layers) placed around the perimeter of the computational domain. Damping layers can also be used to model the partial reflection from harbor structures inside the computational area. Enter a physical width into the *Width* edit field to specify the size of the damping layer. The damping region extends the width on either side of the cell string. The damping value is a non-dimensional damping coefficient that is allowed to vary from 0.0 to 1.0. No damping will occur when a value of 0.0 is used. Waves will be damped when a value of 1.0 is used along the side boundaries. A typical value for shoreline is 0.1. The default damping value is 1.0. SMS will assign the value specified at the cell string and ramp down to 0.0 at a distance of “width” from the cell string.

3) *Porosity BC* – Porosity boundary conditions are used to simulate partial wave reflection and transmission through surface-piercing porous structures such as breakwaters. Enter a physical width into the "Width" edit field to specify the size of the porous structure. Like with the damping regions, this width is extended on both sides of the cell string. The porosity value is a non-dimensional porosity coefficient that is allowed to vary from 0.0 to 1.0. A value of 0.0 corresponds to an impervious structure, while a value of near 1.0 would correspond to a highly porous structure. Typical porosity for stone type breakwaters is 0.4. The default porosity value is 1.0.

4) *Wave-maker BC* –

The wave-maker option is only available when a single cell string is selected and that cell string lies in a single column or row (straight line). Legal cell strings can be created using the *SHIFT* key when creating cell strings, using automatically created cell strings along a grid boundary, or by creating short cell strings. The extent and position of the wave maker can be modified using I,J indices in the dialog. BOUSS-2D generates waves emanating from this cell string. The properties of the waves are defined using the *Wave Generator Properties* dialog (described below) that is accessed through the Options button. The edit fields are used to position and size the wave maker in the computational domain. The first two values are the Start and End cells of the wave maker along the column or row that is specified by the third value, which is the Offset value. The Start and End values are limited to the number of cells in either the I- or J-direction, and the Offset value is limited to the number of rows or columns.

When the **OK** button is clicked, a check is done to see if the wave maker cell string is at a constant depth. If the depth varies by more than 20% and the wave maker is on the edge of the grid (not internal),SMS will ask whether to force constant depth along the wave maker cell string or not. If so, the grid is extended to allow the wave maker to be at the deepest elevation along the string, with a maximum slope of 1:10 from the existing grid to the new wave-maker position. A *Wave Calculator* is provided as part of BOUSS-2D interface in SMS (see Appendix B) to assist in the preparation of wave input parameters required by the model. Note that the *BOUSS-2D* | **Assign BC** menu item is disabled any time multiple wave makers are selected or if a wave maker and one or more other cell strings are selected.

Running the Model

The BOUSS-2D files are written automatically with the SMS project file or can be saved separately using the *File* | **Save BOUSS-2D** or *File* | **Save As** menu commands. See BOUSS-2D Files for more information on the files used for the BOUSS-2D run.

BOUSS-2D can be launched from SMS using the **Run BOUSS-2D** right-click menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the **Model Check** right-click menu command.

Related Topics

- BOUSS-2D
- Create a 2D Cartesian grid

BOUSS-2D Model Control

The **Model Control…** command in the BOUSS-2D simulation right-click menu opens the *Model Control* dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

- *Input Datasets* – These inputs specify spatially varied input parameters for the model. In addition to the bathymetry, BOUSS-2D can utilize spatially varied damping, porosity and currents. In this section define whether these inputs are used ("None" is selected if not), and if so, how they are defined. The Damping and Porosity values can be specified using boundary conditions on cell strings or selected from the scalar functions. The Current is selected from the available vector functions.
- *Time Control* – The interface displays the recommended values for Duration and Time step in parentheses. The Courant number (Nwogu and Demirbilek 2001, page 15, 35) is computed based on the cell size and the specified time step. A value 0.3 to 0.5 is recommended.

- *Laminar and Turbulent Viscous Flow Coefficients for Porosity* – The Turbulent value ranges from 0.0 to 3.6 with a default value of 2.4. The Laminar value ranges from 0 to 1500 with a default value of 800. Stone size represents the characteristic stone size (d50) in meters of the breakwater armor layer and must be greater than zero.
- *Grid Information* – This section displays the attributes of the active grid and cannot be edited from within the Model Control dialog.
- *Parameters* – The *Tidal Offset* value is the elevation of the water level relative to the still water level. Check the *Enable Wave Runup* box to enable wave runup calculations. A minimum flooding depth must be specified for runup calculations with a default value of one-hundredth of the wave height. The Chezy coefficient ranges from 0.0 to 1000.0 (default = 50) and the Smagorinsky coefficient must be greater between 0.0 and 2.0 (default=0.2). The choice for *Nonlinear Option* is either "Strong" or "Weak". Check the *Enable Wave Breaking* box to enable simulation of wave breaking and enter a value for the *Turbulent Length Scale* with a default value equal to the wave height.
- *Variable Roughness* –
  - In the model control options, tselect whether to use a constant Chezy coefficient or variable roughness. With the option variable roughness, an editable dataset of the roughness for the grid can be created from the model control dialog.
  - A roughness coverage may also be used to create variable roughness in a BOUSS2D Cartesian grid. The roughness coverage is created in the same way as in BOUSS2D Runup/Overtopping. In the coverage, polygons can be created and given a Chezy coefficient. With an active BOUSS2D Cartesian grid created, right-clicking on the roughness coverage will allow the option "Map → Active Grid". This will pop-up a dialog, which will ask for a dataset name and a default Chezy roughness value to use where no polygon overlaps the given grid cell. Clicking **OK** will create an editable roughness dataset for the grid and set the model control option to variable roughness.
- *Spatial Output Options* – In this section of the dialog, define what output files BOUSS-2D should create. The *Override default file prefix* box specifies a file name for these solution files. The default file prefix is the filename of the *.par file.
  - By checking *Output WSE* and/or *Output Velocity* the model will output time varying datasets. By default, the model will save data at each time step. This would result in huge files, so normally select the *Override Default* toggle. This enables the *Begin Output* , *End Output* , and *Step* controls. The *Begin Output* & *End Output* values must be greater than 0.0 and less than the run duration. Also, the begin time must be less than the end time. The *Step* control determines how often the model will save time-varying output. SMS computes the number of frames and an approximate amount of memory required for the resulting datasets. If snap shots throughout the simulation are desired, that time range can be the entire simulation with a large interval to prevent the number of time steps from becoming excessive. These time steps are then not close enough together to create a coherent animation. Another option is to view only a couple of wave cycles, and make the time interval 5–10% of the wave period to get enough frames to generate a smooth animation of the waves in that time range. As mentioned, the time varied quantities include:
    - Water Surface Elevation and breaking
    - Velocity
  - BOUSS-2D can output spatial data (1 value for each cell in the grid). These quantities are read by the SMS as stored as datasets on the grid. The options include up to four steady state quantities and up to two transient (time varying) quantities. The steady state options include:
    - Significant Wave Height
    - Mean Currents
    - Average Wave Direction (Theta)

Related Topics

- BOUSS-2D
- Runup/Overtopping Model Control

BOUSS-2D Parameter File

The BOUSS-2D parameter file contains a series of comments and keywords that define the components and control the options of a simulation. Any line that begins withs "#" symbol is a comment and is ignored by the model. However, some utilities exist that may utilize comments.

Keywords begin with a ":" and are followed by values associated with the keyword.

The parameter file format is shown in Figure 1.

# ## # BOUSS-2D Run Parameter File: cirp_ideal.par # Written by: SMS # Creation Date: Tuesday June 22 16:49 2004 ##	File Header
# # Bathymetric Grid Parameters # :BATHY_FILE *filename* .grd :TIDAL_OFFSET offset value	Definition of grid and tidal offset
# # Damping Parameters :DAMPING_FILE *filename* .grd	Keywords for damping or porosity spatial input data */
# # Wavemaker #1 parameters # :START_WAVEMAKER : WM_POS_I1 value (column number) : WM_POS_J1 value (row number) : WM_POS_I2 value (column number) : WM_POS_J2 value (row number) : WAVE_TYPE value (Regular/Irreg_Uni/Irreg_Multi) : WAVE_HEIGHT value (meters) : WAVE_PERIOD value (seconds) : WAVE_DIRECTION value (degrees from North) : WAVE_CYCLES value (integer) :END_WAVEMAKER	The Wave maker definition. The position values must correspond to a single row or column. The values required depend on the wave type and method of defining that wave. The values shown illustrate a regular wave.
# # Simulation parameters # :DURATION value (seconds) :TIME_STEP value (seconds) :CHEZY_COEFF value (unitless default = 50) :SMAGORINSKY_CONST value (unitless default = 0.2) :NONLINEAR_OPTION value (Strong/Weak) :CHECK_WAVE_BREAKING value (Yes/No default - Yes) :TURB_LENGTH_SCALE value (meters0	General model parameters

:CALC_WAVE_RUNUP value (Yes/No default - No)	
# # Output Parameters # :OUTPUT_FILE_PREFIX *file prefix* # # Output File for Significant Wave Height # :HS_FILE *file prefix* _hs.grd # # Output File for Mean Currents # :MEAN_UV_FILE *file prefix* _mean_uv.grd # # Output File for Mean Wave Direction # :THETA_FILE *file prefix* _theta.grd	Single Timestep (Steady State) spatially varied output options
# # Surface Elevation Animation File Output Options # :SAVE_ETA_ANIMATION : ETA_ANIM_FILE *file prefix* .eta : START_TIME value (seconds) : END_TIME value (seconds) : SAVE_TIME_STEP value (seconds) : SAVE_FULL_GRID value (Yes/No default = Yes) :END_SAVE_ETA_ANIMATION # # Velocity Animation File Output Options # :SAVE_UV_ANIMATION : UV_ANIM_FILE *file prefix* .uv : START_TIME value (seconds) : END_TIME value (seconds) : SAVE_TIME_STEP value (seconds) : SAVE_FULL_GRID value (Yes/No default = Yes) :END_SAVE_UV_ANIMATION	Multiple Timestep (transient animations) spatially varied output options
# # Time Series Output File Names # :TS_ETA_FILE *file prefix* _ts_eta.ts1 :TS_U_FILE *file prefix* _ts_u.ts1 :TS_V_FILE *file prefix* _ts_v.ts1 :TS_PRESSURE_FILE *file prefix* _ts_pressure.ts1 :TS_Q_FILE *file prefix* _ts_q.ts1 :TS_RUNUP_FILE *file prefix* _ts_runup.ts1	Header for time series output options - must be included to create TS1 files
# # Time Series Output Options # :SAVE_TIMESERIES value (id) : SAVE_TS_ETA value (Yes/No) : SAVE_TS_UV value (Yes/No) : SAVE_TS_PRESSURE value (Yes/No)	Time Series specification block. Repeated as many times as needed.

: SAVE_TS_Q value (Yes/No) : TS_X value (meters) : TS_Y value (meters) : TS_Z_UV value (meters above bed) :END_SAVE_TIMESERIES #	
# # Time Series Runup Output Options # :SAVE_RUNUP value (id) : TS_X value (meters) :END_SAVE_RUNUP #	Runup Series specification block. Repeated as many times as needed. Currently only supported for 1D mode
# # Output File for XMDF Solution File # :SOLUTION_FILE_OPTION value (0 - BOUSS-2D Native, 1 - XMDF, 2 - Both) :XMDF_SOLUTION_FILE *file prefix* _sol.h5	Output format selection

Figure 1. BOUSS2D Parameter File Format.

Related Topics

- BOUSS-2D Files

BOUSS-2D Probes

BOUSS-2D can output the histories of the computed water surface elevation, velocities, force, and pressure at every grid point and at every time step. However, due to the number of data points in the domain, this is usually done at intervals of 15-30 min.

In order to provide a more complete temporal representation of the results of the calculation, the model allows specifying probes. At a probe location (x,y,z), specify what data should be saved and at what temporal resolutions. The options include water surface, velocity, force, and pressure.

BOUSS-2D Probe Manager

The probe manager allows creating, editing, and deleting probes. This dialog is only accessible when a BOUSS-2D grid exists. The properties associated with probes as follows:

- **.ts1 solution data loaded* – Indicates the highlight color for any fields that have solution data loaded.

- **Create Probes at Selected Cells** – This button will add probes to the probe list using any cells that were selected when the *Probe Manager* was opened.
- **Unload All *.ts1 Solution Data** – If any *.ts1 solution data has been loaded for any of the probe properties, the fields with that data will be highlighted. Clicking this button will unload the the solution data, allowing a new solution file to be generated when BOUSS-2D is run again.
- *Asign position by:* – These options will determine which coordinate fields to use in positioning the probes.
  - *Cell Position (I,J) – Activates the* i *and* j *fields.*
  - *(X,Y) Coordinate – Activates the* x *and* y *fields.*
- *Color* – Clicking on this button will bring up *Color* dialog. Altenatively, a simple color picker can be accessed by clicking on the arrow to the right.
- *Name* – A default name will be entered in this field, but by selecting the field any name can be typed in.
- *x* and *y* coordinates – These fields are only available when the *(X,Y) Coordinate* option is toggled on under the *Assign position by:* section. When active, specific coordinates may be entered which will move the probe on the grid.
- *i* and *j* coordinates – These fields are only available when the *Cell Position (I,J)* option is toggled on under the *Assign position by:* section. When active, specific coordinates may be entered which will move the probe on the grid.
- *WSE* – When checked, specifies using available water surface elevation data.
- *Pressure* – When checked, specifies using available wpressure data. The **Options** button will bring up the *Pressure Probe Options* dialog where elevation data can be entered.
- *Velocity* – When checked, specifies using available velocity data. The **Options** button will bring up the *UV Probe Options* dialog where elevation data can be entered.
- *Force* – When checked, specifies using available force data.

A probe can be deleted from the *Probe Manager* list by selecting the probe row number and hitting the *Delete* key.

Viewing Probe Solutions

After running the BOUSS-2D model,

are generated. These files can be imported into SMS to view the probe data. Right-clicking on a probe cell will have the following options for viewing probe data:

- **Eta Time Series Plot** – Creates an eta time series plot.
- **Pressure Time Series Plot** – Creates a pressure time series plot.
- **U Time Series Plot** – Creates a U time series plot.
- **V Time Series Plot** – Creates a V time series plot.
- **Basic Statistical Analysis** – Brings up a *Basic Statistical Analysis* dialog.
- **Zero-crossing Analysis** – Brings up a *Zero-crossing Analysis* dialog.
- **Spectral Analysis** – Brings up a *Spectral Analysis* dialog showing a spectral plot.

Basic Statistical Analysis

The *Basic Statistical Analysis* dialog includes the following information for each probe:

- *Mean (m)*
- *Minumum (m)*
- *Maximum (m)*
- *Standard Deviation (m)*

Zero-crossing Analysis

The *Zero-crossing Analysis* dialog contains the following information for each probe:

- *HAV (m)* – Average wavelength for the full time series.
- *H13 (m)* – Average of the highest 3% of Water Surface Elevation.
- *H110 (m)* – Average of the highest 10% of Water Surface Elevation.
- *HMAX (m)* – Highest Water Surface Elevation.
- *TAV (s)* – Average time periods for the full time series.
- *T13 (s)* – Average of the highest 3% of time periods.
- *T110 (s)* – Average of the highest 10% of time periods.

Related Topics

- BOUSS-2D
- BOUSS Runup / Overtopping

5.3. BOUSS Runup/Overtopping

BOUSS Runup / Overtopping

BOUSS Runup / Overtopping model can used to in the design of coastal structures. The model runs a simulation based on user-specified parameters.

Simulations are available in BOUSS Runup / Overtopping starting in SMS 11.1.

To create a new simulation:

- Right-click on empty space in the Project Explorer and select *New Simulation* | **Bouss Runup/Overtopping** .
- The *New Runup/Overtopping Simulation* dialog will appear. Select the coverages that will be created for the project and give each coverage the desired name. Click **OK** .

A new tree item for the simulation will appear in the Project Explorer. Also, all coverages will be created that were selectect in the *New Runup/Overtopping Simulation* dialog will appear under the "Map Data" item. Coverages can be added to the simulation by selecting the coverage and dragging it under the simulation item in the project explorer.

Simulation Menu

Right-clicking on the simulation item in the Project Explorer will bring up the simulation menu.

- **Delete** – Standard menu command to remove the simulation.
- **Duplicate** – Standard menu command create a copy of the simulation.
- **Rename** – Standard menu command to give the simulation a different name.
- **Model Control** – Launches the Runup/Overtopping *Model Control* dialog.
- **View Transect Profile** – Brings up the *Transect Profile* .
- **Export Runup/overtopping files** – Initiates creating and saving all BOUSS Run / Overtopping files necessary to run the model. Files will be saved in the project folder.
- **Launch Runup/overtopping** – Starts the BOUSS-2D model. A model wrapper should appear that shows the model run as it happens. Clicking the **Abort** button will end the model run early. Once the model run is complete, the **Exit** button can be selected.
- **Save project, export, and launch Runup/overtopping** – Completes the process of saving the project, exporting the model files, and running the model in one command rather than completing each step separately.
- **Statistics** – The *Statistics* dialog will open showing the values being used.

Coverage Menus

The right-click menus for Runup/Overtopping coverages are fairly standard. For each coverage (damping, probes, roughness, transects, and wave maker) there are the following options in the right-click menu:

- **Delete** – Standard menu command to remove the coverage.
- **Duplicate** – Standard menu command create a copy of the coverage.
- **Rename** – Standard menu command to give the coverage a different name.
- **Projection** – Opens an *Object Projection* dialog.
- **Metadata** – Opens the *Metadata* dialog.
- **Zoom to Coverage** – Fits any feature objects associated with the coverage into the graphics window. This command will proportionally resize and center the objects associated with the coverage so they all fit in the graphics window viewing space.
- *Type* – A sub-menu that allows seeing the current coverage type and changing the coverage type if necessary. The Runup/Overtopping model only uses the damping, probes, roughness, transects, and wave maker coverages and will not make use of other coverage types.

Probes

The probes coverage has an addition command option in the right-click menu.

- **Properties** – Brings up the *Probe Rules* dialog.

Transects

The transects coverage has an addition command option in the right-click menu.

- **Extract Elevations** – Launches the *Interpolation* dialog.

Runup / Overtopping Coverages

The data from various coverages in SMS is combined to create a BOUSS Run-up / Overtopping simulation. By clicking on the coverage in the Project Explorer and dragging the item under the Runup/Overtopping simulation item in the Project Explorer, the coverages become linked to the simulation. The different coverages are as follows:

Transects Coverage

Transects are made by creating arcs in the transects coverage. Each transect represents the 1-d grid used for a run-up simulation. It is recommended that these transects be linear. To assign elevations to the transects, use the right-click option: **Extract elevations** . This will prompt selecting a dataset to use for extracting. This must be done before launching a runup simulation.

Wave Maker Coverage

Wave makers are made by creating arcs in this coverage. The wave maker properties can be edited by double-clicking on the arc or by right-clicking and selecting **Attributes** . Each wave maker can have multiple sets of wave parameters. Each set of wave parameters will be run in a separate simulation. It is necessary that each wave maker arc in a simulation has the same number of wave parameters. The location of the wave makers on the transect is determined by the intersection of the transect arc and the wave maker arc. Each transect arc may have only one wave maker.

Probes Coverage

Probes are also made by creating arcs. The location of each probe on the transect is determined by the intersection of the transect arc and the probe arc. In the arc attributes, set the type of probe which can be any of the following: Velocity, Pressure, Water Surface Elevation and Force. To create a run-up probe, simply create a polygon in the desired location. The runup probe will be the portion of the transect arc that is inside of the polygon.

Auto-Create Probes

The *Probe Rules* dialog for the probes coverage can be used to automatically create probes.

Runup Probes

To automatically create a runup probe, define the following variables:

- **Minumum z** – Probes will not be created below this elevation
- **Maximum z** – Probes will not be created above this elevation
- **Delta z** – The difference in elevation that must occur for a probe to be created

When auto-creating these probes, SMS will traverse the transect arc, starting from the land and going seaward. Once a local maximum that is below "maximum z" is found, this point will be the starting point of the probe. SMS continues to traverse the arc until it reaches a point that is a distance of "delta z" below the starting point. This point will be the end point. However, if SMS reaches another local maximum with a higher elevation than the starting point, this point with the higher elevation will replace the original starting point.

Other Probes

To automatically create other types of probes, it's necessary to set the following parameters:

- **Elevation (m)** – The elevation (along the transect) at which the probe should be placed

- **Location** – Set to "First" or "Last". This is used only when the defined elevation occurs more than once on the transect arc. If "First" is selected, the probe will be placed at the first qualifying location on the transect arc. If "Last" is selected, the probe will be placed at the last location.
- **Probe Types** – The available probe types are: flowrate, WSE, force, pressure, and velocity. Set which type(s) should be created for this rule. For pressure and velocity, define elevation(s) above the seabed where they should be placed. These can be entered by clicking the **Define...** button next to their checkboxes.

Manual Probes

Feature arcs created in the probes coverage can be manually assigned probe properties. Right-clicking on the feature arc and selecting the **Attributes** command will being up an *Arc Attributes* dialog where the probe can be defined. This dialog has the following options:

- Probe – Defines the selected arc as a probe.
- Name – Allows specifying a name for the probe.
- WSE
- Force
- Flow Rate
- Pressure
- Velocity

Damping / Porosity Coverage

Damping and porosity attributes are created by making arcs. The type of attribute (damping or porosity) and its coefficient and width are set in the arc properties.

Roughness Coverage

If there is varying roughness throughout a transect arc, roughness polygons can be used to define the varying roughness. Each polygon that is created will be assigned a Chezy roughness value. To manually set this value, double-click on the polygon and assign use the *Roughness* dialog. The portion of the transect arc that is inside of the polygon will be assigned that value. Otherwise, the default Chezy coefficient (defined in the model control), will be used.

Note: It is helpful to turn on the display of inactive coverages while creating the wavemakers, probes, damping arcs, porosity arcs, and roughness polygons. This enables seeing the location of the transect arcs to ensure that they are intersecting the wavemakers, probes, etc., in the desired locations.

Related Topics

- BOUSS Runup / Overtopping Model Control
- BOUSS Runup / Overtopping Input Files
- BOUSS Runup / Overtopping Viewing Data

BOUSS Runup / Overtopping Model Control

The BOUSS2D *Runup / Overtopping Model Control* dialog is accessed by right-clicking on the Runup/Overtopping simulation and selecting the **Model Control** command. The dialog allows setting the following parameters:

- *Project Title* – Enter the title of the project in this field. The title is used for the folder name where the output files will be stored.
- *Input Datasets* – These inputs specify spatially varied input parameters for the model. In addition to the bathymetry, BOUSS-2D can utilize spatially varied damping, porosity and currents. In this section define whether these inputs are used ("None" is selected if not), and if so, how they are defined.
  - *Damping* – specified using boundary conditions on cell strings or selected from the scalar functions.
  - *Porosity* – specified using boundary conditions on cell strings or selected from the scalar functions.
  - *Current* – selected from the available vector functions.
- *Time Control* – The interface displays the recommended values for *Duration* and *Time step* in parentheses.
  - *Duration* – enter the model run duration.
  - *Time Step* – enter the number of time steps for the model run duration.
- *Grid Information* – This section displays the attributes of the active grid.
  - *Cell Size* – specifies the size of cells the model run will use.
- *Porosity Friction Factors*
  - *Turbulent* – value ranges from 0.0 to 3.6 with a default value of 2.4.
  - *Laminar* – value ranges from 0 to 1500 with a default value of 800.
  - *Store Size* – represents the characteristic stone size (d50) in meters of the breakwater armor layer and must be greater than zero.

    - *Added Mass Coeff.* –
- *Parameters*
    - *Roughness Type* – sets the Chezy coefficient type. Currently, only "Constant Chezy coefficent" is allowed. The Chezy coefficient ranges from 0.0 to 1000.0 (default = 50).
    - *Smagorinsky Number* – must be between 0.0 and 2.0 (default=0.2).
    - *Nonlinear Option* – can be set to either "Strong" or "Weak".
- *Output Options*
    - *Overide Default File Prefix* – allows specifying a file name for these solution files. The default file prefix is the filename of the *.par file.
        - *File Prefix* – becomes active once the *Overide Default File Prefix* is toggled on. A new prefix may be entered when active.
    - *Time Independent*
        - *Output Significant Wave Height*
        - *Output Mean Currents*
        - *Output Mean Water Level*
        - *Output Max Inundation Water Level*
        - *Output Format* – gives options to output in an "XMDF" file, a "BOUSS-2D Native" file, or "Both".
    - *Animation Output*
        - *Output* WSE – tells the model to output time varying datasets.
        - *Output Velocity* – tells the model to output time varying datasets.
            - Number of Frames
            - Required Memory
        - *Override Defaults* – Enables the *Begin Output* , *End Output* , and *Step* controls.
            - *Begin Output* – must be greater than 0.0 and less than the run duration.
            - *End Output* – must be greater than 0.0 and less than the run duration.
            - *Step* – determines how often the model will save time-varying output.

Related Topics

- BOUSS Runup / Overtopping

BOUSS Runup / Overtopping Input Files

The following files are used by BOUSS-2D in a run-up simulation:

- *.par
- bathy.grd
- porosity.grd (optional)
- damping.grd (optional)
- friction.grd (optional)

Each of these can be read/written by SMS. The prefix for each file will follow this format: "coveragename_arcId".

Parameter File

SMS will write a parameter file (*.par) for each set of wave parameters on each transect. This file contains model control information, 1-d grid data, and the wave maker parameters. Each parameter file represents a unique simulation.

Bathymetry File

This file (bathy.grd) defines the elevations along the 1-d grid (transect arc).

Porosity File

A porosity file (porosity.grd) will be written for each transect arc that is overlapped by a porosity arc. This file defines the porosity values along the 1-d grid. If no porosity arcs are defined, the model uses a default value of 1.0 along the entire grid.

Damping File

A damping file (damping.grd) will be written for each transect arc that is overlapped by a damping arc. This file defines the damping values along the 1-d grid. If no damping arcs are defined, the model uses a default value of 0.0 along the entire grid.

Friction File

A friction file (friction.grd) will be written for each transect arc that is overlapped by a roughness polygon. This file defines the roughness values along the 1-d grid. If no roughness polygons are defined, the model uses the Chezy coefficient defined in the model control along the entire grid.

Related Topics

- BOUSS Runup / Overtopping

BOUSS Runup / Overtopping Viewing Data

Transect Profile

By using the plot wizard or by right-clicking on the simulation icon in the project explorer, there is the option to view the profile of an individual transect. Each arc in the simulation can be viewed by selecting the desired arc in the Transect arc list on the bottom left portion of the dialog. The bathymetry is displayed with the locations of probes and wavemakers. On a separate plot, the damping, roughness, and porosity can be viewed. This plot is for viewing purposes only.

Solution View

After running a simulation, the data can be viewed in the *Solution* dialog. This dialog can be accessed through the *Plot Wizard* . There are two different views in this dialog: *Profile* and *Time Series* .

Profile

This view is used to display the bathymetry with the following datasets if they were turned on in the model control: WSE, velocity magnitude, mean water level, significant wave height, mean velocity, and maximum runup height. This allows viewing the data for multiple wave sets for a single transect at the same time. Once the desired data is selected, clicking the **Update** button will refresh the plot window to show the selected data.

Time Series

The time series view displays the probe data from the simulation. This view can display the data for any combination of transects, wave cases, and probes.

Statistics

If a simulation has been run, the **Statistics** option will be available in the right-click menu for the simulation. This *Statistics* dialog will display various statistics for velocity, pressure, force, and water surface elevation probes on each transect.

The *Time Series* Statistics display the following:

- *Minimum* – The minimum value in the time series
- *Maximum* – The maximum value in the time series
- *Mean* – The mean value for the time series
- *Standard Deviation* – The standard deviation for the time series

The *Eta Zero-Crossing* option displays:

- *Hav* – Average value of all peaks.
- *H1/3* – Value exceeded by 1/3 of the peaks.
- *H1/10* – Value exceeded by 1/10 of the peaks.
- *Hmax* – Maximum value of all peaks.
- *Tav* – Average period.
- *T1/3* – Period exceeded by 1/3 of the peaks.
- *T1/10* – Period exceeded by 1/10 of the peaks.

The *Runup Statistics* option displays:

- *Rmax* – Maximum value of all peaks.
- *R2%* – Value exceeded by 2% of peaks.
- *R10%* – Value exceeded by 10% of peaks.
- *R33%* – Value exceeded by 1/3 of peaks.
- *Rmean* – Average of all peaks.

Related Topics

- BOUSS Runup / Overtopping

5.4. CGWAVE

CGWAVE

<table>
<tr><td colspan="2">CGWAVE</td></tr>
<tr><td colspan="2">Model Info</td></tr>
<tr><td>Model type</td><td>General-purpose wave prediction model for simulating the propagation and transformation of ocean waves in coastal regions and harbors, and appropriate for modeling the most significant physical processes in channels, inlets and harbors, open coastal regions, around islands and structures.</td></tr>
<tr><td>Developer</td><td>Vijay Panchang, Ph.D.
Zeki Demirbilek, Ph.D.</td></tr>
<tr><td>Web site</td><td>CGWAVE web site</td></tr>
<tr><td>Tutorials</td><td>General Section
• Data Visualization
• Mesh Editing
• Observation
Models Section</td></tr>
</table>

	• CGWAVE Several sets of sample problems and case studies are available. These include: • Aquaveo Sample Problems • Model Validation cases from the CGWAVE website

The model CGWAVE (Demirbilek and Panchang 1998) is a two-dimensional wave transformation model that can be used to predict wave properties (wave heights, velocities, pressures, radiation stresses) in domains of complex shape and depth variations when an input wave condition (amplitude, direction, and period; or a spectral combination of these) is provided.

The CGWAVE model can be added to a paid edition of SMS.

Graphical Interface

SMS provides a graphical interface that is designed to to visualize the projects being created, easily modify project parameters, and view the solutions produced by the CGWAVE model.

The CGWAVE Graphical Interface contains tools to create and edit an CGWAVE simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to CGWAVE. If a mesh has already been created for a CGWAVE simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to CGWAVE. See Building a Mesh for more information.

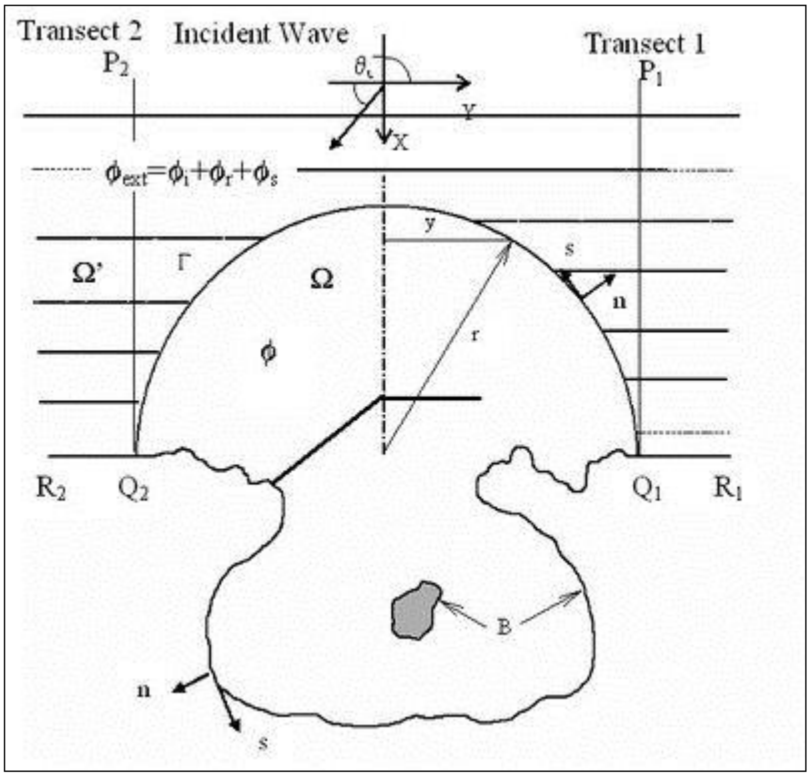
The interface consists of the 2D Mesh Module Menus and tools augmented by the *CGWAVE* menu . See CGWAVE Graphical Interface for more information.

Functionality

The model is based on extensions of the "combined refraction-diffraction" equation, which is applicable to both long and short waves and hence finds wide application in coastal engineering and harbor resonance studies. Being elliptic, the equation represents a boundary value problem, which can accommodate internal non-homogeneities (islands, structures, etc.) and boundaries. It hence forms a well-accepted basis for performing wave simulations in regions with arbitrarily-shaped (manmade or natural) boundaries and arbitrary depth variations without limitations on the angle of wave incidence or the degree and direction of wave reflection and scattering that can be modelled. In essence, it represents the complete two-dimensional wave-scattering problem for the non-homogeneous Helmholtz equation. Irregular wave conditions may be simulated using (1) by superposition of monochromatic simulations (e.g. Chawla et al. 1998; Panchang et al. 1990; Zhao et al. 2001.)

The wave phenomena that can be simulated with CGWAVE are: bathymetric refraction, diffraction by structures (e.g. breakwaters) and the bathymetry, reflection (from structures and natural boundaries (seawalls, coastlines, etc) as well as from bed slopes), friction, breaking, and floating (fixed) docks. The model uses a triangular finite-element formulation with grid sizes varying throughout the domain based on the local wavelength; the grids can be efficiently generated using the SMS graphical interface when a bathymetry file is provided. The model allows one to specify the desired reflection properties along the coastline and other internal boundaries. It is therefore particularly well-suited for simulating waves in harbors. While the basic equation is intended for monochromatic waves, irregular (i.e. spectral) wave conditions are simulated in CGWAVE through a linear superposition of monochromatic simulations (e.g. Panchang et al. 1990; Zhao et al. 2001.) Typically, simulations involving a domain containing hundreds of thousands of finite element nodes can be performed in a few minutes on a PC.

For harbor applications, the model also uses a semi-circle (as an open boundary) to separate the model domain from the outer sea. A typical CGWAVE model domain is shown in Fig. 1. The input conditions are provided at the offshore ends of two one-dimensional cross-shore sections. (In practice, the input condition is known at the end of one of the transects. The condition at the offshore end of the other transect is obtained by appropriate phase translation.) A combination of the incident and reflected waves is computed along these transects using a one-dimensional version of the governing equation; this partial solution is then mapped on to the semicircle to force the two-dimensional model. The remainder of the solution on the boundary consists of a scattered wave that emanates from within the domain; this component is allowed to radiate out through the use of an impedance boundary condition. In the model interior, a finite-element grid is used to represent the depth field, and the coastlines (denoted by B in Fig. 1) are assigned a reflection coefficient. For open ocean applications, all variations must be included inside a circle, outside which the depths are assumed constant (Panchang et al. 2000).



Functionalities

- CGWAVE Flies
- CGWAVE Graphical Interface
- CGWAVE Model Checker
- CGWAVE Model Control
- Saving CGWAVE files

Using the Model / Practical Notes

Methodology

When beginning a new project, first a domain must be created to perform the calculations. This domain comes from a combination of the coastline (land side of the domain) and the bathymetric survey. Begin by reading the survey into SMS and making sure it is an adequate and accurate representation of the study area. Ensure that the data covers the entire domain, and that the existing (or natural) coastline is adequately defined. If the survey includes topographic data, the coastline can be extracted as a contour of constant depth. Since CGWAVE does not support calculation of wave run-up, this should always be a positive depth (0.5 m or so is recommended). Follow the mesh generating techniques demonstrated in the tutorial problem for CGWAVE.

After creating the finite element mesh, it may be useful to make an initial run of CGWAVE in linear mode with fully absorbing coastlines and an incident wave condition. If desired, include both a normally incident wave run, and a separate obliquely incident wave. When CGWAVE is run in linear mode, the solution is unlimited and technically wave heights coming from the solution can grow uncontrolled. Numerically, there is no limit on wave heights. This is not realistic since waves in nature are finite in height (wave heights are depth-limited in shallow-water depths). However, the resulting solutions, especially the phase diagrams, can be used to perform visual quality control, since forward propagating waves are the easiest for making intuitive assessments (e.g. spacing between crests should be smaller and wave heights generally increasing in shallower water; bending of phase-lines for oblique incidence to approach the shore in a normal fashion, high wave heights in very shallow water because breaking was not applied, etc).

Repeating the incident runs with full reflection should lead to some standing wave patterns (large and small wave heights; rapid change of phase). A visual examination of the results will enhance confidence in later simulations.

Domain Shape

A mesh for use with CGWAVE must match one of two domain shapes. For cases involving a continuous coastline, the ocean boundary should be semi-circular. For cases examining waves around an island, the domain should be circular. The interface includes tools to build rectangular domains along coastlines, however, this is for research purposes only and should not be applied in projects with the current versions of CGWAVE. SMS includes tools in the Map Module to create a network with a valid domain shape (see the tutorial for CGWAVE). Therefore, it is strongly recommended that the Map Module be the principal method for creation of new networks. The Mesh Module may be used to create a mesh, however in the case of CGWAVE the Mesh Module primarily is useful for network editing and assigning model parameters required by CGWAVE. These model parameter commands are grouped in the *CGWAVE* menu.

To the extent possible, it is necessary to have the coastlines outside the semicircle be as straight as possible, and ideally, parallel to one of the axes (as shown in Fig 1). The semicircle should be so located that the depths outside vary, to the extent possible, in a 1-d (cross-shore) direction only in the exterior. These depths are introduced in the model through the two 1-d sections. It's necessary to specify a reflection coefficient for the exterior coastline.

Of course, these conditions are not encountered in practice in an ideal manner, but it is important to understand that the model works under these assumptions. V-shaped coastlines do not meet these criteria. Also, not all real domains have 1-dimensionally varying exterior depths. Regions with 2-d depth variations should preferably be incorporated inside the semicircle and it may be necessary to enlarge the domain so that solutions in the area of interest (hopefully) are not affected by the coastline shape. It is impossible to accommodate all real topographic variations of the model’s exterior domain. Since a part of what happens outside the semicircle influences the solution within (i.e. how the backscattered waves behave), some simplified representation of the exterior is needed. The exterior representation used in CGWAVE is more sophisticated than that used in earlier models (which assumed collinear and fully reflecting coastlines and constant exterior depths).

A full circle option is available for open sea problems where the exterior is of constant depth; the theoretical ideal for this is the Fourier Bessel Series (which works for unit wave input only); the parabolic mode may also be used (Panchang et al. 2000) with little loss of accuracy.

Domain Size

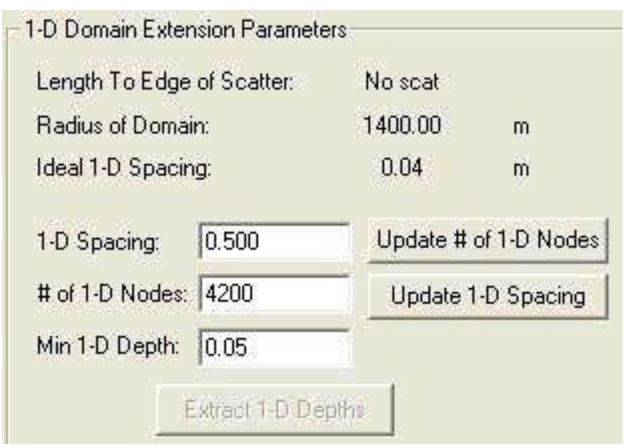
The size of the domain is also governed by the wavelength. Typically, the minimum radius for the semicircle should be about 2 or 3 times the wavelength, which also dictates the overall number of grids. It is important that the modeler first estimate the number of grid points needed, at least in an approximate sense. This can be done by estimating a nominal wavelength L (based on the period and depth), and using L/10 as a rough measure of the grid size. Obviously, large domains with short period waves will lead to hundreds of thousands of nodes. This is not a problem physically, but can be very time consuming to simulate. For spectral conditions, the domain size should be dictated by the longer wavelengths L_1 (say radius = $3L_1$), but the resolution for the shorter wavelengths (L_2) would be, say L_2 /10, which may be excessively fine for the longer wavelengths. Obviously, some care (and perhaps compromise) is needed in designing the grid and also selecting the input spectral components. If some components have little energy, it may be best to delete them from the modeling.

Boundary Conditions

The input wave condition (amplitude, period, direction) must be specified at the end of the 1-d cross-section. Simulations are less reliable (not necessarily wrong) if the waves approach the coast at a glancing angle (say within 20° of the coastline). The 1-d sections MUST reach out further than the outermost point of the semicircle. The domain should have no zero or negative depths; the land-water interface must be demarcated using a small water depth (say 0.5 m, but the limit depends on the overall water depths and wave periods and the problem being solved). Note that making this too shallow may result in a large number of grids in this area which may not be necessary for all applications).

For basic specifications, the incident wave angle specification is as follows. Waves going in the +x axis (i.e. to East) are specified as incident angle =0°. Waves going in +y direction (i.e. to North) are specified as incident angle =90°, to West =180°, to South = 270° . Incident wave direction is used in degrees. Convergence tolerance is typically assigned 10^{-8} , output frequency is 100 every iterations.

For the boundary condition specification, the *parabolic plus two 1d sections* option (Option 3 in *.cgi) should be chosen for most harbor problems, with a semicircle open boundary; in earlier versions, it is possible to use the option with just one 1d section (Option 2 in *.cgi) but it really provides no benefit over Option 3. The other options consist of the *parabolic approximation* (Option 1 in *.cgi) which is suitable if the exterior region is of constant depth. The *Fourier Bessel Series* (Option 0 in *.cgi) must be used if its limitations, described above in Section 2.2 (equations 15-17), are met; typically it is sufficient to specify 15 terms in the infinite series. For harbor problems, SMS will create the 1d sections based on depths based on depths on semicircle, and the desired grid spacing (used to solve the 1d equation (20)), based on (approximately) L/10 must be specified in the dialog box:

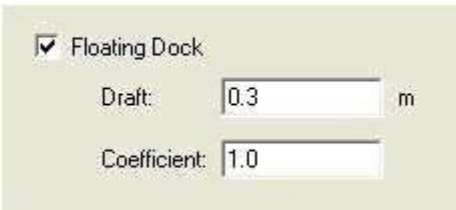


Wave Breaking

There are two methods for applying wave breaking. For practical problems, it's recommended to make a non-breaking simulation and then applying an H/d limit (generally between 0.6 and 0.85) to alter the calculated wave heights to the limit (say 0.78d). The other, more rigorous approach is to run the model in the nonlinear breaking mode where breaking properties are continuously recalculated based on prior solutions. This requires several rounds of iterations and is recommended for special problems and/or research applications. For nonlinear runs, the maximum number of nonlinear iterations is generally specified as 15 with a tolerance of 10^{-6}. For spectral simulations, obviously the former approach is recommended.

Docks

For incorporating docks, the actual "without-dock" water depth is originally specified while generating the grid; areas covering the docks must then be highlighted, and a draft and coefficient as shown in the dialog below.



The draft depth is a physical quantity and can be measured on an existing structure or designed. The coefficient, which is called alpha here, is a numerical term that impacts the conveyance of energy under the dock. It is a function of the wave number of the wave passing the dock (k), the characteristic structure size (a), the depth (h) and the draft (d) of the dock. A review of the theory can be found in Tsay and Liu (1983).

To compute an appropriate coefficient, the following process can be followed:

- Compute ka, kh and d/h
- Limit value of kh
  - if (kh > 4) kh = 4
  - if (kh < 0.1) kh = 0.1
- Compute initial value of alpha as:
  - if (kh <= 2) alpha = 0.97 - (0.65*kh) + (0.1*kh*kh)
  - if (kh > 2) alpha = 0.09 - 0.02*kh
- Adjust alpha for d/h
  - alpha = alpha*(1.4-0.8*(d/h))
- Adjust alpha for ka
  - if (ka < 1) alpha = 0.8*alpha

Visualizing Solutions

Solutions generated by CGWAVE can be visualized in SMS in two ways. The most straightforward method is to read the solution file. When SMS reads this solution file, it translates the data and creates datasets for wave amplitude, direction and phase. If additional functions are desired from the wave characteristics, they must be generated using a CGWAVE translating utility. An example of this is included in the CGWAVE reference manual. The translating utility creates generic dataset files that can be imported using the Data Browser.

Mesh Generation

CGWave computes properties that illustrate the characteristics of waves in a coastal area. To adequately perform this approximation, the mesh must contain several elements per wavelength. Steps for generating a mesh for CGWAVE are spelled out in the description of the Graphical Interface .

Saving CGWAVE

When completed a *File* | **Save As...** command, the following files get saved in the *.sms

- *.mat referenced to new save location
- *.map referenced to new save location
- *.cgi referenced to new save location
- *.h5 referenced to new save location
- *.cgo referenced to old save location unless rerun then referenced to new save location

What does the error message “Number of Nodes Exceeded the Dimension” mean?

This message indicates the mesh has more elements than the program is dimensioned for. Running with more nodes/elements than the program is dimensioned for is not recommended. CGWAVE can be compiled for larger dimensions if necessary for a specific case. Contact tech support for more information.

Memory Requirements and Notes

Currently the CGWAVE engine is written in FORTRAN. All memory management in the program is handled with static arrays. An effort is currently underway to rewrite the code to use dynamic arrays. The executable distributed with SMS has been compiled to handle meshes with up to approximately 480,000 nodes. The number is not exact because some of the arrays dimensions are set based on the number of nodes on the open boundary and the coastline, which are very domain specific. In order to obtain an executable that will handle a larger number of nodes, contact technical support at Aquaveo. Larger executables are often posted on their ftp site.

It should be noted that due to the limitation of static arrays, the size of model that can be evaluated on a Windows PC is limited to a maximum of around 2,500,000 nodes. Be warned that running such a large model on a PC may take an extended period of time (possibly days).

With the availability of 64-bit machines, it is possible to access larger amounts of memory. However, this will not be an option for CGWAVE until the code is reworked to support dynamic arrays and tested on 64-bit machines.

Theoretical Basis / Mathematical Details

- Governing Equations
- Boundary Conditions
- Numerical Solution

Additional Documents:

- Simulation of Waves in Harbors Using Two-Dimensional Elliptic Equation Models [96]
- Solution of the Mild-Slope Wave Problem by Iteration [97]
- A Finite Element Model for Wave Refraction and Diffraction. Tsay, T.-K. & P.L.-F. Liu (1983). Appl. Ocean Res., v5, 1, 30-37.
- Simulation of Wave Breaking Effects in Two-Dimensional Elliptic Harbor Wave Models [98]
- Simulation of Waves at Duck (North Carolina) Using Two Numerical Models [99]
- Incorporating Rubble Mound Jetties in Elliptic Harbor Wave Models [100]
- Exterior Reflections in Elliptic Harbor Wave Models [101]
- Exterior Bathymetric Effects in Elliptic Harbor Wave Models [102]
- Improved Coastal Boundary Condition for Surface Water Waves [103]
- Outgoing Boundary Conditions for Finite-Difference Elliptic Water-Wave Models [104]

External Links:

- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [105]
- Aug 2005 The Usage of CGWAVE in SMS: A User's Guide [106]
- Mar 2004 ERDC/CHL CHETN-I-68 How to Use CGWAVE with SMS:An Example for Tedious Creek Small Craft Harbor [107]
- Jun 2003 ERDC/CHL CHETN-I-67 Tedious Creek Small Craft Harbor:CGWAVE Model Comparisons Between Existing and Authorized Breakwater Configurations [108]
- Aug 1998 Technical Report CHL-98-xx CGWAVE: A Coastal Surface Water Wave Model of the Mild Slope Equation [109]

Related Topics

- SMS Models page

CGWAVE Graphical Interface

The CGWAVE Graphical Interface includes tools to assist with creating, editing, and debugging a CGWAVE model. The CGWAVE interface exists in the 2D Mesh Module .

Model Construction Steps

There is a very consistent method that can be used to apply the CGWAVE model. The steps to this process include:

1) Load bathymetry – This data can come from LIDAR surveys, digital elevation maps (DEMs), previous grids or a variety of other sources. They must be referenced to the same datum the wave data will reference, and must have positive values represent depths. SMS includes functionality to convert datums, reverse directions and smoothor filter data.
2) Limit bathymetry to positive values – CGWAVE does not handle wetting/drying or runup processes. All the nodes in the model must have a positive depth. This limiting process can be handled later in the mesh generation process, or the bathymetry itself can be modified (in a copied dataset) using the data calculator . For example, if wanting to limit the domain to areas of at least one meter of depth, use the following equation in the data calculator *max(d1, 1.0)* (where d1 is the label for the depth dataset).
3) Compute the wave length – This is also done in the dataset toolbox, using the *Wave Length and Celerity* tool. Enter the smallest wave period of interest (to generate the highest needed resolution).
4) Create a size function – Typically this is simply a scaled version of the wave length dataset. For example, a basic rule would be to have at least six elements per wave length. Some experts recommend at least 10 elements per wave length. The real issue is that there needs to be enough resolution to represent the wave shape.
    1) To create a "Size" function with "N" elements per wave length, go to the *data calculator* and enter the equation *d3/N* (where d3 is the label for the wave length dataset).
    2) An alternative that may be needed with large domains is to create a spatially varied scale for the wave length function. For example to have 15 elements per wave length in the shallow region or the model (i.e. depths less than 3 meters), but only have 7 elements per wave length in the deep regions (i.e. depths greater than 200 meters). To create this size function, still use the data calculator and enter the equation *d3/max(min(15+(d1-3)/(200-3)*(7-15),7))* (where d1 is the label for the depth function and d3 is the label for the wave length dataset).
5) Define the coastline or land edge of the domain – This can be done using a contour of the bathymetry or reading a coastline vector file. It should be stored in SMS as a coastline arc in a CGWAVE coverage in the map module .

6) Define the ocean boundary – This is also an arc in the CGWAVE coverage. It must be either a semi circle or circle and can be defined in SMS by selecting the coastline (or extreme locations on the coastline) and issuing the *Feature Objects | Define domain...* command in the map module .
7) Build polygons in the map module and assign polygon attributes for the polygon to use the depth and size functions for bathymetry and size respectively in the mesh generation process. Then generate the mesh.
8) With this mesh constructed, the rest of the graphical interface, defined below can be used to control the numerical simulation.

CGWAVE Menu

CGWAVE makes use of the standard menus along with the *CGWAVE* menu. The following menu commands are available in the *CGWAVE* Menu:

Spectral Energy

Brings up the *Spectral Energy* dialog to define/view wave energy spectra. This command also allows visualizing wave spectra.

Assign BC

Assigns boundary conditions along a selected nodestring(s) .

Material Properties

Opens the *CGWAVE Material Properties* dialog.

Model Check

Check for common problems. The *model checker* performs the generic mesh checking along with optionally checking to insure:

- that all boundaries mesh boundaries are assigned as land with a reflection coefficient or as open ocean.
- that all water depths are positive.

Model Control

Brings up the *Model Control* dialog to specificy model parameters.

Reset 1D Spacing

Brings up the *Modify 1D Wave Line* dialog.

Run CGWAVE

Brings up a dialog that checks what executable of CGWAVE should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

Model Control

The *CGWAVE Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In CGWAVE boundary conditions are defined on nodestrings . The default boundary condition is a closed boundary (no flow). See CGWAVE Boundary Conditions Dialog for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

- *Bottom friction* – The bottom friction can be specified for the element(s) and material selected in this field.

- *Floating dock* – Represent an object anchored in place, but floating in or on the water and thereby obstructing wave fields. Elements will be treated as floating barriers when the simulation is saved.

Running the Model

The CGWAVE Files are written automatically with the SMS project file or can be saved separately using the *File* | **Save CGWAVE** or *File* | **Save As** menu commands.

CGWAVE can be launched from SMS using the *CGWAVE* | **Run CGWAVE** menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *CGWAVE* | **Model Check** menu command.

Processing Solutions

CGWave creates a single output file (normally including *.cgo extension when run with SMS). This file can be brought into SMS to graphically view the results. As SMS reads the file, it translates (using an embeded version of the CGWAVE "trans" code) the complex numbers representing wave heights and phases for each wave component into spatial and temporal datasets including:

- Steady State
    - Wave Height
    - Wave Phase
    - Direction of Maximum Particle Velocity
- Time Varying (through a single wave period. SMS breaks the period into 20 time steps)
    - Pressures (at surface, mid depth and bed)
    - Particle Velocities (at surface, mid depth and bed)
    - Sea Surface Elevation
    - Wave Velocity.

When the solution is read in, SMS allows limits to the wave heights in the solution. This only applies to linear runs of CGWAVE to allow the heights to be adjusted to be more realistic. This is accomplished by applying a factor, whose value ranges from 0 to 1 (defined as H/d). (Because wave height cannot exceed the value of depth or factor=1). SMS recommends a range of 0.4 to 0.8 and defaults to 0.64. If desired, enter another factor dependent on knowledge of wave mechanics, type of problem, etc.

Related Topics

- CGWAVE

CGWAVE Files

The input and output files for a CGWAVE model are as follows:

Input Files

The *.cgi File

CGWVE always requires an input file to run. This input file is created by SMS with an extension of *.cgi (CGwave Input). It is an ASCII file that defines the model parameters, the boundary definition, the nodal locations and the connectivity. Lines that begin with a "%" indicate the line is a comment. These are terminated with a "&" character. There is a comment for each value on the data line, so five comment lines indicate the following data line will have five values. This format is generally followed throughout. Some utilities available for processing CGWAVE data files require that all the "standard" comments exist. The model parameter data is sequentially read, and follows the following format:

%number of characters in title &

```
%number of terms in the fourier-bessel series solution (default = 35) &
```

```
%output echo frequency to screen & %
maximum number of iterations for convergence &
%max no of iters. for nonlinear mecha (fri,breaking,dispersion) &
           30          35         100       500000          10
```

Title of this Run, jul 10 2007

```
%type of application &
%type of open boundary condition &
%bottom friction &
%wave breaking &
%nonlinear dispersion relation &
%choice of solver &
           0           3           1           1           0           2
%number of components &
           1
%Incident Wave Angle || Wave Period(s) || Incident Wave Amplitude &
  225.000000000000000E+00    30.00000000000000       1.000000000000000
%exterior reflection &
%bottom friction coefficient &
%wave breaking parameter &
%tolerance for the equations &
%tolerance for nonlinear mechanisms &
   0.00E+00     0.12E+00     0.15E+00     0.10E-09     0.10E-06
```

After the model parameter data comes the boundary definition. This includes two blocks of data marked with "&C" to indicate a boundary open to have waves enter, and "&B" for sections of coastline and islands (land boundaries). There can only be one "Open" boundary (&C). Its type will be 0 or 1 indicating semi-circle or full circle. The full circle is applied when modeling waves around an island. Multiple closed boundaries (&B) may be included. One for each segment of the coastline (changing reflection coefficients), and at least one for each island in the domain.

%Open boundary flag

```
%type of open boundary &
%number of nodes on it &
%center of the circle - x &
%center of the circle - y &
%initial angle &
%semicircle orientation angle (ccw from +x axis, E=0, N=90, W=180, S=270)
&
%node id, ... ...
```

&C

```
0   220  0.000000000E+00  0.000000000E+00  0.000000000E+00  0.230036700E+02
          220          219          218          217          216          215
          214          213          212          211          210          209
          208          207          206          205          204          203
          .
          .
          .
          .
         .
```

```
%Coastline boundary flag
%number of nodes on it &
%reflection coefficient &
%node id, ... ...
```

&B

```
 360   5.0000000000000003E-02
   1          221          440          659          880          881
```

```
1103         1329         1559         1793         1794         2036
2285         2538         2798         2799         2800         3070
 .
 .
 .
 .
 .
```

Following the boundary definition, the file includes the locations of the computational nodes with a "&N" identifier:

```
%Node  coordinates  and  Depth  &
%number  of  nodes  &
```

&N

```
       236211
-0.452985572E+04   0.259162993E+04   0.215026303E+00  -0.452888341E+04
0.259197240E+04   0.242541197E+00
-0.452784808E+04   0.259233707E+04   0.271839587E+00  -0.452671893E+04
0.259273479E+04   0.303792731E+00
-0.452546078E+04   0.259317794E+04   0.334700922E+00  -0.452409490E+04
0.259365905E+04   0.361240031E+00
          .
          .
          .
          .
          .
```

Following the nodal locations comes the material definitions with a "&M" identifier. Material types define two important options for CGWAVE. First, roughness is specified as a material. Second, floating breakwaters are defined in the material block. Each type of breakwater, and each roughness zone will require it's own material type. The materials are then assigned to each element in the element block below. In the example below, two materials are defined. The first has a roughness of 0.5 and is type 0, indicating it is not a floating breakwater. The second has the same roughness and is of type 1, indicating it is a floating breakwater. For the floating breakwater, two additional parameters are included, a draft and a coefficient of transmission.

```
%Materials
%number  of  materials  &
%material  id  &
%bottom  friction  &
%floating  dock  &
%draft  &%coefficient.
```

&M 2 1 0.5000 0 0.0000 0.0000 2 0.5000 1 0.0000 1.0000

Following the material definitions comes the element connectivity with a "&E" identifier. Each line includes the three node identifiers that make up an element and the material type that fills that element.

```
%Elements
%number  of  elements  &
```

&E

```
       456220
            1          221            2         1
          221          222            2         1
          222            3            2         1
          .
          .
          .
          .
          .
```

*The *.cg1 File*

When running CGWAVE from for varied sloped, semi-circular boundaries, it is recommended that the two "Semi 1D Lines" option be used. This option transforms the input wave conditions from the offshore depth to a variety of depths as those conditions propagate shoreward. The transformed conditions are mapped to the variable depth boundary of the semi-circle (open boundary). SMS extracts depths from the surrounding survey if it exists. Otherwise it is necessary to create a profile from the deep water location where the wave conditions are sampled, through to the coastline. The file format includes two such profiles. One marches down the left side of the semi-circle, and the other down the right. The format of each profile includes the distance between sample points and the number of sample points. Two other values are included as space holders for now. SMS always writes out "0.0 1" for these values. The file then includes one line for each sample point including a depth and an optional roughness value at that depth as follows:

```
0.5000  4912   0.0   1
    73.394101     0.120000
    73.223910     0.120000
    73.053719     0.120000
          .
          .
          .
          .
    9.177065     0.120000
    7.433389     0.120000
    5.914170     0.120000
```

Output Files

CGWAVE creates a single ASCII solution file. It includes a block of data for each input wave condition. The file includes a complex number that defines the wave amplitude and phase at each node for each wave condition. Upon trying to read this file into SMS, the CGWAVE – Trans post-processor will be executed in order to turn the *.cgo solution file into an XMDF file that can be easily read by SMS.

Related Topics

- CGWAVE

CGWAVE Math Details: Governing Equations

Governing Equations

In its basic form, the methodology is based on solving the following two-dimensional elliptic equation:

$$\nabla \bullet (CC_g \nabla \phi) + k^2 CC_g \phi = 0 \quad (1)$$

where

$\phi(x, y)$ = complex surface elevation function $(= \phi_1 + i\phi_2)$

i = $\sqrt{-1}$

σ = wave frequency under consideration

$C(x, y)$ = phase velocity = σ / k

$C_g(x, y)$ = group velocity = $\partial\sigma / \partial k$

$k(x,y)$= wavenumber $(=2\pi/L)$, related to the local depth $d(x,y)$ through the dispersion relation:

$$\sigma^2 = gk\tanh(kd) \quad (2)$$

The wave height H can be obtained from complex surface elevation function as follows:

$$H = \partial\phi/\partial k \quad (3)$$

Essentially (1) represents an integration over the water column of the three-dimensional Laplace equation used in potential wave theory. The integration, originally described by Berkhoff (1976) and Smith and Sprinks (1975), is necessary because the solution of the three-dimensional problem is computationally difficult for harbors with a characteristic length that is several times the wavelength. The integration is based on the assumption that the vertical variation of the wave potential is largely the same as that for a horizontal bottom, i.e.

$$\phi(x,y,z) \approx \frac{\cosh k(d+z)}{\cosh kd}\phi(x,y) \quad (4)$$

This approximation is obviously valid for a "mild slope", characterized by $|\nabla d|/kd \ll 1$, a criterion that is usually met in practice. (Extensions to steep slopes are described later). Unlike "approximate" mild slope wave models (e.g. REFDIF and RCPWAVE described by Dalrymple et al. 1984; Kirby, 1986; and Ebersole, 1985), there are no intrinsic limitations on the shape of the domain, the angle of wave incidence, or the degree and direction of wave reflection and scattering that can be modeled with (1). While (1) is valid for a monochromatic (single incident frequency-direction) wave condition, irregular wave conditions may be simulated using (1) by superposition of monochromatic simulations.

As noted earlier, (1) incorporates the effects of refraction, diffraction, and reflection induced by any nonhomogeneity in the model domain. We now provide extensions of (1) that include, in addition, dissipative effects (friction and wave breaking), steep-slope effects, and floating docks.

Dissipation

To include dissipative effects, we consider the following extended form of (1):

$$\nabla \bullet (CC_g\nabla\phi) + (k^2CC_g + iC_g\sigma W)\phi = 0 \quad (5)$$

in which a dissipation term (with W) has been included. By separating the real and imaginary parts of (5), Booij (1981) has shown that (5) satisfies the energy balance equation in the presence of dissipation. The term W may represent breaking and/or friction and is described later.

In (5), W represents the combined effects of friction and breaking, which may be separated as follows:

$$W = w/C_g + y \quad (6)$$

where is the friction coefficient defined by Dalrymple et al. (1984) and g is a breaking factor. These coefficients are empirical, and parameterizations for these have been described by Dalrymple et al. (1984), Tsay et al. (1989), and Chen (1986) for friction and by Battjes and Janssen (1978), Dally et al. (1985), Massel (1992), Chawla et al. (1998), and Isobe (1999) for breaking. Some of these parameterizations have been extensively validated against field data (e.g. Larson 1995; Kamphuis 1994). The parameterization used in CGWAVE is based on the formulation by Dalrymple et al. (1984).

Published studies demonstrating the effects of friction in harbor models (e.g. Chen 1986; Tsay et al. 1989; Demirbilek and Panchang 1998; Kostense et al. 1986) have estimated w on the basis of the incident wave amplitude. It is then easy to pre-specify w while solving (5). These studies appear to show that friction can change the magnitude of resonant peaks in harbor models quite substantially; at other frequencies, the effect seems to be minimal.

As to breaking, Zhao et al. (2000) applied a finite element wave model to several tests involving breaking. These tests involved a sloping beach, a bar-trough bottom configuration, shore-connected and shore-parallel breakwaters on a sloping beach, and two field cases in the North Sea and Ponce de Leon Inlet (Florida). Five breaking formulations, given by Battjes and Janssen (1978), Dally et al. (1985), Massel (1992), Chawla et al. (1998), and Isobe (1999) were examined. In general, they found that the formulations of Battjes and Janssen (1978) and Dally et al. (1985) were the most robust from point of view of incorporation into an elliptic model based on (5) and provided excellent results compared to data. CGWAVE includes the Battjes and Janssen (1978) formulation.

Unlike friction the breaking coefficient is a function of the wave amplitude which is unknown *a priori* inside the domain, and its inclusion makes the problem nonlinear and requires iteration. For the first iteration, W is set equal to 0 and (1) is solved (e.g. non-breaking solutions are obtained). The resulting wave heights are used to estimate W via the Battjes and Janssen (1978) parameterizations and (5) is solved again. The process is repeated until convergence is obtained. This can be a very time –consuming process. For many practical applications, it is therefore suggested that simple non-breaking simulations be performed, and an H/D breaking limit be then applied to artificially cut off the excessively large waves. This option is available in CGWAVE.

Steep-slope effects

Unlike the nonlinear mechanisms described above, the “mild slope” requirement discussed in Section 1 is relatively easy to eliminate. Massel (1993), Porter and Staziker (1995), Chamberlain and Porter (1995), and Chandrasekera and Cheung (1997) developed extensions of (1) to include steep-slope effects. Their extensions may be described by the following equation:

$$\nabla \bullet (CC_g \nabla \phi) + (k^2 CC_g + d_1(\nabla h)^2 + d_2 \nabla^2 h)\phi = 0 \quad (35)$$

Where d_1 and d_2 are functions of local depths. Reference may be made to these publications for the various definitions of d_1 and d_2; in general, though, differences in the proposed definitions of these functions impact model results to a very small extent. The steep-slope terms are fairly straightforward to include in the model because they are linear. Further, they have the advantage of being “automatic”, i.e. they have little contribution for mild slopes and do not change the solution technique; when they are significant, the additional computational demand is negligible. However, steep slopes lead to breaking and model performance in the vicinity of steep slopes (e.g. see the work of Massel and Gourlay (2000) that include breaking and steep-slope effects near coral reefs). CGWAVE uses the formulation by Chandrasekera and Cheung (1997).

Floating Docks

One problem frequently encountered by engineers when using models based on (1) pertains to the presence of floating structures in the modeling domain (e.g. floating breakwaters or docks in marinas). These structures of course violate the “free-surface” requirement of (1). The problem near the dock is 3-dimensional, whereas the model (eq. 1) is solved in a 2-dimensional framework. CGWAVE uses an approximate method suggested by Tsay & Liu (1983) for tackling floating structures in the context of 2-d harbor wave models. This approach merely calls for a suitable modification to the second term on the left-hand side of (1). (Tsay and Liu (1983) examined suppressing this term). As a consequence, the method is extremely simple to implement with existing finite element models. A model grid is first generated as usual with no regard to the floating structure, grid elements covering the floating structure (in plan view) are selected, they are assigned a depth value equal to the under-keel clearance, and the coefficient of the second term in (1) is set to zero for these elements. Clearly, this is an ad-hoc method intended for convenience in engineering practice, and although Tsay & Liu (1983) provided heuristic arguments in support of this approach, their testing of this procedure was rather limited. Li et al. (2005) found, however, that the method produces results which deviate considerably from the solution of the Laplace equation, and hence developed a simple modification to the original Tsay & Liu (1983) approximation. This involves adjusting the under-keel depth by a factor $\alpha = A\ln(ka)+B$, where a = half width of the structure, and A and B are given in Figure 2 (and Table 1) for different values of relative submergence (defined by draft/water-depth = d/h). The modified approximation yields improved results, when compared to both laboratory data and theoretical results, for a wide range of conditions. By way of practical demonstration, simulations in Douglas Harbor (Alaska) was described by Li et al. (2005) for examining the effects of proposed floating dock configurations. The factor α must be provided to CGWAVE along with the draft depths during grid generation.

Related Topics

- CGWAVE
- Boundary Conditions
- Numerical Solution

CGWAVE Math Details: Boundary Conditions

Domains on which the elliptic eq. (5) is solved are enclosed by closed boundaries (represented by coastlines and surface-penetrating structures like pier walls or pier legs, breakwaters, seawalls, etc.) and open boundaries (which represent an artificial boundary between the area being modelled and the sea region outside. A separation between the model domain and an outer water area from where no waves enter the model domain (e.g. a creek or tributary at the backbay or down wave end of the domain) may be considered to be a fully-absorbing closed boundary. An open boundary is considered to be one where an incident wave is specified (and may contain other radiated waves). Along these boundaries, appropriate conditions must be specified to solve (1); however, even in the best of circumstances, only approximate boundary conditions can be developed (e.g. see Dingemaans, 1997).

Closed Boundary Conditions

Along coastline and surface-protruding structures, the following boundary condition has traditionally been used (e.g. Berkhoff 1976; Tsay & Liu 1983; Tsay et al. 1989; Oliveira and Anastasiou 1998; Li 1994a)

$$\frac{\partial \phi}{\partial h} = \alpha \phi \quad (6)$$

Where n is the outward normal to the boundary and a is related to a user-specified reflection coefficient as follows:

$$\alpha = ik \frac{1 - K_r}{1 + K_r} \quad (7)$$

K_r varies between 0 and 1 and specific values for different types of reflecting surfaces have been compiled by Thompson et al. (1996).

It may be verified that (6) is strictly valid only for fully-reflecting boundaries (i.e. K_r = 1). For partially reflecting boundaries, it is valid only if waves approach the boundary normally. For other conditions, (6) is approximate, and may produce distortions in the model solutions. These limitations may be eliminated by describing the solution at the boundary more fully as a sum of incident and reflected waves:

$$\phi = A\{\exp[ik(n\ \cos\theta + s\ \sin\theta)] + K,\ \exp[ik(-n\ \cos\theta + s\ \sin\theta + \beta)]\} \quad (8)$$

where A is the amplitude of the approaching waves, θ is the direction at which they intersect the boundary (θ = 0 for normally incident waves), is the coordinate along the tangent to the boundary, and β is a phase shift between the incident and the reflected wave. (8) leads to the following boundary condition.)

$$\frac{\partial \phi}{\partial n} = ik \cos\theta \frac{1 - K_r \exp(ik\beta)}{1 + K_r \exp(ik\beta)} \phi \quad (9)$$

Unfortunately, θ and β are not known a priori inside the model domain, and must be estimated by approximation. For fully absorbing boundaries ($K_r = 0$), Li and Anastasiou (1992) and Li et al. (1993) have used (9) after estimating θ from Snell's Law and the deep-water incident wave angle. Alternatively, Isaacson and Qu (1990) estimated as follows:

$$\theta = \arctan\{(\partial\chi/\partial s)/(\partial\chi/\partial n)\} \quad (10)$$

Where χ is the argument of the complex quantity θ (i.e. the phase of θ). For implementation, they first used (6) as a boundary condition, obtained χ from the results, determined θ from (10), used (9) as a boundary condition to perform a second iteration of the model, recalculated χ and θ, performed a third model iteration using (9), and so on. Like Pos (1985), they assumed $\beta = 0$ while using (9), based on limited numerical tests that showed little sensitivity to . Clearly, like the Snell's Law approach, (10) is valid only for $K_r = 0$ (although problems with non-zero K_r were also considered). To include the effect of the reflected waves (i.e the second term in the right hand side of (8)), Isaacson et al. (1993) suggested estimating θ as follows:

$$\theta = (1/k)\arcsin\{\partial\chi/\partial s\} \quad (11)$$

Again an iterative method with repeated model calculations were needed. Steward and Panchang (2000) analyzed these methods and noted difficulties with convergence of the above iterative methods and with the quality of the solutions obtained with (10) and (11). They were able to eliminate these difficulties by estimating from the following expression:

$$\tan\theta = \frac{\partial\chi}{\partial s} \Bigg/ \left(\frac{\partial\chi}{\partial n} + \frac{2K_r k(\cos(k\beta) + K_r}{1 + 2K_r\cos(k\beta) + K_r^2}\cos(\theta)\right) \quad (12)$$

(12) is a generalization of (10) that allows non-zero K_r and β;. For a detailed comparison of results, see Steward and Panchang (2000). Fig. 2 shows a simulation with of waves propagating into a rectangular harbor area. Clearly, solutions obtained with (12) are qualitatively superior to those obtained with (10).

Despite the increasing sophistication seen progressively in (6) and (9) and in the various ways of estimating θ, some fundamental problems remain. The most important one is inherent in (8), i.e. the assumption that the total wave field near the boundary can be represented either by one set of plane waves (in the case of (10) or the Snell's Law approach of Li et al. (1993)) or by two sets of plane waves (in the case of (11) and (12)) propagating in constant depth. In domains of complex shapes (as in Fig. 1) with arbitrary bathymetry and boundaries with varying reflectivities, a complex pattern of waves can result; simple wave trains are not easily discernible and, as noted by Isaacson and Qu (1990), the definition of a single θ (and β) can become meaningless. Further, even when there is a well-defined train of waves near the boundary (justifying the use of the above methods), precise estimation of K_r and β is still problematic. Values K_r of provided by Thompson et al. (1996) certainly do not cover full range of reflecting surfaces that the modeller encounters, nor do they cover the dependence of these parameters on the incident wave frequency. Efforts to incorporate the work of Dickson et al. (1995) and Sutherland and O'Donoghue (1998) pertaining to β in models such as the one described here are lacking.

In some ways it may be best to recognize these difficulties at the outset and use the simplest expressions (6) and (7) by combining all the uncertainties noted above into a single parameter α, which may be regarded as a tuning parameter. This is the approach followed in CGWAVE.

Open Boundary Conditions

Along the open boundary, an incident wave φ_i must be specified. Along this boundary, however, waves backscattered from within the domain will also exist, and their magnitude is generally not known. In the context of simple rectangular domain models, with one side (aligned, say, in the y direction) constituting the open boundary, Panchang et al. (1988, 1991), Li (1994a, b), and Oliveira and Anastasiou (1998) have used the following condition

$$\frac{\partial\phi}{\partial x} = ik(2\phi_i - \phi) \quad (13)$$

(13) is obtained by assuming that the incident and backscattered components along this boundary can be described by $\phi_i = A_i\exp(ikx)$ and $\phi = B\exp(-ikx)$ respectively (where A_i is the (specified) amplitude of the incoming wave and B is an unknown), adding the two components, and differentiating. Obviously, this is valid only if the incident and backscattered waves near the boundary are plane waves propagating in the +/- x direction.

For more complex domains involving multidirectional scattering, (13) is inappropriate. Harbor applications generally use model domains such as that described in Fig. 3, where the semicircle is used to separate the model area from the open sea. In the exterior domain Ω' the potential ϕ is comprised of three components:

$$\phi = \phi_i + \phi_r + \phi_s \quad (14)$$

where ϕ_i = the incident wave that must be specified to force the model, ϕ_s = a reflected wave that would exist in the absence of the harbor, and ϕ_s = a scattered wave that emanates as a consequence of the harbor and must satisfy the Sommerfeld radiation condition. With appropriate descriptions for these components, a boundary condition can be developed along the semicircle.

In traditional harbor models (Mei 1983; Tsay and Liu, 1983; Thompson et al. 1996; Chen and Houston, 1987; Xu and Panchang, 1993; Demirbilek and Panchang, 1998), the exterior wave conditions are described as follows:

$$\phi_i = A_i[ikr\cos(\theta - \theta_i)] \quad (15)$$, which is the specified input

$$\phi_r = A_i[ikr\cos(\theta - \theta_i)] \quad (16)$$

$$\phi_s = \sum_{n=0}^{\phi} H_n(kr)(A_n \cos n\theta + B_n \sin n\theta) \quad (17)$$

where (r, ϕ) denotes the location of a point in polar coordinates, H_n is the Hankel function of the first kind and order n, and A_n and B_n are unknown coefficients.

For the specified incident wave field given by (15), equations (16) and (17) result from the solution of the relevant eigenvalue problem in the traditional method. As demonstrated by Xu et al. (1996), however, this eigenvalue problem, in which ϕ_s and ϕ_r are coupled, may be solved only under the following conditions:

(i) The exterior region must have a constant depth, (ii) the exterior coastlines A1D1 and A2D2 must be fully reflecting and collinear.

These requirements usually cannot be met in practice where the exterior geometry varies arbitrarily, and the unrealistic bathymetric representation used perforce by the modeller invariably has an adverse influence on the solution. In field applications, the exterior bathymetry is irregular and the depth generally increases in the x-direction. Condition (i) is thus violated, causing two problems as demonstrated by Panchang et al. (2000). First, the modeller must arbitrarily select a representative “constant” depth and test the sensitivity of the solutions to these depths. This can be extremely time-consuming. Second, the effect of reflections from the sloping exterior bathymetry is ignored. These effects are often significant, especially for long periods that are of interest in harbor resonance studies. Condition (ii) is also problematic. Exterior coastlines are not always fully reflecting for all wave conditions, and imposing full reflection in such cases yields extremely large amplification factors and rapid variations in the wave pattern in the outer regions of the domain. (See examples in Xu et al. (1996), Demirbilek et al. (1996), and in Thompson et al. 1996)). One may of course enlarge the interior region in the hope that these effects do not contaminate the results in the area of interest; however, there is no guarantee that these effects are confined to specific regions. In addition, the extra memory requirements and grid-generation for a larger domain are usually exceedingly demanding. Thus, while (16)-(17) constitute rigorous solutions of the eigenvalue problem, their use renders the application of harbor wave models problematic in practice. (One consequence of the above is that many of the models in this category cannot correctly simulate fairly simple phenomena like waves approaching a sloping beach. Investing confidence in model results when applied to field situations is therefore difficult.)

An effective alternative, followed in CGWAVE, is to use a “parabolic approximation” to describe ϕ_s :

$$\frac{\partial \theta_s}{\partial_r} = -p\phi_s - q\frac{\partial^2 \phi_s}{\partial \theta^2} \quad (18)$$

where

$$p = ik_0 + \frac{1}{2r} - \frac{i}{8k_0r^2}, \quad \frac{d}{dx}(CC_g\frac{d\psi}{dx}) + kCC_g(k\cos^2\theta + iW)\psi = 0 \quad (18a)$$

where and θ represent the polar coordinates of a point on the open boundary. (p and q are not unique, and alternative forms, each obtained with an appropriate rationale, have been investigated by Givoli (1991), Xu et al. (1996), Panchang et al. (2000)). The parabolic approximation (18) allows the scattered waves to exit only through a limited aperture around the radial direction. Unlike (17), therefore, it does not rigorously satisfy the Sommerfeld radiation condition. However, using this formulation decouples ϕ_s from the other components. These components (ϕ_i and ϕ_r) may be obtained by making a compromise between a detailed exterior bathymetric representation (which as noted earlier, is difficult) and the constant depth representation (which is unrealistic). A one-dimensional representation, where the depths vary in the cross-shore direction only (Fig. 1), may be selected. This is reasonable, since in general, this is often the direction in which the depths vary the most. If natural variations do not permit the representation of the exterior depths by only one section, a second one-dimensional section, shown as transect 2 in Fig. 1, may be constructed. For transects 1 and 2 with varying depths, no simple analytical expression (such as (16)) can be found for the reflected wave (since ϕ_i and ϕ_r are coupled). However, the quantity:

$$\phi_0 = \phi_i + \phi_r \quad (19)$$

may be obtained by the solution of the one-dimensional version of (5), since the depths along these transects vary in one direction only. This one-dimensional equation is (Schaffer and Jonsson, 1992; Panchang et al. 2000):

$$\frac{d}{dx}(CC_g\frac{d\psi}{dx}) + kCC_g(k\cos^2\theta + iW)\psi = 0 \quad (20)$$

where, for one-dimensional geometry,

$$\phi_0 = \psi(x)\exp(iky\sin\theta) \quad (21)$$

(20) is an elliptic ordinary differential equation requiring two boundary conditions. It may easily be solved via a simple finite-difference scheme. (For the present, the dissipation factor W is considered to be prespecified). Assuming that transect 1 extends out to a region of constant depth (or deep water), a condition at P_1 may be obtained by combining a specified incident wave

$$\phi_i(P_1) = A_i\exp(ikx\cos\theta_i + iky\sin\theta_i) \quad (22)$$

(where A_i = a given input wave amplitude) and an unknown reflected wave:

$$\phi_r(P_1) = B\exp(-ikx\cos\theta_i + iky\sin\theta_i) \quad (23)$$

Without loss of generality, the point P_1 may be located at x = 0, which allows elimination of B to yield

$$\frac{\partial\psi}{\partial x} = ik\cos\theta_i(2A_i - \psi) \quad (24)$$

At the coastal boundary point Q_1, the partial reflection boundary condition (9) may be used in the following form:

$$\frac{\partial\psi}{\partial x} = \frac{i\sqrt{k^2 - k^2\sin^2\theta}(1-K_r)}{1+Kr}\psi \quad (25)$$

where K_r is the reflection coefficient for the exterior coastline (i.e. near Q_1) and $k\sin\theta$ is constant for the one-dimensional problem.

The solution of (20) using boundary conditions (24) and (25) along with (21) produces ϕ_0 along transects 1 and 2. These solutions are denoted by ϕ_{01} and ϕ_{02}. The desired ϕ_0 along the semicircle may be obtained by laterally translating φ_{01} and φ_{02} via interpolation between transects 1 and 2 as follows:

$$\phi_0 = (1-m)\phi_{01}\exp(-ik(r-y)\sin\theta) + m\phi_{02}\exp(ik(r+y)\sin\theta) \quad (26)$$

where we have set $y = 0$ at the center of semicircle, the interpolation function $m = (r - y)/2r$, r is the radius of the semicircle, y is the lateral coordinate of the open boundary node relative to the origin of semicircle (Fig.3).

The boundary condition for φ along the semicircle Γ may be obtained by using the continuity of the potential (equations 14 and 19) and its derivative along with (18) and (26):

$$\frac{\partial \psi}{\partial x} = \frac{i\sqrt{k^2 - k^2 \sin^2 \theta}(1 - K_r)}{1 + K_r}\psi \quad (27)$$

Thus, the solution of (20) provides along the 1-dimensional transects. These values can be translated laterally and substituted into (27) to obtain the open boundary condition for the two-dimensional equation (1). Zhao et al. (2000) and Panchang et al. (2000) have demonstrated that this procedure provides extremely satisfactory solutions for a large number of test cases.

In CGWAVE, if non-linear breaking is implemented, a version of (21) is solved by iteration since W depends on the (unknown) amplitude.

Related Topics

- CGWAVE
- Governing Equations
- Numerical Solution

CGWAVE Math Details: Numerical Solution

Equation (5) is generally solved using the boundary element method, the finite-difference method, or the finite element method. In general, finite-difference discretizations are not well-suited to represent the complex domain shapes described, for example, in Fig. 1. Not only are the boundaries distorted, but the number of uniformly spaced grids may also be excessively large. (Adequate resolution, typically 10 points per wavelength, demands that the spacing be determined from the smallest wavelength.) Most studies with the finite-difference method have been limited to largely rectangular domains (e.g. Li 1994a, 1994b; Panchang et al. 1991; Li and Anatasiou 1992). Boundary element models can handle arbitrary shapes and require minimal storage since only the boundaries are discretized; however, they are limited to subdomains with constant depths only (e.g. Isaacson and Qu 1990; Lee and Raichlen 1972; Lennon et al. 1982). Finite element models, on the other hand, allow the construction of grids with variable sizes (based on the local wavelength) and give a good reproduction of the boundary shapes. Most finite element models (e.g. Tsay and Liu 1983; Tsay et al. 1989; Kostense et al. 1988; Demirbilek and Panchang 1998; Panchang et al. 2000) have used triangular elements, and modern graphical grid generating software permits efficient and accurate representation of harbors with complex shapes. The Surface Water Modeling System can be used to conveniently generate as many as 1,000,000 elements of varying size, based on the desired (user-specified) resolution, and to specify the desired reflection coefficients on various segments of the closed boundary. The solution of (1) by the finite element method is described in detail by Mei (1983) and by Demirbilek and Panchang (1998) when different types of open boundary conditions are used.

Whether one uses finite differences or finite elements for discretization, the numerical treatment of (1) with appropriately chosen boundary conditions leads to system of linear equations:

$$[A][\phi] = [B] \quad (28)$$

where $[\phi]$ represents the vector of all the unknown potentials. For solving (5), a similar system results as long as W is prespecified. The matrix $[A]$ is usually extremely large. In earlier models (e.g. Tsay and Liu 1983; Tsay et al. 1989; Chen, 1990; Chen and Houston, 1987) the solution of (28) was accomplished by Gaussian Elimination, which requires enormous memory and is prohibitive when the number of wavelengths in the domain is large (i.e. short waves or a large domain). Pos and Kilner (1987) were able to alleviate this difficulty somewhat by using the frontal solution method of Irons (1970).

In recent years, the solution of (28) has been obtained with minimal storage requirements for $[A]$. This is due to the development by Panchang et al. (1991) and Li (1994a) of iterative techniques especially suited for (1). These techniques, based on the conjugate gradient method, guarantee convergence and have been found to be extremely robust in a wide variety of applications involving both finite differences and finite elements for several kinds of boundary conditions. For a review of other methods, see Panchang and Demirbilek (2000). Options based on the work of both papers, viz. Panchang et al. (1991) and Li (1994a), are available in CGWAVE. It is found that the latter often leads to faster convergence, but in an oscillating fashion. The former leads to a monotonically decreasing error which can be more reassuring while the iterations are in progress.

Related Topics

- CGWAVE
- Governing Equations
- Boundary Conditions

CGWAVE Model Control

The CGWAVE model requires several user-specified parameters to control the analysis. The **Model Control** command from the *CGWAVE* menu opens the *Model Control* dialog. This dialog contains parameters that control the execution of CGWAVE. The parameter description for each field is displayed in SMS using the interactive help messages.

Controls include:

- *Title*

  Enter the simulation name. This will be used for naming the generated datasets during the model run.

- *Incident Wave Conditions*

  The *Incident Wave Conditions* section specifies wave conditions to be simulated in a run. See CGWAVE Incident Wave Conditions for more information.

- *Open Boundary*

  The *Open Boundary* section specifise the open boundary type. This should match the type selected when generating the domain in the Map Module. If the mesh has been generated using other methods, the domain shape must match the *Open Boundary* type.

  - *Semicircular*
  - *Circular*
  - *Boundary condition:*

- *1-D Options*

  The 1-D section specifies the number of nodes and the nodal spacing for the one-dimensional data that is used to distribute incident wave data to the open boundary. The file format is described in the CGWAVE user manual [110] . The **Compute 1-d Length** button calculates the 1-d Spacing variable such that the specified number of nodes will extend to the limits of the bathymetry data. This bathymetry data is defined by a scattered dataset. Therefore, a scatter dataset must exist to in order for this button to be active. If no scatter data exists, generate an appropriate file containing the one-dimensional depth values.

  - *Current 1D Wave Line Settings*
    - *Spacing:*
    - *Length:*
    - *Number of nodes:*
  - *1D Domain Extension Parameters*
    - *Length to edge of scatter:*

        - *Radius of domain:*
        - *Ideal 1D spacing:*
        - *1D spacing*
        - *Num. of Nodes*
        - *Minimum 1D Depth*
        - *Update Num. of Nodes*
        - *Update Spacing*
        - *Extract Depths*
- *Solver Options*

  CGWAVE support two separate numerical solvers for robustness. Select which solver to use via the *Solver* radio group. The *Standard solver* should be utilized first. If CGWAVE fails to converge, the *Modified* method can be utilized.

    - *Solver:*
    - *Convergence tolerance:*
    - *Output echo frequency:*
    - *Maximum iterations:*
    - *Num. of bessel terms used:*
- *Iteration Control*

  CGWAVE allows specifying the maximum number of iteration to be performed by the numerical solver. This number is specified in the *Maximum Iterations* edit field. The solver will terminate after performing the maximum number of iterations or when the change in the solution is less than the convergence tolerance specified in the *Convergence Tolerance* edit field. It is recommended that the convergence value be between 1.0e-6 and 1.0e-9. CGWAVE will print a progress report on the tolerance at the user specified report interval entered in the *Convergence Interval* edit field.

- *General Parameters*

  CGWAVE includes options to model bottom friction, wave breaking and nonlinear dispersion. These options may be enabled using the appropriate toggle box. See the CGWAVE user manual for more information about these parameters.

    - *Bottom friction*
    - *Wave breaker*
        - *Breaking coefficient*
    - *Nonlinear dispersion relation*
        - *Convergence tolerance*
        - *Maximum iterations*

Related Topics

- CGWAVE Graphical Interface

CGWAVE Incident Wave Conditions

The *Incident Wave Conditions* section of the *CGWAVE Model Control* dialog allows specifying wave conditions to be simulated in a run. In simplistic terms, these are the boundary conditions for the simulation.

Each condition (term) is defined as a wave direction, period and amplitude. The wave direction is specified in degrees measured from the positive x-direction in a counter clockwise direction. This means that waves propagating from West to East have a direction of 0°, and waves propagating from South to North have a direction of 90°. The amplitude is specified in meters, and the period in seconds. These values are used as boundary conditions at a location *out to sea* and are propagated to the open ocean boundaries on the domain using one-dimensional wave propagation. A set of values (one direction, frequency and period) can be selected by clicking in the window. The selected set can be changed via the edit boxes above the window. The buttons below the window can be used as follows:

New

The **New** button creates a new condition or wave term below the selected term. The values are defaulted to zero.

Copy

The **Copy** button copies the selected condition or waver term as a new set in the list. The new set is placed directly below the selected set. (Note: it does not make sense to include the same term twice since the two terms are equal to a wave of double the amplitude in the specified direction at the specified frequency. This option is provided for convenience in generating terms with common paramters.)

Delete

The **Delete** button deletes the selected condition or wave term. The rest of the terms are unaffected.

Import

The **Import** button allows selecting a file containing Incident Wave Conditions data. SMS prompts to replace or append to existing data. See the CGWAVE documentation for the Incident Wave Conditions file format.

Clear

The **Clear** button deletes all of the Incident Wave Conditions data.

Wave Condition Generation

The SMS interface for CGWAVE allows generation of multiple wave conditions from a directional spectrum or to represent long wave or infra-gravity wave conditions. These are accessed using the buttons below the incident wave conditions.

- **Generate From Spectrum** – Wave conditions that make up a spectral wave condition can be added by selecting the spectrum to be simulated. A spectrum can be generated or read in using the **Spectral Energy** command CGWAVE menu .
- **Long Wave Input Toolbox** – The long wave toolbox generates infra-gravity wave terms using a 1 dimensional Boussinesq model run and user specified parameters.

Long Wave Input Toolbox

Infragravity (IG) waves are surface gravity waves with frequencies lower than the wind waves (hence the reference to them as *long* waves). IG waves consist, among others, of long-period oceanic waves generated along continental coastlines by nonlinear wave interactions of storm-forced shoreward-propagating ocean swells. They differ from normal oceanic gravity waves which are created by wind acting on the surface of the sea.

While normal wind waves typically have periods in the order of 20 seconds (or less), these waves can interact with coastlines which filter out the energy and covert some of the energy to subharmonic frequencies (ranging from 50 - 350 seconds). These are the IG waves. IG waves could also refer to phenomena such as tides and oceanic Rossby waves.

Motivation

When modeling harbors, it is possible energy in the IG range could result in resonance in the harbor. Therefore, it may be useful to simulate these terms in a CGWAVE analysis of a harbor.

IG waves may also influence navigation, coastal inlets, and coastal structural design projects.

The Toolbox

To define wave conditions consistent with infra-gravity waves, the interface in SMS includes the Infra-Gravity Wave-input Toolbox (IGWT). This tool utilizes a one dimensional Boussinesq model to compute wave conditions for input to CGWAVE.

Input

The IGWT requests for the following inputs:

- An input wave spectrum (from file or from SMS) – This defines the wave conditions at a deep water point.
  - When coming from a file, this refers to a frequency spectrum or **.spf* file . These are directly input to the one dimensional Boussinesq model. The file format is defined by the developer of BOUSS1D and BOUSS2D.
  - When coming from a spectrum, the user chooses a spectra that has been loaded into or created by the SMS spectra generator. SMS creates a series of frequencies and energy densities from this spectrum that are fed into the one dimensional Boussinesq model.
- Offshore water depth – This parameter refers to the depth in meters at the deepwater buoy (or site). If this is deeper than the Boussinesq limit, the depth will be set to the limit.
- Nearshore water depth – This refers to the depth in meters at the predominant breaking location. This value can be approximated as twice the incident significant wave height.
- Minimum/maximum long wave period – This refers to the cutoff periods for the IG wave spectrum. Typical values are from 15 to 600 sec.
- Number of components – using this parameter the user specifies how many wave components will be generated **in each** specified direction for CGWAVE. This term is referred to as 'N' in the discussion below.
- Maximum oblique angle – The one dimensional Boussineesq model ignores wave direction. In fact, the directional bins from the input spectrum are ignored as they are converted to a frequency spectrum. This value defines a total variation (in degrees) for the directions to be considered. They are centered around the shore normal direction of the CGWAVE mesh.
- Number of angles – This value should be a positive integer. The toolbox will create N different wave components for each of this number of directions. If this number is 1, all the components will be generated in a shore normal direction. If this number is 2, then two sets of components will be generated, each half the oblique angle either side of the shore normal direction.

Process

The IGWT is actually setting up and running a one dimensional Boussinesq model as well as a pair of utilities to compute wave components to be applied on the CGWAVE mesh. Since this process is not always successful, it can be useful for the user to understand the process and in some cases, work through the process manually. As the tool is exercised and modified (feedback is welcome), the need for manual operation should be reduced, but understanding the process is still valuable. The three executables used in this process are distributed with SMS and can be found in the models area of the installation in the BOUSS2D folder. Each can be run in a DOS prompt in interactive mode if needed.

The 1D Boussinesq Run

The IGWT uses the BOUSS-1D model executable *bouss1d.exe* which is a one dimensional version of the Boussinesq equations. SMS creates a one dimensional profile starting at the specified **deep** water condition, maintaining that depth for five wavelengths and then transitioning up a gentle slope (1:50) to a depth of zero (0.0). The IGWT uses a grid spacing of 1/20 of the wavelength computed from the provided frequency spectrum.

The one dimensional run creates a number of output files at the specified **near shore** depth. These including significant wave height, mean water level, runup and a time series of water surface. The IGWT will use the water surface file (**_ts_eta.001*) as input to the spectral analysis utility in the next step.

Spectral Analysis

The IGWT uses the utility *spec_anal.exe* to process the water surface time series at the near shore depth and create a spectrum. This analysis can also be run manually at a DOS prompt after the one dimensional Boussinesq run generates the time series file. The utility will prompt the user for the name of an output file (qspec.001 by default). This output file in used as input in the next utility to generate wave components.

Wave Component Extraction

The IGWT uses the utility *export_v1.exe* to process the frequency spectrum at the near shore location and extract wave components at regular freqency spacing from the minimum to the maximum defined range. The utility creates a text file (out.txt) that lists these components.

Once the three processes have been run, the IGWT reads the out.txt file and distributes the wave components over the specified directions (centered around shore normal).

Output

The IGWT creates a series of wave components for each of the directions specified in the input. If the number of angles is 0, then there will be N components distributed through the frequency range. In the number of angles is 1, the toolbox will generate 3*N components (one direction offset from shore normal in each direction.

It is possible that zero energy components may be added. The user should verify the generated components.

Approach

The IGWT utilizes a one-dimensional version of the BOUSS-2D model to transform wave spectrum from the "deep-water" limit of the Boussinesq model (H < L/2). A constant 1:50 slope is assumed between the offshore and nearshore water depths. If complex offshore topography exists, use BOUSS-2D to bring the waves to the nearshore.

External Links:

- Infra-Gravity Wave Input Toolbox (IGWT): User's Guide (May 2007) PDF

CGWAVE Boundary Conditions Dialog

In the CGWAVE model, a wave direction, amplitude and frequency must be specified at open boundaries and reflection coefficients are defined for all closed boundaries. In addition to these exterior boundaries, the model also includes the capability to simulate interior islands, and floating barriers.

Assign Boundary Condition

The *CGWAVE Boundary Conditions* dialog is used to assign boundary conditions to individual nodestrings. This dialog is invoked with the **Assign BC** command in the *CGWAVE* menu . Before assigning boundary conditions to nodestrings, at least one nodestring must be selected using the **Select Nodestring** tool. To assign boundary conditions to the selected nodestring(s), select one of the boundary condition options.

Boundary Types

- **None**
- **Open Ocean** – Delineate the region where waves will enter the domain. The attributes of the waves that enter the domain are defined as incident wave characteristics in the *CGWAVE Model Control* dialog . These values are propagated from an offshore location to the open ocean boundaries.
- **Coastline** – Represent a region where the wave is obstructed. At these locations, the Reflection coefficient should be set. The coastline reflection term defines to what degree a section of coastline reflects incoming waves. Legal values vary from 0.0 for a gradual sandy incline to 1.0 for solid vertical rock wall. This boundary condition may be assigned to either exterior regions of the domain, or interior holes which represent islands.
    - **Reflection coefficient**

- **Floating Barrier**

Related Topics

- CGWAVE
- CGWAVE Graphical Interface
- CGWAVE Boundary Conditions
- Feature Objects Menu

CGWAVE Practical Notes

Domain Shape: To the extent possible, it is necessary to have the coastlines outside the semicircle be as straight as possible, and ideally, parallel to one of the axis (as shown in Fig 1). The semicircle should be so located that the depths outside vary, to the extent possible, in a 1-d (cross-shore) direction only in the exterior. These depths are introduced in the model through the two 1-d sections. It is necessary to specify a reflection coefficient for the exterior coastline.

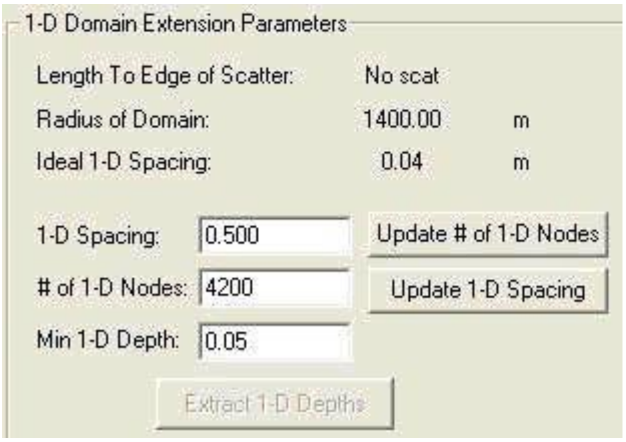
Of course, these conditions are not encountered in practice in an ideal manner, but it is important to understand that the model works under these assumptions. V-shaped coastlines do not meet these criteria. Also, not all real domains have 1-dimensionally varying exterior depths. Regions with 2-d depth variations should preferably be incorporated inside the semicircle and it may be necessary to enlarge the domain so that solutions in the area of interest (hopefully) are not affected by the coastline shape. It is impossible to accommodate all real topographic variations of the model's exterior domain. Since a part of what happens outside the semicircle influences the solution within (i.e. how the backscattered waves behave), some simplified representation of the exterior is needed. The exterior representation used in CGWAVE is more sophisticated than that used in earlier models (which assumed collinear and fully reflecting coastlines and constant exterior depths).

The size of the domain is also governed by the wavelength. Typically, the minimum radius for the semicircle should be about 2 or 3 times the wavelength, which also dictates the overall number of grids. It is important that the modeler first estimate the number of grid points needed, at least in an approximate sense. This can be done by estimating a nominal wavelength L (based on the period and depth), and using L/10 as a rough measure of the grid size. Obviously, large domains with short period waves will lead to hundreds of thousands of nodes. This is not a problem physically, but can be very time consuming to simulate. For spectral conditions, the domain size should be dictated by the longer wavelengths L_1 (say radius = $3L_1$), but the resolution for the shorter wavelengths (L_2) would be, say $L_2/10$, which may be excessively fine for the longer wavelengths. Obviously, some care (and perhaps compromise) is needed in designing the grid and also selecting the input spectral components. If some components have little energy, it may be best to delete them from the modeling.

The input wave condition (amplitude, period, direction) must be specified at the end of the 1-d cross-section. Simulations are less reliable (not necessarily wrong) if the waves approach the coast at a glancing angle (say within 20° of the coastline). The 1-d sections MUST reach out further than the outermost point of the semicircle. The domain should have no zero or negative depths; the land-water interface must be demarcated using a small water depth (say 0.5 m, but the limit depends on the overall water depths and wave periods and the problem being solved). Note that making this too shallow may result in a large number of grids in this area which may not be necessary for all applications).

For basic specifications, the incident wave angle specification is as follows. Waves going in the +x axis (i.e. to East) are specified as incident angle =0°. Waves going in +y direction (i.e. to North) are specified as incident angle =90°, to West =180°, to South = 270° . Incident wave direction is used in degrees. Convergence tolerance is typically assigned 10^{-8}, output frequency is 100 every iterations.

For the boundary condition specification, the “parabolic plus two 1d sections” option (Option 3 in *.cgi) should be chosen for most harbor problems, with a semicircle open boundary; in earlier versions, it is possible to use the option with just one 1d section (Option 2 in *.cgi) but it really provides no benefit over Option 3. The other options consist of the “parabolic approximation” (Option 1 in *.cgi) which is suitable if the exterior region is of constant depth. The Fourier Bessel Series (Option 0 in *.cgi) must be used if its limitations, described above in Section 2.2 (equations 15-17), are met; typically it is sufficient to specify 15 terms in the infinite series. For harbor problems, SMS will create the 1d sections based on depths based on depths on semicircle, and the desired grid spacing (used to solve the 1d equation (20)), based on (approximately) L/10 must be specified in the dialog box:

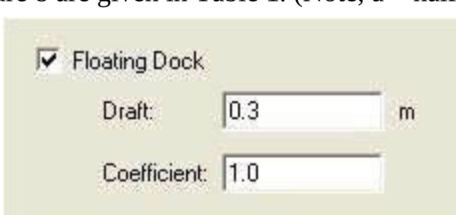


A full circle option is available for open sea problems where the exterior is of constant depth; the theoretical ideal for this is the Fourier Bessel Series (which works for unit wave input only); the parabolic mode may also be used (Panchang et al. 2000) with little loss of accuracy.

It is useful to perform initial sample runs, with no breaking and with fully absorbing coastlines, for a normally incident and obliquely incident wave input. The resulting solutions, especially the phase diagrams, can be used to perform visual quality control, since forward propagating waves are the easiest for making intuitive assessments (e.g. spacing between crests should be smaller and wave heights generally increasing in shallower water; bending of phase-lines for oblique incidence to approach the shore in a normal fashion, high wave heights in very shallow water because breaking was not applied, etc). Repeating the above runs with full reflection should lead to some standing wave patterns (large and small wave heights; rapid change of phase). A visual examination of the results will enhance confidence in later simulations.

There are two methods for applying wave breaking. For practical problems, we recommend making a non-breaking simulation and then applying an H/d limit (generally between 0.6 and 0.85) to alter the calculated wave heights to the limit (say 0.78d). The other, more rigorous approach is to run the model in the nonlinear breaking mode where breaking properties are continuously recalculated based on prior solutions. This requires several rounds of iterations and is recommended for special problems and/or research applications. For nonlinear runs, the maximum number of nonlinear iterations is generally specified as 15 with a tolerance of 10^{-6}. For spectral simulations, obviously the former approach is recommended.

For incorporating docks, the actual “without-dock” water depth is originally specified while generating the grid; areas covering the docks must then be highlighted, and a draft and coefficient, selected from Fig. 2 (Li et al. 2005) or Table 1, must then be specified for these areas. The coefficient = α = Aln(ka)+B. For kh < 0.1 and kh > 4, we recommend using the A and B values corresponding to these thresholds. For convenience, the numerical values corresponding to Figure 8 are given in Table 1. (Note, a = half width of the structure, relative submergence= draft/water-depth = d/h).



For shallow water, α is roughly equal to unity for shallow draft, and α ~ 0.7 for deep draft. For intermediate and deep water, α is not constant but shows an increasing trend with ka. In very deep water, the mismatch is large. Note that for short waves (relative to submergence), T tends to 0. This requires us to create a high level of wave blockage, which can be accomplished by α ~ 0 (Li et al. 2005). Of course this result would not hold if d/h were much smaller than the smallest value investigated by Li et al. (2005), which is 0.25 (typical in engineering practice).

It is worthwhile to refine the grid in these areas since the model uses the under-keel clearance multiplied by the coefficient as the depth for calculation. This modified depth is smaller than the original depth (and hence the resolution should be finer).

Finally, it is noted an extensive list of test-cases are provided in the appendix. CGWAVE has been validated against these tests (which represents possibly the most rigorous testing for wave models). The model results are compared to lab data or analytical model results. The input and output files are provided. It is highly encouraged to perform these simulations, alter parameters, etc. so that an examination of the results may help show what can be expected. At a minimum, we recommend a visual inspection of the results provided. Often real-life problems have complex solutions which are difficult to explain or even anticipate. The fact that the model reproduces the correct result in so many cases may enhance confidence in the results, assuming the modeling was performed with due diligence. The test-cases can also be used as a teaching tool.

Related Topics

- CGWAVE

CGWAVE Test Cases

The following test cases are available with grids, input, and output files for review.

Test cases are available here with grids, input, and output files for review and instruction. An extensive list of test-cases are also provided by the Coastal and Hydraulics Laboratory [111] . CGWAVE has been validated against these tests (which represents possibly the most rigorous testing for wave models). The model results are compared to lab data or analytical model results. The input and output files are provided. It is highly encouraged to perform these simulations, alter parameters, etc., so that an examination of the results may help show what can be expected. At a minimum, a visual inspection of the results provided is recommended. Often real life problems have complex solutions which are difficult to explain or even anticipate. The fact that the model reproduces the correct result in so many cases may enhance confidence in the results, assuming the modeling was performed with due diligence. The test cases can also be used as a teaching tool.

The following tutorials may also be helpful for learning to use CGWAVE in SMS:

- General Section
  - Data Visualization
  - Mesh Editing
  - Observation
- Models Section
  - CGWAVE

Tests 1 & 2

These tests involve monochromatic wave propagation over the shoal-slope bathymetry of Berkhoff et al. (1982). Grid has 15 points per wavelength. Parabolic approximation open boundary condition. Test 1 – input amplitudelitude = 1 meter (linear). Test 2 – input amp = 0.0232 meter (nonlinear). Resulting amplification factors along Transect 5 are shown – they match results and data in Demirbilek and Panchang (1998). Wave direction and phases diagram shows largely progressive waves except near the shoal where the waves become multidirectional.

All runs involve no breaking.

Tests 3 & 4

Wave propagation over flat bottom and a shoal, after Vincent and Briggs (1989, JWPCOE). Monochromatic (T = 1.3 s) and broad-directional spectral (BI) input based on Panchang et al. (1990, JWPCOE). For spectral simulation, input consists of 29 directional components in the ±60° bandwidth and 5 frequency components. All runs involve no breaking. Results match numerical and experimental data described in Demirbilek and Panchang (1998), Vincent and Briggs (1989), and Panchang et al. (1990)

Test 5

Wave propagation (T=5.05 s, d = 0.25 meter) over a flat bottom surrounded by infinite ocean. **Depth = 0.25 meter** . Test 5 – using Bessel-Fourier boundary conditions (this is the most accurate boundary condition for the problem as specified although the exterior conditions are unrealistic in practice). See Xu et al. (1996, JWPCOE) for details. Note the Bessel-Fourier boundary condition works only for input amplitude = 1 meter. For other input amplitudes, solution should be appropriately scaled.

Test 6

As in test 5, but with friction which seems to become effective for long waves. Test 6 – the circular domain is assigned f = 0.5 everywhere, waves propagate in from the right. Friction leads to smaller wave heights.

Test 7

As in Test 5, but only the central area has a non-zero friction.

Test 8

Waves propagating towards a coastline on a flat seabed, parabolic and one-dimensional open boundary condition. No breaking or friction used.

Test 9

As in Test 8, but with parabolic boundary condition only. This is used to demonstrate correctness of this boundary condition since solution for constant exterior depth is known.

Test 10

As in Test 8, but friction f = 0.5 for the whole domain. Note wave height input = 2 m. at the end of one-dimensional section which extends beyond the semicircle. Wave heights decrease in shoreward direction due to friction. No breaking used.

Test 11

As in Test 8, but with f = 0.5 in a central square region (can be seen on .cgi file). f = 0 elsewhere. No breaking.

Test 12

As in Test 10, but with central square area indicated as a “floating dock”.

Test 13 - Circular Island/Shoal

Long wave propagation past the circular island/shoal combination of Homma (1950). Bessel-Fourier open boundary condition. Input T = 240 sec. Results match analytical solution given in Demirbilek & Panchang (1998). No breaking.

Test 14

As in Test 5, but with a circular pile in the domain. Input T = 10 s and constant depth = 15.03 meters. Results match analytical solution (see Panchang et al. 2000, ASCE JWPCOE).

Test 15

As in 14, but the pile is off-center. Parabolic open boundary condition. Results match analytical solution (Panchang et al. 2000, JWPCOE).

Test 16

Long wave (T = 260 s) propagation up a sloping beach. Parabolic and one-dimensional boundary condition. Solution is completely one-dimensional. Results match those in Panchang et al. (2000).

Test 17

Oblique wave incidence on uniformly sloping beach. Results match analytical solution of Radder (1979) given in Panchang et al. (2000, JWPCOE).

Test 18

Propagation of obliquely incident waves (incidence angle =20°) past a shore-perpendicular thin fully-reflecting breakwater on a sloping beach (beach is fully absorbing). Parabolic and one-dimensional open boundary condition. Results match analytical results given in Kirby (1986) and Panchang et al. (2000).

Tests 19 & 20

As in Test 18, but with nonlinear breaking on and off. Test pertains parameters in Zhao et al., (2000, Coastal Engineering).

Test 21

As in Test 19, but with shore-parallel breakwater. Parameters and results as in Zhao et al. (2000). Results are for no breaking.

Test 22

As in Test 19, but with shore-parallel breakwater. Parameters and results as in Zhao et al. (2000). Results are for nonlinear breaking.

Test 23

Wave propagation/resonance in a rectangular harbor. Results match analytical solution plotted in Demirbilek & Panchang (1998). With friction f = 0.12, the resonant peak amplification reduces substantially for kl =1.4 (T = 1.0447 s).

Test 24

Wave propagation around a floating square platform in circular domain. While developing the 2-d grid, the area covering the dock is also filled with finite elements; each node is assigned a depth equal to the local under-keel clearance times the correction factor a. The parameters in the simulations are 2a = 2m, h = 1m, d/h = 0.5 and ka = 2 (corresponding to the cases described by Tsay and Liu (1983). So the depth used for calculation = α.d = α.(0.5) = 0.04. Correction factors such as a = 0.08 are given in Li et al. (2005, Canadian J. of Civil Engr). Results are similar to 3d results given in Tsay & Liu (1983).

Test 25

Wave propagation around a circular shoal in a circular domain. This is intended to show the effects of the “steep slope” terms. The test is based on Fig. 7 in Chandrasekhara and Cheung (1997, JWPCOE).

Test 26

Radiation Stress calculations. Waves propagating towards a coastline on a flat seabed, parabolic and one-dimensional open boundary condition. No breaking or friction used. Five folders are given. Three separate monochromatic cases (260 degrees, 270 degrees, and 280 degrees, i.e. normal incidence and 10 degrees off-center incidence) each of amplitude 0.5 m and period T = 1 s, for fully absorbing coastline. For 270 degrees, results for fully reflecting coastline are also given. Results match theoretical solution (eq. 6-9 and eq. 54 in Copeland, 1985, Coastal Engg).

For spectral tests, the same 3 waves were added to form the input spectrum. Radiation stresses for the spectrum are an integration of individual components (eq. 1 in Fedderson 2004, Coastal Engg). The spectral results can be used also to check the mean wave direction (which should be 270 degrees) and the mean frequency (= 6.28 radians/s).

Test 27

Wave propagates (incidence angle =0°) over a rectangular friction region in a constant-depth domain. With friction f = 2.122, input T = 20s, H = 6.1m and constant depth = 15.2m. Results match solution obtained by Dalrymple et al. (1984).

Test 28

Obliquely incident wave propagates in a rectangular channel with the fully-reflecting side walls. Input T = 12s and constant depth = 8m. Results match analytical solution plotted in Dalrymple and Martin (2000, JWPCOE).

Related Topics

- CGWAVE

5.5. Coastal Modeling System

CMS

The Coastal Modeling System

The Coastal Modeling System (CMS) has been a research and development area of The Coastal Inlets Research Program (CIRP) at the United States Army Corps of Engineers – Engineering Research and Development Center (USACE-ERDC), Coastal and Hydraulics Laboratory (CHL) since 2006. It was build from a group of numerical models that have been under development since 2002. Information on the CIRP and publication on the CMS can be found at [112]

The system is a coordinated system of major multidimensional numerical models integrated to simulate waves, currents, water level, sediment transport, and morphology change in the coastal zone. Emphasis is on navigation channel performance and sediment exchanges between the inlet and adjacent beaches in the coastal zone. The CMS has been verified with field and laboratory data.

System Components

- CMS-Flow
- CMS-Wave

Steering

In order to combine the capabilities of the two main numeric engines of Flow and Waves, information must be passed from one engine to the other. In the case of CMS-Flow, this means reading in wave data from CMS-Wave. Information passed to CMS-Flow includes reading radiation stress gradients that directly impact currents (wave driven currents) and height fields, wave directions and breaking data which enter into the sediment transport rate formulations. In the case of CMS-Wave, the option exists to read in currents and simulate their transformation by the current. For either situation, the data fields must be interpolated onto the native domain (interpolate wave data onto the flow grid and/or flow data onto the wave grid).

This may be done interactively using the tools in SMS. However, it is much more efficient to read or define simulations for both engines, and invoke the steering module from the Data menu. This tool runs the engines separately, but interpolates the output and passes it to the other engine automatically.

External Links:

- CIRP Wiki – online help database for CIRP information/publications and CMS modeling software [113]
- US Army Engineer Research and Development Center – Ongoing Research [114]
- Presentations
  - Coastal Modeling System (CMS) for Integrated Calculation of Waves, Flow, Sediment Transport, and Morphology Change [115]
  - Introduction to CMS-Wave [116]
  - Additional information on CMS-Flow capabilities [117]
  - CMS-Wave Demonstrations to Louisiana (Levees & Muddy coast) [118]
  - Future of the CMS [119]

CMS-Flow/CMS-Wave Steering

SMS version 12.1 and later

The Inline Steering Model Control option allows CMS-FLOW ↔ CMS-WAVE steering with data stored in memory. The steering is controlled by the CMS-Flow model executable and not by SMS.

In SMS version 12.1 and later the Inline steering option is reached by going to the CMS-Flow **Model Control** (by right-clicking on the simulation in the Project Explorer and selecting **Model Control**) selecting the *Wave* tab and selecting the *Inline Steering* option under the *Wave Information* heading. Then the CMS-Wave simulation file is specified.

SMS version 12.0 and earlier

Steering CMS INLINE page – Provides the ability to perform CMS-FLOW ↔ CMS-WAVE steering with data stored in memory. The steering is controlled by the cms-flow model executable and not by SMS.

The *Steering Wizard – CMS-Flow ↔ CMS-Wave* dialog is reached by going to the *Data* menu and selecting the **Steering Module** command. From the *Steering Wizard* dialog, select *CMS INLINE* and click on **Next** . The following dialog is the *Steering Wizard – CMS-Flow ↔ CMS-Wave* dialog. This dialog has the following options:

- CMS-FLOW Source Grid option – Select the cms-flow grid to be used in the steering.
- CMS-WAVE Source Grid option – Select the cms-wave grid to be used in the steering.
- Time options – Select the time parameters from the cms-flow model control interface.
  - Total Simulation Time:
  - CMS-Flow Time Control – this button opens the *CMS-Flow Model Control* dialog.
  - Run CMS-Wave every: – this field lets the intervals in which SMS will run CMS-Wave
- CMS-FLOW → CMS-WAVE options – Disabled. These options are set in the cms-flow model executable.
  - Current Field
  - Water Surface Elevation
- CMS-WAVE → CMS-FLOW options – Disabled. These options are set in the cms-flow model executable.
  - Wave Data
- Zero extrapolation
- Extrapolate Out – selecting this option causes the *Distance* field to become active where the distance can be specified.

Instructions for CMS-Flow/CMS-Wave steering can be found on the CIRP Wiki .

Related Topics

- Steering
- CMS-Flow
- CMS-Wave

External Links

- CIRP Wiki for CMS-Flow and CMS-Wave Model Coupling

5.5.a. CMS-Flow

CMS-Flow

CMS-Flow	
Model Info	
Model type	Hydrodynamic model intended for local applications, primarily at inlets, the nearshore, and bays
Developer	Christopher W. Reed, Ph.D. Alejandro Sanchez Mitchell E. Brown
Web site	http://cirpwiki.info/wiki/CMS-Flow
Tutorials	General Section • Data Visualization • Observation Models Section • CMS - CMS-Flow

CMS-Flow is a component of the Coastal Modeling System (CMS). Until 2007, it was developed under the name M2D. At that point in time, it was revised, file formats were updated for better flexibility and expandability, and it was incorporated into the CMS suite.

The model developers at the United States Army Corps of Engineers maintain a wiki specifically for the numerical engine. It can be viewed at: cirpwiki.info/wiki/CMS-Flow . For more information on the model itself, refer to the users manual published by USACE-ERDC.

CMS-Flow is a finite-volume numerical engine which includes the capabilities to compute both hydrodynamics (water levels and current flow values under any combination of tide, wind, surge, waves and river flow) sediment transport as bedload, suspended load, and total load, and morphology change.

The interface in SMS allows setting up and editing computational grids, specifying model parameters, defining interaction of this model with the wave counterpart (CMS-Wave), launching the model, and visualizing the results.

The model is intended to be run on a project-scale, meaning the domain should only be on the order of 1-100 kilometers in length and width. The following sections describe the interface and make recommendations for application of the model.

The CMS-Flow model can be added to a paid edition of SMS.

Graphical Interface

CMS-Flow makes use of SMS’s simulation interface. The simulation interface works by creating a simulation object in the Project Explorer then adding components for the simulation. CMS-Flow makes use of the following components:

- A quadtree grid
- A CMS-Flow Boundary Conditions coverage
- A CMS-Flow Save Points coverage (optional)
- An Activity Classification coverage. (optional)

The CMS-Flow simulation has its own menu commands that can be accessed by right-clicking on the simulation object. Each of the CMS-Flow coverage may also have a right-click menu to access dialogs and functions specific to the coverage or objects in the coverage.

From the simulation menu, the CMS-Flow *Model Control* dialog can be accessed. In the *Model Control* , parameters for the simulation run can be set.

Using the Model / Practical Notes

For new simulations, create the CMS-Flow grid based on a conceptual model. The conceptual model includes:

- Grid Generation – It's recommend generate a CMS-Flow grid using the conceptual model and a Quadtree Generator Coverage. This coverage has attributes associated with a two-dimensional quadtree grid and the model parameters associated with CMS-Flow. The grid position and extents are defined in the coverage using a grid frame, which can be defined with three clicks of the mouse (recommendation is to click the lower left corner, lower right corner and then upper right corner, but the position, orientation and size can all be edited during the grid generation process. The grid's *i,j* origin, however, will always be at the lower-left corner regardless of where the clicks are done. The coverage also defines the location of land and water in the grid using one of three methods:
    - Land/Water cells defined based bathymetric values – CMS-Flow uses depths, so positive depth indicates water, negative depth indicates land. Cells with depth less than the negative value of the water surface are dry. This option requires a geometric survey that includes both the bathymetric area and the areas that could potentially be flooded. This is the most intuitive option and the preferred method if geometric data is available.
    - Land/Water interface defined by coastline arcs – This option allows defining, reading, or importing arc definitions that delineate the water area. These arcs include an orientation. To the left of the arc is land, to the right is water. If necessary, select an arc and swap its orientation. All the area inside the grid frame on the "water" side of the arc must have elevations defined either from a survey, or by specification. Cells created on the "land" side of the arc will never be included in calculations (they are permanently dry). These arcs also include an attribute defining how cells spanning this interface are to be classified. They may be forced to be water (ocean preference), forced to be land (land preference) or split based on the percentage of the cell on each side of the arc (percent preference).
    - Land/Water interface defined by polygons – This option also requires defining arcs that delineate the extents of the computational area. However, these arcs must be closed into polygons. Each polygon is specified to enclose land or water and cells are classified accordingly.
- Model Output – The numerical engine consists of several components. The base engine computes hydrodynamics. To this, sediment transport and salinity can be enabled as well. Each process produces spatially varied solutions (values for each wet cell) that SMS can display as spatial datasets . Additional observation cells can be created to view output at a higher temporal resolution.

CMS-Flow Files

See the article CMS-Flow Files .

Here are tables of some of the available input and output files for CMS-Flow.

- For more information on these files see pages 224 and 242 of the manual .

Required Input Files

Name	Description
DB3	dBASE III
_Depth.H5	Grid Depth Values XMDF
_Quadtrees.H5	Telescoping Grid
MAP	Grid and Projection Information
MATERIALS	Material Values
VTU	Visual Toolkit Unstructured Mesh

Output Files

Name	Description

CMCARDS	Coastal Modeling Card Settings
Hot_Start.H5	Hot Start Information
_Datasets.H5	Mannings Number Dataset
_Diag.H5	Diagnostic Solutions XMDF
_MP.H5	Model Parameters XMDF
_Vel.H5	Current Velocity XMDF
_WSE.H5	Water Surface Elevation XMDF
TEL	Telescoping Quadtree Mesh
TXT	Output Text

Case Studies / Sample Problems

The following tutorials may be helpful for learning to use CMS-Flow in SMS:

- Models Section
  - CMS – CMS-Flow

Related Links

- SMS Models page
- CMS-Wave

External Links

- CMS-FLOW Users Manual
- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [120]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [121]
- 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 [122]
- 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) [123] Updated FAQ Website [124]
- May 2005 ERDC/CHL CHETN-IV-63 Representation of Nonerodible (Hard) Bottom in Two-Dimensional Morphology Change Models [125]
- 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 [126]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D and STWAVE [127]

CMS-Flow Coverages

The CMS-Flow model makes use of the simulation based modeling approach. This requires defining coverages in the Map module to build the components for use in the CMS-Flow simulation .

Boundary Conditions

All numeric models require boundary condition data. In CMS-Flow, boundary conditions are defined on feature arcs in a boundary conditions coverage.

The **Create Feature Arc** can be used to click out boundary conditions for the model or arcs can be converted from other coverages or modules. Arcs can also be imported.

One the boundary condition arcs have been created, right-click on the arc with the **Select Feature Arc** tool and select the **Assign Boundary Conditions** command to bring up the *Arc Boundary Condition* dialog. This command is unique to the CMS-Flow Boundary Condition coverage and is only accessible by right-clicking on a selected arc.

Arc Boundary Conditions Dialog

This dialog has the following options for boundary condition parameters.

- *Name* – Assign a name to the boundary arc.
- *Type* – Has the following options:
  - "Unassigned" – The default option.
  - "Cross-shore"
  - "Flow rate forcing" – Specifies an inflow rate (flow in cubic meters/second at each cell). This can be used to represent a river flowing into the domain.
    - *Flow Source*
    - *Inflow direction*
    - *Conveyance coefficient*
  - "WSE forcing" – Specifies the water surface elevation as a function of time for the cellstring. Options include specifying a single curve (water level -vs- time) and all the cells will have the same water level at the specified time and extracting individual curves for each cell either from a regional tidal database (ADCIRC database) or from a regional (larger) circulation model.
    - *WSE Source*
    - *WSE offset*

Save Points

CMS-Flow includes save points which can be used to output calculations at specific locations.

Save points are created in the Save Points coverage using the **Create Feature Point** tool. When the coverage is linked to the CMS-Flow simulation data will be collected during the simulation model run.

The coverage has two unique commands. The coverage right-click menu in the Project Explorer has a **Properties** command that will bring up the *Save Points Properties* dialog. Right-clicking on a point in the graphics window and selecting the **Assign Save Points...** command bring up the *Assign Save Points* dialog.

Save Points Properties Dialog

In the *Save Points Properties* dialog the output interval can be specified for data collected at each save point. The interval options can be specified for any of the following data types:

- Hydro
- Sediment
- Salinity
- Waves

All interval options can be specified in seconds, minutes, or hours.

Assign Save Points

Each save point created in the coverage needs to be given parameters as to what type of data to collect during the simulation model run. Using the **Select Feature Point** tool, right-click on each save point and use the **Assign Save Points...** command. This will bring up the *Assign Save Points* dialog where the type of data to be gathered can be specified.

The dialog has the following options:

- *Name* – Each save point can be given a unique name. The given name will appear next to the point in the graphics window after being assigned.

- *Hydro* – Sets the save point to collect hydrologic data.
- *Sediment* – Sets the save point to collect sediment data.
- *Salinity* – Sets the save point to collect salinity data.
- *Waves* – Sets the save point to collect wave data.

Related Topics

- CMS-Flow
- CMS-Flow Simulation

CMS-Flow Simulation

CMS-Flow makes use of simulations starting in SMS 12.1 and later. Simulations are useful as multiple simulations can be used in the same project.

Simulations for CMS-Flow are created by right-clicking in the Project Explorer and selecting *New Simulation* | **CMS-Flow** . The simulation will appear in the Project Explorer under the "Simulation Data" item.

Right-clicking on the simulation gives access to the CMS-Flow menus , including access to the CMS-Flow *Model Control* dialog. Each CMS-Flow simulation can have its own parameters based on what is entered into the *Model Control* . Each simulation can share the same components or use different components or any combination of shared and different components.

CMS-Flow Components

Components for CMS-Flow usually include:

- A quadtree grid
- A CMS-Flow Boundary Conditions coverage
- A CMS-Flow Save Points coverage (optional)
- An Activity Classification coverage (optional, but recommended to speed up the model run)

A simulation cannot include multiple coverages of the same type. So if building multiple boundary condition coverages, only one of the boundary condition coverages can be included in the simulation. Coverages can be merged if needed.

Linking Components

After a simulation has been created, components may be added to the simulation. Components are usually added by clicking on the component item in the Project Explorer and dragging the item under the CMS-Flow simulation. A link is then created between the component and the simulation. If the component is updated, it is updated automatically in all simulations that are linked to it.

Components can also be added to a simulation by right-clicking on the component in the Project Explorer and selecting the simulation name in the *Link To* submenu. Simularly, components can be unlinked from a simualtion by right-clicking on the component and selecting the simulation name in the *Unlink From* submenu. The *Link To* and *Unlink From* submenus become available once a simulation has been created.

CMS-Flow Model Check

Whenever the using the **Model Check** command, and just before launching an analysis, SMS performs a check on the model . If conditions are detected that are out of the ordinary or recommended range, warnings are given with suggestions to correct the problem. If serious problems are detected, errors are reported, again with suggestions.

These checks include:

- All water boundaries have boundary conditions specified.
- Cell aspect ratio is no greater than 2.

- Max time step size is suggested according to an internal calculation made within SMS.
- Make sure model control has been set up correctly (i.e. if a tidal boundary exists, make sure that tidal constituents have been defined to force with).
- Ensure that CMS-Flow has been set up to output the specific data for any existing observation station and that observation stations exist for any specified output data. Observation is not required to run CMS-Flow so this is a reminder that something may have been intended but forgotten.
- Ensure that no invalid hard bottom specifications exist in the grid. An invalid specification may be created, for example, by setting an infeasible hard bottom scalar value in the *Edit Window* or adjusted the grid's geometry without updating the hard bottom.

The model check is not all inclusive. Just because a model passes the model check does not guarantee the model will run and produce viable solutions.

Running a CMS-Flow Simulation

After all components have been added to the simulation and the model parameters have been established, the simulation can be run. This is done by right-clicking on the simulation and choosing the **Launch CMS-Flow** command or the **Save, Export, and Launch CMS-Flow** command.

The CMS-Flow Model Wrapper should appear. If there are any errors in the model run, the model wrapper will exit early. The model run can also be exited early by clicking the **Abort** button. When the model run is completed, select **Exit** to close the model wrapper.

The *Load solution* option in the model wrapper is selected by default. This option will load the solution from the CMS-Flow model run in to SMS as soon as the model run is complete.

Related Topics

- CMS-Flow
- Simulations

CMS-Flow Menu

The CMS-Flow graphical interface includes tools to assist with creating, editing, and debugging a CMS-Flow model. The CMS-Flow interface exists as a simulation. The simulation requires components in the Quadtree module and the Map module .

CMS-Flow Menu

The interface consists of a right-click CMS-Flow simulation menu and CMS-Flow coverage righ-click menus with model specific commands and visualization tools for setting up a simulation and analyzing the datasets computed by the model.

CMS-Flow Simulation Right-Click Menu

The simulation right-click menu includes the general simulation commands **Delete** , **Duplicate** , and **Rename** along with the following model specific commands:

Model Control

Brings up the CMS-Flow *Model Control* dialog. The dialog allows viewing and editing the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation.

Model Check

The CMS-Flow Model Check performs a number of checks on the simulation to ensure a valid model.

Export CMS-Flow

Tells SMS that it is time to write out the files that CMS-Flow needs to run. Note that the project must be saved before the files may be exported. The user will not see anything happen, but SMS uses the input information from the model control, the boundary conditions, and quadtree data to create the files in preparation for CMS-Flow to work.

Launch CMS-Flow

When the model is ready and has been exported, it may be launched. Launching CMS-Flow initiates the model run for the simulation. Upon successful completion of the launch, the analysis is complete and results are ready to be read and displayed by SMS.

Save, Export, and Launch CMS-Flow

This is a combinination of the previous steps put together into one.

Boundary Condition Coverage Right-Click Menu

When clicking on the CMS-Flow boundary condition coverage in the Project Explorer, the coverage has the standard coverage menu. After creating feature arcs in the coverage, the **Select Feature Arcs** tool can be used to bring up a right-click menu. This menu has the following additional command when the CMS-Flow boundary condition coverage is active.

Assign Boundary Conditions

Brings up the CMS-Flow *Arc Buondary Conditions* dialog where boundary condition parameters can be assigned to the selected arc.

Save Points Coverage Right-Click Menu

The CMS-Flow save points coverage has additional right-click menu commands for both the coverage and for feature points created in the coverage.

Right-clicking on the save points coverage in the Project Explorer has the following additional menu command:

Properties

Brings up the *Save Points Properties* dialog where the data type that will be collected in during the model run can be specified.

Using the **Select Feature Point** tool, right-clicking on a feature point in the CMS-Flow save points coverage will show the following additional menu command:

Assign Save Points

Brings up the *Assign Save Points* dialog where the type of data the feature point will gather can be specified.

Related Topics

- CMS-Flow Simulation

CMS-Flow Model Control

The CMS-Flow *Model Control* dialog allows viewing and editing the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation. The *Model Control* dialog is access by right-clicking on the CMS-Flow simulation object and selecting the **Model Control** command.

The dialog includes several tabs which partition the parameters into related groups. The tabs and their related parameters include:

General

Specifies general flow model parameters, controlling which model options the simulation will employ. The controls include:

Time Control

This section sets the starting time, duration and hydraulic time step. The *Ramp duration* defines the length of an incremental loading portion at the beginning of the simulation. Controls include:

- *Start date/time* – The hour and day at which to begin the model. This will be the first time step listed in any solution files.
- *Simulation duration* – the length of the simulation in hours.
- *Ramp duration* – The period of time in days at which to allow the model to “build up” or “ramp up” to a legitimate solution. Aids in the convergence of the model.
- *Second order skewness correction* – Turns on or off the second order skewness correction which account for non-orthogonality in the telescoping grids.

Hot Start

These controls allow specifying a previously saved hot start file to be used as initial conditions or instruct CMS-Flow to save hot start files for future use.

- *Initial conditions file*
- *Write Hot Start output file*
- *Time to write out*
- *Automatic recurring Hot Start file*

- *Interval* for writing recurring Hot Start file

See CMS-Flow Hot Start File for more information.

Solution Scheme

CMS-Flow now has two solution schemes. These include the traditional "explicit" scheme. This method requires that flow be tracked through each element, resulting in hydrodynamic time steps in the order of one to two seconds. The new option is an "implicit" scheme which uses finite volume technology, supports much larger time steps resulting in fewer time steps (shorter run times) and is identically mass conserving. When the implicit scheme is being used, one of the following matrix solver types should be specified.

- *Solution scheme* – Can be set as "Implicit" or "Explicit".
- *Matric solver* – Can be specified when using the "Implicit" option. Options are "Gauss-Seidel", "Gauss-Seidel-SOR", "BiCGSTab", or "GMRES".

Threads

CMS-Flow can take advantage of multiple processors using Open-MP parallelization technology. This control allows specifying the maximum number of threads that the engine should occupy.

- *Number of threads* – Determines the number of threads used for parallel processing.

Flow

Specifies general hydrodynamic or flow model parameters, controlling which model options the simulation will employ. The controls include:

Hydrodynamic Time Step

This control allows setting the hydrodynamic time step for the model. If the explicit solution scheme is being used, a recommended time step will be calculated based on grid cell size and depths.

- *Hydrodynamic time step* – Sets the time step for the hydrodynamics.

- *Wetting and drying depth* – Sets the minimum depth for wet cells.
- *Wave fluxes* – Turns on and off the wave volume flux velocities.
- *Roller fluxes* – Turns on or off the contribution to the wave volume flux velocities due to the surface roller in the surf zone. Only used if surface roller model is turned on.
- *Average latitude for Coriolis* – Specifies the average latitude for the grid which is used for the Coriolis parameter.

Water Parameters

This section allows specifies various general parameters to be used by the simulation. This include:

- *Water temperature* – Used in the calculation of the water kinematic viscosity and water density if not specified.
- *Water density* – Assumed to be constant for the simulation. If not specified, calculated based on the water temperature and salinity.

Turbulence Parameters

Specifies turbulence parameters.

- *Model* – Specifies the turbulence model used: "Subgrid", "Falconer", "Parabolic", or "Mixing length".
- *Base value* – Constant contribution of eddy viscosity.
- *Current bottom coefficient* – Coefficient related to the contribution to eddy viscosity from the bottom shear.
- *Current horizontal coefficient* – Coefficient related to the contribution to eddy viscosity from horizontal velocity gradients.
- *Wave bottom coefficient* – Coefficient related to the wave bottom friction contribution to eddy viscosity.
- *Wave breaking coefficient* – Coefficient related to the wave breaking contribution to eddy viscosity.

Bottom and Wall Friction

Friction datasets can be created and specified to define a spatially variable bottom roughness to provide a resistance to flow. This value has little impact in deep ocean applications but can be important in shallow regions. This is a user editable dataset. The available dataset types are:

- *Wave-current bottom friction coefficient* – Used for quadratic com-bined wave-current mean bed shear stress calculation.
- *Coefficient*
- *Bed-slope friction coefficient* – Specifies whether to include the bed slope friction factor or not in the calculation of the bed friction.
- *Wall friction* – Turns on or off wall friction.
- *Bottom roughness dataset*

Salinity

Specifies model parameters relating to salinity modeling in CMS-Flow. The controls include:

- *Calculate salinity* – Turning on this option allows specifying the salinity options. If this is option is not used, salinity will not be included in the model run.

Time Steps

In addition to the time step specified in the *general parameters* tab for hydrodynamic calculations, the explicit model also allows the specification of longer time step for the salinity dispersion process.

- *Transport rate* – Specify this time step if salinity is enabled in this tab and the solver option is set to "explicit" in the *Flow* tab. It must be larger than the hydrodynamic time step and is generally at least a factor of 20. The default transport rate time step is 60 seconds.

Initial Condition

Salinity transport requires that each cell have an initial value for salinity concentration (measured in ppt). CMS-Flow allows this value to start as a constant for each cell, in which case, during the initial (or ramp time)of the simulation the concentrations are distributing to natural values.

- "Global concentration" – This control specifies an initial concentration for the entire domain. The value 0.0 ppt (fresh water) is a commonly used value.
- "Spatially varied" – This option allows to bring up a *Dataset* dialog where it is possible to create or select a spatially varied dataset that will be used by CMS-Flow as the initial concentration at each cell. The *Dataset* dialog has the following options:
  - **Create** – Brings up the *Dataset Toolbox* where a new dataset can be created using the *Data Calculator* .
  - **Select** – Brings up an option to select an existing dataset already in the project.
  - **Unselect** – Removes the selected or created dataset.

Wave Data

CMS-Flow includes three options on how to use wave data.

None

No wave data is imputed into the model.

Single wave condition

The effects of waves are input to the model in the form of spatially varying datasets. These datasets must already exist on the grid. Simply select which dataset to use.

- *Significant wave height*

- *Peak wave period*
- *Mean wave direction*
- *Wave breaking dissipation*
- *Wave radiation stress gradients*
- *Surface roller stress gradients*

Clicking on the **Select** button under any of the options with being up a *Dataset* dialog with two options. The **Select** button in the *Dataset* dialog will bring up a *Select Dataset* dialog where a dataset in the project can be chosen. The **Unselect** button will remove the selected dataset.

Inline steering

- *CMS-Wave file* – Location of the grid file or the simulation file. Can either "Browse for file" or use the "Wave grid".
- *Steering Interval* – Sets the recurring hot start output time. Option to use a "Constant" value or "Automatic" value.
- *Wave Water Level* – Determines the method used to calculate the water levels passed to the wave model. Option to select "Tidal", "Last time step", or "Tidal plus variation" (default value).
- *Extrapolation Distances* – If checked this will write out FLOW_EXTRAPOLATION_DISTANCE and WAVE_EXTRAPOLATION_DISTANCE cards.
    - *Flow to Wave* – Determines the extrapolation distance used for flow variables on the wave grid. Option to set a "User specified" value or "Automatic" value.
    - *Wave to Flow* – Determines the extrapolation distance used for wave variables on the flow grid. Option to set a "User specified" value or "Automatic" value.

Wind Data

Turns on the option to tell CMS to include wind calculations in the simulation. Wind is simulated in CMS-Flow as a spatially constant, but temporally varying quantity. The model does not currently support tropical cyclonic winds.

For more wind file information, see: CMS-Flow Wind Forcing on the CIRP wiki .

None

No wind data is imputed into the model.

Spatially constant

- *Parameters* – Set through the time series editor below. Clicking the button will add a row to the editor and clicking the button will remove a row. The follow options can be specified in each row:
  - *Time* (hr) – Specify time of wind measurement.
  - *Direction* (deg) – Specify the variation in direction of the wind in the simulation.
  - *Velocity* (m/s) – Specify the variation of wind velocity in the simulation.
- *Anemometer height* (m)

Meteororological stations

This option allows inserting data for multiple locations

- *Mereorological stations* – This section contains a properties editor where each station can be defined. A station is added using the button or removed using the button. The options are:
  - *Name* – The designated name of the station.

  - *X(m) – X-axis location of meteorologic station.*
  - *Y(m) – Y-axis location of meteorologic station.*
  - *Height (m)*
  - *Direction Curve* – Launches an XY Series Editor window.
- *Parameters* – Set through the time series editor below. Clicking the button will add a row to the editor and clicking the button will remove a row. The follow options can be specified in each row:
  - *Time* (hr) – Specify time of wind measurement.
  - *Direction* (deg) – Specify the variation in direction of the wind in the simulation.
  - *Velocity* (m/s) – Specify the variation of wind velocity in the simulation.

Temporally and spatially varying from file

The *File Type* chosen will determine the rest of the options available for wind type. There are three *File Type* options available:

- "Navy fleet numeric with pressure "
- "OWI/PBL"
- "Single ASCII file"

Navy fleet and ASCII files require the following input under "Parameters":

- *Number of values (X)*
- *Number of values (Y)*
- *Minimum X location*
- *Maximum Y location*
- *Time increment(s)*
- *Distance between X values (m)*
- *Distance between Y values (m)*

Alternatively, the "XY File" option can be selected and a file imported with the above parameters.

OWI/PBL requires three different files:

- *Oceanweather Wind File*
- *Oceanweather Pressure File*
- *Oceanweather XY File*

Output

This dialog allows specifying global output options for the simulation. This controls the datasets created by the engine which consist of spatially and temporally varied quantities (values per cell at each output time value).

These output datasets are stored in their respective XMDF (*.h5) files (e.g. current velocity in *vel.h5* and water surface elevation in *wse.h5*). In previous versions of CMS-Flow all the datasets were stored in a single solution (sol.h5) file, though that is not an option in current versions.

Output Times Lists

Due to the potentially long simulation times, CMS-Flow does not output at a constant interval during the simulation. Instead, define one or more lists of times. Select one time list for each dataset output by the engine. The selected quantity will be saved at each time in the time list.

CMS-Flow supports up to four different and individual sets of time values. The options for each list include:

- *Start time (hrs)*
- *Increment (hrs)*
- *End time (hrs)*

Each list can have multiple time sets. To add a time set, click on the Add (+) button. To remove a time set, click on the Remove (-) button.

Output Datasets

CMS-Flow can output each of the following datasets. For each group, a time list must be specified. The optional datasets may turned off/on by clicking on their checkboxes in the output dataset tree.

- *Water Surface Elevation*

- *Current Velocity*
  - *Current Magnitude* (optional)
  - *Current Velocity*
- *Morphology*
  - *Depth (through time)*
  - *Morphology Change* (optional)
- *Transport*
  - *Sediment Total-Load Capacity* (optional)
  - *Sediment Total-Load Concentration* (optional)
  - *Fraction Suspended* (optional)
  - *Total Sediment Transport*
  - *Salinity Concentration* (optional)
- *Waves*
  - *Wave Height*
  - *Wave Period*
  - *Wave Height Vector*
  - *Wave Dissipation* (optional)
- *Wind*
  - *Wind Speed* (optional)
  - *Wind Speed Vector*
- *Eddy Viscosity*

Statistical Output

Statistical data can be written out for the following:

- *Hydrodynamics*
- *Sediment Transport*
- *Salinity*

The options are turned on by toggles. For any option that is turn on, specify the following:

- *Start time (hrs)*
- *Increment (hrs)*
- *End time (hrs)*

Output Options

Output can be written out in ASCII format and the XMDF files can be compressed. These options are turned on by toggle boxes. All datasets created by the model will be labeled with a simulation label and quantity label.

The following file output options are available:

- *Tecplot snap shot (*.dat) and history files (*.his)*
- *SMS Super ASCII files (*.sup, *.xy, *.dat)*
- *XMDF file compression*
- *Simulation label* – Specify a name for output files.

Sediment Transport

Sediment transport options can be specified by toggling on the *Calculate Sediment Transport* option in the *Model Control* . Once this option is selected, sediment transport data will be calculated during the model run.

More information on sediment transport can be found at: Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change [128]

The options for sediment transport are:

Timing

The sediment transport and morphologic time steps are the time steps at which the transport and bed change equations are calculated.

- *Transport Time Step (not used for implicit scheme)* – Used for transport equation.
- *Morphologic Time Step (not used for implicit scheme)* – Used for updating bed elevation.
- *Morphologic change start time* – Sets start time for the morphology change calculation.

Formulation

- "Equilibrium Total Load" – Assumes both the bed load and suspended load to be in equilibrium. The bed change is solved using a simple mass balance equation known as the Exner equation.
- "Equilibrium Bed load plus Nonequilibrium Susp Load" – Conducted separately the calculations of suspended load and bed load. The bed load is assumed to be in equilibrium and is included in the bed change equation while the suspended load is solved through the solution of an advection-diffusion equation. Actually the advection diffusion equation is a non-equilibrium formulation, but because the bed load is assumed to be in equilibrium, this model is referred to the "Equilibrium A-D" model.

- "Nonequilibrium Total Load" – Assumes neither the bed nor suspended loads to be in equilibrium. The suspended- and bed-load transport equations are combined into a single equation and thus there is one less empirical parameter to estimate (adaptation length).

Transport Formula

- *Transport Formula* – Selects the transport formula.
    - "Lund-CIRP"
    - "van Rijn"
    - "Soulsby-van Rijn"
    - "Watanabe"
- *Concentration Profile* – Selects the concentration profile to be used either in the equilibrium A-D or NET models. In the A-D model, it is used to estimate the near bed concentration, whereas in the NET, it is used in the total load correction factor.
    - "Exponential"
    - "Rouse"
    - "Lund-CIRP"
    - "van Rijn"
- *Watanabe Transbort Rate Coefficient* – Sets the empirical coefficient which goes into the Watanabe transport formula.

Properties

- *Sediment Density* – Sets the sediment density in kg/m^3.
- *Sediment Porosity* – Sets the sediment porosity.

Scaling Factors and Coefficients

- *Bed load scaling factor* – Calibration factor for bed load transport capacity formula
- *Suspended load scaling factor* – Calibration factor for suspended load transport capacity formula
- *Morphologic acceleration factor* – Directly multiplies by calculated bed change.
- *Bed slope diffusion coefficient*
- *Hiding and exposure coefficient*

Adaptation

The adaptation coefficient is an important parameter to consider in setting up the CMS sediment transport model. The sensitivity of results to the adaptation coefficient depends on the spatial and temporal scales of the problem.

- *Total load adaptation method*
    - "Constant length" – Temporally and spatially total load adaptation length is used for the whole domain.
        - *Total load adaptation length*
    - "Constant time" – Temporally and spatially constant total-load adaptation time is used for the whole domain.
        - *Total load adaptation time*
    - "Maximum of bed and suspended adaptation lengths" – Temporally and spatially constant total-load adaptation time that uses a maximum suspended load adaptation length and maximum bed load adaptation length.
        - *Bed load adaptation method* – Specifies the bed load adaptation coefficient method. Options include:
            - "Constant Length" – Allows specifying a *Bed load adaptation length* .

    - "Constant Time" – Specifies the bed-load adaptation time.
    - "Depth Dependent" – Allows specifying a *Bed load adaptation depth factor* .
  - *Suspended load adaptation method* – Specifies the suspended load adaptation coefficient method. Options include:
    - "Constant length" – Allows specifying a *Suspended adaptation length* .
    - "Constant time" – Allows specifying a *Suspended adaptation time* .
    - "Constant coefficient" – Allows specifying a *Suspended adaptation coefficient* .
    - "Armanini and Di Silvio"
    - "Lin"
    - "Gallappatti"
- "Weighted average of bed and suspended adaptation lengths" – Has options similar to the "Maximum of bed and suspended adaptation lengths" option. A temporally and spatially constant total-load adaptation time that uses a suspended load adaptation length, a bed load adaptation length, and a fraction of suspended load of the total load.

Size Classes

The transported sediment material is discretized into different groups each representing the sediments within a specific size range.

- *Sediment size class diameters* – To add a sediment class, click on the button. To remove a sediment class, click on the button.
  - *Diameter* – Sediment size class characteristic diameter.
  - *Fall Velocity Method* – Sediment size class fall velocity formula.
  - *Fall Velocity* – Sediment size class fall velocity.
  - *Cory Shape Factor* – Sediment size class Corey Shape Factor. Used in the Wu and Wang (2006) sediment fall velocity formula.
  - *Critical Sheer Method* – Sediment size class characteristic critical shear stress formula.
  - *Critical Sheer Stress (Pa)* – Sediment size class characteristic critical shear stress.

Bed Composition

The initial bed layer thickness and composition are specified at least one layer for the whole grid.

- *Bed layer block* – To add a bed layer block, click on the button. To remove a bed layer block, click on the button.
  - *Layer ID* – Specifies the bed layer number from the surface. If not specified then set the sequential bed layer block number.
  - *Thickness Dataset* – Specifies the file name and dataset path (within the file) for the bed layer thickness dataset.

Percentile Diameters – Indicate the percentage of diameters smaller than a specific diameter.

- *D05*
- *D10*
- *D16*
- *D20*
- *D30*
- *D35*
- *D50*
- *D65*
- *D84*
- *D90*
- *D95*

Avalanching

Avalanching is the process of sediment sliding when the critical angle of repose is reached.

- *Calculate Avalanching* – Turns on or off the avalanching.
- *Critical bed slope* – Specifies the sediment repose angle. Avalanching is activated when the bed slope exceeds the repose angle.
- *Maximum number of iterations (implicit only)* – Maximum number of iterations for implicit solution scheme. When using the explicit scheme, one iteration is performed every time step.

Hardbottom

A morphologic constraint that provides the capability to simulate mixed bottom types within a single simulation.

- *Hardbottom depth* – Turning on this option and using the **Select** button will bring up a *Dataset* dialog. The *Dataset* dialog has the following options:
  - **Create** – Brings up the *Dataset Toolbox* where a new dataset can be created using the *Data Calculator* .
  - **Select** – Brings up an option to select an existing dataset already in the project.
  - **Unselect** – Removes the selected or created dataset.

Related Topics

- CMS-Flow

CMS-Flow Files

CMS-Flow creates a number of files during the model run.

Hot Start File

In the CMS-Flow *Model Control* , it is possible to specify a previously saved hot start file to be used as initial conditions or instruct CMS-Flow to save hot start files for future use.

To create a hot start file, either select *Write Hot Start output file* and select an output time, or select *Automatic recurring Hot Start file* and choose an output interval.

- Choosing to write the hot start file at a specific output time will create the following file:
  - *hot_start.h5* This file has the simulation data including elevations and velocities.
- Choosing automatic recurring hot start files will create the following files:
  - *HOTSTART.INFO* This file records what time the hot start file was written and which hot start file is the most recent.
  - *HOTSTARTx.H5* (where "x" is a counter) These files have the simulation data including elevations and velocities.

Once the hot start files are created, they can be read into CMS-Flow. Open the CMS-Flow *Model Control* and check the *Initial conditions file* check box, then select the hot start file to be used.

When using a hot start file, the following parameters should be changed as follows:

- *Start Date* : no change
- *Start Time* : no change
- *Simulation Duration* : decrease by the duration of the hot start file
- *Boundary Conditions* : no change

Note: For a simulation using a hot start file, the first time step of the solution will be the start time plus the value of one time step plus the duration of the hot start file.

Save Point File

*Storing Save Points in the *.cmcards File*

When saving the SMS project, the save points get stored in the *.cmcards file. An example looks like this:

Save Points

```
HYDRO_OUTPUT_INTERVAL         5.0  MINUTES
SEDIMENT_OUTPUT_INTERVAL      5.0  MINUTES
SALINITY_OUTPUT_INTERVAL      5.0  MINUTES
WAVE_OUTPUT_INTERVAL          5.0  MINUTES
```

The SAVE_POINT is formatted [name][x location][y location][hydro (if on)][sediment (if on)][salinity(if on)][wave(if on)]

```
SAVE_POINT         "6,  61"  -2867.5  2022.0  HYDRO  SEDIMENT  WAVE
SAVE_POINT         "20,  59"  -2517.5  1966.0  HYDRO
SAVE_POINT         "10,  31"  -2767.5  1182.0  SEDIMENT
SAVE_POINT         "11,  31"  -2742.5  1182.0  SEDIMENT
SAVE_POINT         "10,  32"  -2767.5  1210.0  SALINITY
SAVE_POINT         "11,  32"  -2742.5  1210.0  WAVE
```

**.sp/*.spx File for Reading in Save Point Information*

Simulation output data gets stored in *sp and *.spx file. The *.sp file is and individual output file. The 8.spx file simply identifies all of the *.sp files that should be loaded for a simulation. An *.sp file looks something like this:

```
SAVE_POINT_OUTPUT     eta
REFERENCE_TIME        2001/01/01  00:00  -0000  GMT
CREATION_DATE         2012/05/28  09:02  -0600  GMT
CMS_VERSION           4.00.11
TIME_UNITS            HOUR
SOUTPUT_UNITS          'm'
NUMBER_POINTS         2

NAME_BEGIN
   '6,  61'
   '20,  59'
NAME_END

XY_BEGIN
   -2867.5000    2022.0000
   -2517.5000    1966.0000
XY_END

SCALAR_TS_BEGIN
       0.0000    0.0000E+00    0.0000E+00
       0.0333    1.7545E-09    1.7434E-09
       0.0667    1.4042E-08    1.3971E-08
       0.1000    4.6679E-08    4.6524E-08
       0.1333    1.1143E-07    1.1117E-07
```

Sms will read in a *.sp file and create a CMS-Flow save point coverage with the points.

Projection Cards

CMS-Flow uses a local coordinate system in which all vector values are positive along the I and J axis. All output vector arrays are specified in the local coordinate system. Any input that is specified on the local grid must be specified in the local coordinate system (e.g. initial condition for currents, interpolated wave forcing, etc). If input vector arrays are specified on a different grid then the vectors are assumed to follow the coordinate system of their native grid. The grid is always created in SMS with the origin is by default always at the lower left hand corner of the grid.

Below are two examples of CMS Flow Projection Cards:

```
HORIZONTAL_PROJECTION_BEGIN                  !Optional
    DATUM                               NAD83       !NAD27|NAD83|LOCAL
    SYSTEM                              UTM       !UTM|STATE_PLANE|GEOGRAPHIC|LOCAL
    UNITS                               METERS       !METERS|FEET|DEGREES
    ZONE                                15       !Only  if  necessary
HORIZONTAL_PROJECTION_END
VERTICAL_PROJECTION_BEGIN
    DATUM                               LOCAL       !NGVD29|NAVD88|LOCAL
    UNITS                               METERS       !METERS|FEET
    OFFSET                              2.0  m       !Positive  is  upwards
VERTICAL_PROJECTION_END
```

Output Files

Output files specified here are associated with observation cells that have been assigned within the grid. For example, if a time series observation cell exists, an output file will be written out by M2D for every file type that is checked within this dialog. The same holds true for flow rate observation cells. All observation cell output files are given the file extension of ".m2o". A prefix is specified for all time series and flow rate output files. Also specify the time step increment (in seconds) at which to write to the output files. This increment should be a multiple of the simulation time step.

A brief explanation of the information that each of the following observation cell output file types contains is given (* = prefix):

Time Series Output Files:

- U Output (*_u.m2o): Velocity in the x-direction
- V Output (*_v.m2o): Velocity in the y-direction
- ETA Output (*_h.m2o): Water level
- U DETA/DX (u □□/□x) (*_udhdx.m2o):
- V DETA/DY (v □□/□y) (*_vdhdy.m2o):
- ETA DU/DX (□□□u/□x) (*_hdudx.m2o):
- ETA DV/DY (□□□v/□y) (*_hdvdy.m2o):
- X-Momentum Advection U DU/DX (u □u/□x) (*_xmomu.m2o): U component of the momentum advection term in the x-direction
- Y-Momentum Advection U DV/DX (u □v/□x) (*_ymomu.m2o): U component of the momentum advection term in the y-direction
- X-Momentum Advection V DU/DY (v □u/□y) (*_xmomv.m2o): V component of the momentum advection term in the x-direction
- Y-Momentum Advection V DV/DY (v □v/□y) (*_ymomv.m2o): V component of the momentum advection term in the y-direction
- X Bottom Friction (*_xbfric.m2o): X component of the bottom friction

- Y Bottom Friction (*_ybfric.m2o): Y component of the bottom friction
- X Wind Stress (*_xwnd.m2o): X component of the wind stress
- Y Wind Stress (*_ywnd.m2o): Y component of the wind stress

Flow Rate Output Files:

- X Direction (*_qx.m2o): Flow rate in the x-direction
- Y Direction (*_qy.m2o): Flow rate in the y-direction

Related Topics

- CMS-Flow
- CMS-Flow Model Control

CMS-Flow Spatial Datasets

Spatially varied data, or spatial datasets are integral to the use and application of the CMS-Flow numerical engine. The engine uses spatial data for input and each process simulated by the engine creates spatial datasets that can then be visualized and used for post processing in the SMS.

Input Datasets

CMS-Flow has several spatially variable input fields. The inputs are sorted by process and labeled with units below.

Hydrodynamics

Variable	Units	Symbol	Description
Bottom Friction	NA	(none)	Roughness of the bottom used to simulate resistance to flow. For example, this could be based on a Manning value.

Sediment transport/Morphology change

Variable	Units	Symbol	Description
D50	milimeters	(*mm*)	Median grain size for particles in each cell of the grid domain
Hard Bottom	meters	(*m*)	Constraint for maximum erosion for each cell of the grid domain

Salinity

Variable	Units	Symbol	Description
Initial Concentration	parts per thousand	(*ppt*)	Initial salinity concentration for each cell of the grid domain

Output Datasets

Hydrodynamics

Variable	Units	Symbol	Description
Water Surface Elevation	meters	(*m*)	Elevation above mean sea-level computed at the centroid of each cell.
Current Velocity	meters per second	(*m/sec*)	Velocity components in the u and v directions computed at the top and right side of each cell. The SMS interface interpolates these values over the cell, converts them to global world (x, y)

			values and displays these as vectors.
Flow Rate	cubic meters per second	(m^3/sec)	Flux over each cell.

Sediment transport

Variable	Units	Symbol	Description
Concentration Capacity	kilogram per meter cubed	(kg/m^3)	Total transport capacity for flow in the cell. Computed for LUND-CIRP, van Rijn, and Watanabe transport equations.
Sediment Concentration	kilogram per meter cubed	(kg/m^3)	Computed suspended sediment concentration at the centroid of each cell
Sediment Transport Rate	cubic meters per second per meter	($m^3/sec/m$)	Computed transport rate across the right and top faces of each cell. Displayed as a vector quantity in SMS

Morphology change

Variable	Units	Symbol	Description
Bed depth	meters	(m)	Depth below mean sea level to current floor of the model (positive values measured downwards)
Change in depth	meters	(m/sec)	Change in the depth at each cell since the start of the simulation

Salinity

Variable	Units	Symbol	Description
Salinity Concentration	parts per thousand	(ppt)	Concentration at each cell centroid

Related Topics

- CMS-Flow

Lund Cirp and Watanabe Formula

The Lund Cirp and Watanabe formula can be found on page 16 of the *Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR* [3] .

The total load sediment transport rate of Watanabe is given by:

$$q_{tot} = A\left[\frac{(\tau_{b,max} - \tau_{cr})U_c}{p_w g}\right]$$

where

q_{tot} = total load (both suspended and bed load)

$\tau_{b,max}$ = maximum shear stress at the bed

τ_{cr} = shear stress at incipient sediment motion

U_c = depth averaged current velocity

ρ_w = density of water

g = acceleration of gravity

---

3 Aug. 2006 - Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR [129]

A = empirical coefficient typically ranging from 0.1 to 2

Transport Slope Coefficient

The Transport Slope Coefficient can be found on page 32 of the *Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR* .

$$\bar{q}'_{tot,x} = \bar{q}_{tot,x} + D_s \left|\bar{q}_{tot}\right| \frac{\partial h}{\partial x}$$
$$\bar{q}'_{tot,y} = \bar{q}_{tot,y} + D_s \left|\bar{q}_{tot}\right| \frac{\partial h}{\partial y}$$

D_s = empirical slope coefficient with typical range of 5 to 30.

The Transport Slope Coefficient can vary site by site and even within a single site domain in that some areas have constraints with naturally occurring steep bed slopes (e.g., channels) and other areas have gentle slopes (e.g. beach profiles, or tidal flats). It is a diffusion coefficent for increasing downhill transport or decreasing uphill transport (if D is >1) This is a good parameter to use as a morphology change calibration factor (along with the scalesus and scalebed coefficients). One thing to note is that what may calibrate well for one area will not calibrate well for another so an average value may be necessary.

Additional Information

Karambas, T.V. 2003. "Nonlinear wave modeling and sediment transport in the surf and swash zone," Advances in Coastal Modeling, V.C. Lakhan (ed.), Elsevier Oceanography Series, 67, Amsterdam, The Netherlands, 267-298

Larson, M., Hanson, H., and Kraus, N.C. 2003. "Numerical modeling of beach topography change," Advances in Coastal Modeling, V.C. Lakhan (ed.), Elsevier Oceanography Series, 67, Amsterdam, The Netherlands, 337-365

Chapter 4 of

Horikawa, K. 1988. (ed.) "Nearshore dynamics and coastal processes. Theory, measurement, and predictive models," University of Tokyo Press, Tokyo, Japan

Related Topics

- CMS-Flow
- Q&A CMS-Flow

5.5.b. CMS-Wave

CMS-Wave

CMS-Wave	
Model Info	
Model type	Designed for accurate and reliable representation of wave processes affecting operation and maintenance of coastal inlet structures in navigation projects as well as in risk and reliability assessment of shipping in inlets and harbors.
Developer	Lihwa Lin, Ph.D.
Web site	http://cirp.usace.army.mil/

Tutorials	General Section • Data Visualization • Observation Models Section • CMS - CMS-Wave (pending)

CMS-Wave (formerly known as WABED) is one of the principal components of the Coastal Modeling System (CMS) . WABED stood for "Wave-Action Balance Equation Diffraction" model. The model is a 2-D wave spectral transformation (phase-averaged). The term "phase-averaged" means that it neglects changes in the wave phase in calculating wave and other nearshore processes. This class of wave models represent changes that occur only in the wave energy density. It was originally built to represent theoretically developed approximations for both wave diffraction and reflection in a nearshore domain.

The SMS interface to CMS-Wave includes tools for creating input files as well as post-processing capabilities.

The CMS-Wave model can be added to a paid edition of SMS.

Graphical Interface

The CMS-Wave Graphical Interface contains tools to create and edit a CMS-Wave simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to CMS-Wave. If a grid has already been created for a CMS-Wave simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to CMS-Wave. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian grid menus and tools augmented by the CMS-Wave Menu . See CMS-Wave Graphical Interface for more information.

Model Data Files

Four input files are required for a CMS-Wave simulation. Up to six optional input files may also be utilized depending on the processes being modeled and the selected model parameters. Depending on which options are selected for the simulation, CMS-Wave will generate from one to six output files. See CMS-Wave Files for more information.

Sample Simulations

Simulations illustrating the model capabilities will be posted here soon.

Model Development History

- July 2006 – Initial CHETN announcing the release of WABED is published. Interface is available through the STWAVE interface in SMS.
- December 2007 – Four new features were recently added to CMS-Wave. These include:
  - wave run-up – The wave run-up calculation includes both wave setup and maximum vertical swash that enter, for example, in beach erosion during storms.
  - wave transmission and overtopping at structures – The calculation of wave transmission and overtopping is possible for either vertical wall or rubble- mound jetties and breakwaters, and for submerged reefs. These wave run-up and wave overtopping structure calculations are necessary, for example, in the study of overwash and flanking of beaches adjacent to jetties.
  - card-format for convenient model control

- option for running in a Fast Mode – With the new *Fast-Mode* option, CMS-Wave calculates the spectral transformation on five directional bins (each 30-deg angle for a broad-band input spectrum) or on seven directional bins (each 5-deg angle for a narrow-band input spectrum, or on 25- deg angle for wind input) to minimize execution time. In Fast Mode, the wave model is at least five times faster than in the Normal Mode, which operates on 35 directional bins in the half plane. Fast Mode is recommended when needing rapid calculation in reconnaissance or test applications. Wave information estimated in the Fast Mode is expected to be less accurate than in the Normal Mode because the calculation is based on fewer directional bins. Normal Mode should be specified for final runs.

Related Topics:

- SMS Models Page
- CMS-Flow

External Links:

- CMS-Wave Explanatory Report Aug 2008
- CMS-Wave User Manual Mar 2012
- CIRP CMS Wiki
- August 2008 ERDC/CHL TR-08-13 CMS-Wave: A Nearshore Spectral Wave Processes Model for Coastal Inlets and Navigation Projects [130]
- May 2007 ERDC/CHL CHETN-I-74 WABED Model in the SMS: Part 2. Graphical Interface [131]
- Jul 2006 ERDC/CHL CHETN-III-73 Wave-Action Balance Equation Diffraction (WABED) Model: Tests of Wave Diffraction and Reflection at Inlets [132]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [133]
- 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) [134] Updated FAQ Website [135]
- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [136]
- April 2012 ERDC/CHL CHETN-IV-81 Full-plane Wave Transformation and Grid Nesting. [137]

CMS-Wave Cell Attributes Dialog

The *CMS-Wave Cell Attributes* dialog is used to assign a cell type to selected Cells. It is opened using the menu command *CMS-Wave* | **Assign Cell Attributes** . The following cell types can be assigned:

Default

The elevation values of these cells will be used in order to determine whether flow exists in each cell or not.

Structure

The cell attributes dialog of CMS-Wave supports various structure types. Each type of structure can specify an attribute value. The following structure types are supported:

- **Bathymetry modification** – For adding alternative feature or structure (immersed or exposed) without modifying the input depth
- **Wave runup** – For calculation of wave runup and overwash on beach face or structure, and adjacent land
- **Floating breakwater** – For calculation of transmitted waves of a floating breakwater
- **Wall breakwater** – For a vertical wall breakwater
- **Rubble-mound** – For a composite or rubble-mound breakwater

- **Piers/docks** – High permeability; for a pier or dock
- **Rubble-mound breakwater** – Low permeability; for a rubble-mound breakwater

Each of the structures can provide an optional attribute value. The value specified is as follows:

CMS-Wave Structure Attributes

Structure Type	Attribute Name	Description	If Not Provided
Bathymetry modification	Depth	Feature structure depth	Assumed land if not provided
Wave runup	Elevation	Beach/structure elevation above mean water level (no effect if < 0.0)	Assumed input depth
Floating breakwater	Draft	Floating breakwater draft (no effect if < 0.05 m)	No effect
Wall breakwater	Elevation	Beach/structure elevation above mean water level (immersed if < 0.0)	Assumed input depth
Rubble-mound	Elevation	Beach/structure elevation above mean water level (immersed if < 0.0)	Assumed input depth
Piers/docks – High permeability	Porous layer thickness	See permeable structures below.	
Rubble-mound breakwater – Low permeability	Porous layer thickness	See permeable structures below.	

Permeable Structures

CMS-Wave treats cells designated as "Piers/Docks" and "Rubble Mound Breakwater" as if they have a structure that fills the entire water column. If the water level rises, the assumption is made that the structure is still all the way to the water surface (and beyond if needed). CMS-Wave ignores the specified depth for these cells when the structure type is specified. The layer thickness defines the thickness that will convey wave energy. Physically, this is like the distance from the mean sea level down to the footing of the pier group or dock structure. For a rubble mound, it is the distance below mean sea level where the mound sits on solid material (non-permeable).

Monitoring Station

Spectral output is generated for cells designated as monitoring station cells. Will also generate a station output file .

Nesting output

Spectral output is generated for nesting output cells cells to be used as input for a nested child grid .

GenCade monitoring station

Generates 1D grid data for use with GenCade.

Displaying Cell Attributes

The symbols and colors used to identify the attributes assigned to cells can be changed in the *Display Options* dialog.

Related Topics

- CMS-Wave
- CMS-Wave Menu

CMS-Wave Graphical Interface

The CMS-Wave graphical interface includes tools to assist with creating, editing, and debugging a CMS-Wave model. The CMS-Wave interface exists in the Cartesian Grid Module .

Model Control

The *CMS-Wave Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The CMS-Wave files are written automatically with the SMS project file or can be saved separately using the *File* | **Save CMS-Wave** or *Save As* menu commands. See CMS-Wave Files for more information on the files used for the CMS-Wave run.

CMS-Wave can be launched from SMS using the *CMS-Wave* | **Run CMS-Wave** menu command. A check of some of the common problems, called the Model Checker, is done each time the model is launched, or by selecting the *CMS-Wave* | **Model Check** menu command.

Visualizing Results

Select the spectral grid as the grid to use in the *Spectral Energy* dialog. This will open the spectral grid for viewing in the *Spectral Energy* dialog. Make sure to set the original grid back before leaving the dialog to ensure the model does not change.

CMS-Wave Menu

See CMS-Wave Menu for more information.

Related Topics

- CMS-Wave
- Create a 2D Cartesian grid

CMS-Wave Menu

The following menu commands are available in the CMS-Wave Menu:

Assign Cell Attributes

Opens the *CMS-Wave Cell Attributes* dialog, which specifies the cell type.

Merge Cells

Merge multiple rows or columns into a single row or column.

Model Check…

Check for common problems. If problems are found, the *Model Checker* dialog will open.

Model Control…

Opens the *CMS-Wave Model Control* dialog to specify model parameters.

Run CMS-Wave

Launches the CMS-Wave model using the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

Obsolete Commands

The following commands are no longer available in the current version of SMS.

Nest Grid

Opens the *CMS-Wave Nesting Options* dialog, which specifies nesting options.

Spectral Energy

Opens the *Spectral Energy* dialog to define/view wave energy spectra. Generally, CMS-Wave will generate wave conditions internally, but a spectrum may be input. This command also allows visualizing wave spectra that are generated inside of the model.

Related Topics

- CMS-Wave

CMS-Wave Model Control

The **Model Control…** command in the *CMS-Wave* Menu opens the *CMS-Wave Model Control* dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

Input Forcing

- **Currents** – A vector dataset can be used to define the currents for the simulation
- **Water Level** – A constant value or a scalar dataset can be used to define the water level for the simulation. If "constant" is selected, the constant value will be defined in the *CMS-Wave Case Definition* dialog for each case.
- **Spectra**
  - *Source* :
    - **Spatially Varied** – this option uses a spectral coverage with spectral data defined at one or more locations. The spectral coverage is assigned to the model in the *CMS-Wave Case Definition* dialog.
    - **Parent Grid** – Nesting output from a parent grid will be used to drive the simulation. The parent grid is specified by clicking on the **Select…** button and selecting the desired grid. The spectral grid to be used for the simulation will match the spectral grid for the parent simulation.
  - *Plane Type* – this option is only used if the *Spatially Varied* source option is selected.
    - **Half plane** – appropriate for nearshore coastal applications allowing wave input and generation on two boundaries resulting in a faster run-time.
    - **Full plane** – used with enclosed or semi-enclosed bays, estuaries, and lakes where there is no clear "offshore" direction and seas and swells may oppose each other. Allows wave input and geration on all four boundaries.
    - **Full plane with input reverse spectra** – this option allows spectral input to be used on two opposite boundaries.
  - *Interpolation Type* – sets the method for STWAVE to use when interpolating between spectra.
    - **Average spectra**
    - **IDW interpolation** – Inverse Distance Weighting
  - *Spectral Grid*
  - *Create Nesting Points* – if the *From parent grid* source option is selected and a parent grid has been specified, clicking on this button will allow specifying which points should be used from the parent grid for creating the nesting file.

- **Wind** – a constant value or a vector dataset can be used to define the wind for the simulation. If constant is selected, the wind direction and magnitude will be specified in the *CMS-Wave Case Definition* dialog for each case.
- **Define Cases**

Settings

The settings section is used to specify model input options including:

- Bed friction
- Matrix Solver
  - Gauss-Seidel
  - ADI
- Allow wetting and drying
- Infragravity wave effect
- Diffraction intensity
- Forward reflection – A constant value for the entire simulation or spatially varying data using a scalar dataset can be used to define the forward reflection for the model.
- Backward reflection – A constant value for the entire simulation or spatially varying data using a scalar dataset can be used to define the backward reflection for the model.
- Muddy bed – Spatially varying data using a scalar dataset can be used to define the muddy bed (values??) for the model.
- Non-linear wave effect
- Run up
- Quick mode

Output

This section is used to specify option output from the model. The output includes:

- Radiation stresses
- Sea/Swell
- Wave Breaking

Input Datasets

Here specify the format for SMS to write out the input datasets. They can be written in either ASCII or XMDF format.

Related Topics

- CMS-Wave

5.5.b.1. CMS-WAVE Files

CMS-Wave Files

A CMS-Wave simulation consists of:

- A Cartesian grid which defines the bathymetric depths over the model domain

- A set of spectra that represent the input wave conditions to be modeled
- Model parameters controlling what options should be used in this simulation
- Optional spatially varied input fields over the same domain such as wind datasets, surge (water level) datasets, and current datasets

SMS stores all of this information internally in its own formats. This includes:

- An XMDF (HDF5) file to store the bathymetry and other datasets on the grid
- An XMDF (HDF5) file to store the spectral grids and wave states entering the domain from the boundary
- Model parameters in the project (sms) file.

In order to run the numeric model, the data must be exported into model native file formats. SMS exports these files when instructed to do so from either the *CMS-Wave* menu or by right-clicking on the CMS-Wave grid. SMS includes three commands in either location. These include:

1) **Export CMS-Wave Files** – Creates the native files described below for the simulation. If the data has not been previously saved to establish the name of the simulation, SMS prompts for a simulation name.
2) **Launch CMS-Wave** – Uses the saved files to invoke the simulation
3) **Save project, export and launch CMS-Wave** – This command, as described, saves the SMS project and then replicates the functionality of the other two commands.

The model native files are referenced by a simulation file, which includes all the names of the other files used as input for a simulation, or that will be created by CMS-Wave when it runs.

SMS stores a simulation name associated each CMS-Wave grid loaded into a session. If a new grid is created, the name is blank. SMS will prompt for a name when it is needed.

If the **Launch CMS-Wave** command is issued without having first saved the model native files, SMS will bring up a message indicating that the files must first be saved.

To change the name of the simulation associated with a grid, select the *File* | **Save as...** command and select the *CMS-Wave simulation* option. SMS will then prompt for the new simulation name to be associated with the active CMS-Wave grid. It is recommended that this option be utilized only after duplicating the CMS-Wave grid, since the link to the old simulation files is lost when the new simulation name is specified.

Input Files:

- Required files
  - *.sim – Simulation file.
  - *.std – Control file.
  - *.dep – Depth file.
  - *.eng – Spectral energy file.
- Optional files
  - *.cur – Current file
  - *.eta – Water level file (uses the same format as the *.dep file)
  - *.struct – Wave structure file
  - *.fric – Bottom friction coefficient file (uses the same format as the *.dep file)
  - *.fref – Forward reflection coefficient file (uses the same format as the *.dep file)
  - *.bref – Backward reflection coefficient file (uses the same format as the *.dep file)
- Auxiliary files
  - *.txt – A spectral table file

Output Files

- Spatially varied data (value for each cell)
  - *.wav – A wave output file.
  - *.brk – A wave breaking file.
  - *.rad – A radiation stress file.
- Full spectra at selected cells (have the same format as the *.eng file)
  - *.obs
  - *.nst
- Additional files
  - setup.wave – Text file of wave runup/setup at specified structure locations.
  - selhts.out – Output text file of wave height, period and direction at specified observation locations.

Related Topics

- CMS-Wave

CMS-Wave Control File

CMS-Wave parameter files (*.std) have been reworked for SMS 11.1 and later versions and their associated versions of CMS.

File Format

The new CMS-Wave parameters file (also referred to as an options or std file) begins with an identifier line. This line includes the text CMS_WAVE_STD and a version number.

Each subsequent line on the file includes a value, followed by a comment defining the option. The design was intended to replicate a card based ability to omit lines if the default value is to be used.

Sample File

```
CMS_WAVE_STD 1 !free format
1 !iprp
0 !icur
0 !ibreak
0 !irs
0 !kout


0 !ibnd
0 !iwet
0 !ibf
0 !iark
0 !iarkr
```

```
0 !iwvbk
0 !nonlin
0 !igrav
0 !irunup
1 !imud
1 !iwnd


4.0000 !akap
0.025 !bf
0.5000 !ark
0.3000 !arkr
0 !isolve
0 !ixmdf
1 !iproc
0 !iview
0 !inest
```

Old File Format

The old parameter file always contain a single line of data defining the parameter options for the simulation. This line will include either 6, 10 or 15 values. SMS writes all 15 values to the input files it creates. The other formats are supported for backward compatibility.

If only the first six values are present, the other nine values will be assigned default values. The default for the diffraction intensity factor is weak diffraction (akap = 1.0). The default for the other eight parameters is 0 (defined in table below).

If ten values exist in the file, bottom reflection is defaulted to off, forward reflection is defaulted to 50% and reverse reflection is defaulted to 30%.

After this single line, additional lines may be present to define the location of observation and nesting cells. CMS-Wave will save additional output (spectra) at these cells into the specified *.obs and *.nst files.

A sample file and table of the parameters is defined below.

```
iprp icur ibrk irs kout ibnd iwet ibf iark iarkr akap bf arc arkr iwvbk
kout rows (each row includes the I and J indices of a selected output cell
nestout
nestout rows (each row includes the I and J indices of selected nesting output cells
```

Sample Old File

```
1 0 0 0 2 0 0 0 0 0 4.000000 0.005000 0.500000 0.300000 0
10 15
```

```
23 41
4
28 7
27 8
28 9
29 10
```

Parameter definitions

Parameter	Description
iprp	-1, for fast-mode simulation
	0, for wave generation and propagation (uses wind input and spectra)
	1, for propagation only (neglect wind input)
	2, for wind only (implies a zero energy spectra)
icur	0, no current file read
	1, read current file with a dataset for each input spectra
	2, read single data set from current file for all spectra
ibrk	0, no breaking file will be created
	1, breaking file with be created with breaking index for each cell
	2, breaking file with be created with energy dissipation flux saved for each cell
irs	0, no radiation stress gradient file will be created
	1, radiation stress gradient file will be created

	2, radiation stress gradient file and wave setup/maximum water level files will be created
kout	0, no selected output data files will be created
	n, output spectra (.obs) and parameters (selhts.out) files will be created at n specified cells
ibnd	0, simulation will use single spectra (*.eng) file along offshore boundary
	1, simulation will use nesting file (.nst) with linear interpolation of boundary input
	2, simulation will use nesting file (.nst) with morphic interpolation of boundary input
iwet	0, normal wetting/drying (based on water level input)
	1, no wetting/drying (water level input ignored)
ibf	0, no bottom friction
	1, for bottom friction with constant Darcy-Weisbach type coefficient (bf)
	2, for bottom friction with variable Darcy-Weisbach type coefficient (friction.dat)
	3, for bottom friction with constant Manning coefficient (bf)
	4, for bottom friction with variable Manning coefficient (friction.dat)
iark	0, no forward reflection
	1, with forward reflection
iarkr	0, no backward reflection

	1, with backward reflection
akap	diffraction intensity factor (0 for no diffraction, 4 for strong diffraction)
bf	constant bottom friction coefficient
ark	constant forward reflection coefficient (0 for no reflection, 1 for maximum forward reflection)
arkr	constant backward reflection coefficient (0 for no reflection, 1 for maximum backward reflection)
iwvbk	0, for extended Goda wave breaking (Sakai et al. 1989)
	1, for extended Miche wave breaking (Battjes 1972; Mase et al. 2005b)
	2, for Battjes and Janssen wave breaking (1978)
	3, for Chawla and Kirby wave breaking (2002)

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Depth File

CMS-Wave requires a depth at each cell in the domain. In SMS terminology, the depth values comprise a dataset on the grid.

The depths are passed into CMS-Wave in the form of an ASCII data file. There is no header or identifier as to what the data in the file is, other than the default extension of ".dep". However, that is not even required.

The first line of the file contains the dimensions of the grid (number of columns, number of rows) and the default size of a grid cell (in the I direction and then the J direction). If the grid does not consist of cells of constant size, the size for the J direction will be written as 999. In this case, the cell dimensions (width of each column and height of each row) are included at the bottom of the depth file.

The values in the file are organized in row major format, starting with the "top" row (farthest away from the grid origin) and ending with the "bottom" row (closest to the origin and following the I direction).

SMS writes files with five values per row.

Sample File

25 12 10.000 999.000	// number of columns, number of rows, cell dimensions
46.65 42.56 38.85 35.47 32.40	// first 5 depths of cells on the top row
29.61 27.07 24.77 22.67 20.76	// next 5 values, second row will continue on same line
. . .	// remaining depth values
7.97 7.25 6.55 5.99 5.44	// first five column widths
4.95 4.50 4.09 3.72 3.38	// next 5 widths, heights will follow on same line
. . .	// remaining widths and heights

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Monitoring Station Output File

CMS-Wave can include monitoring stations that will extract the calculated data values, including the wave height, period and direction. The extracted values are stored in the CMS-Wave monitoring station output file (selhts.out) generated during the model run. This file will only be created if a monitoring station data is specified in the control file . In SMS, monitoring stations are created as a cell attribute in the Cartesian grid.

File Format

The selhts.out file uses a fifteen column format with values described in the table below:

Column Number	Value
1	Spectrum label (or time stamp)
2	I index in Cartesian (i,j) or ij index in quadtree mesh
3	J index in Cartesian (i,j) or 0 (dummy) in quadtree mesh
4	Significant wave height (meters)
5	Spectral peak wave period (seconds)
6	Mean wave direction (degrees, local coordinate system)
7	Swell height (meters)
8	Swell wave period (seconds)
9	Swell direction (degrees)
10	Local-generated wave height (meters)
11	Local-generated wave period (seconds)
12	local-generated wave direction (degrees)
13	Wave breaking index (non-zero for breaking)
14	Max water elevation mark (meters), include wave run-up if calculated
15	Flow discharge rate (m*m/sec) at structure cells if calculated

Example

Below is an example of the selhts.out text file.

```
1    50    70   1.036  20.00    23.2     0.00   0.00     0.0     0.00   0.00     0.0
0   0.518    0.00
1    92    66   0.763  20.00   -13.9     0.00   0.00     0.0     0.00   0.00     0.0
0   0.382    0.00
1   110    60   0.169  20.00    -8.2     0.00   0.00     0.0     0.00   0.00     0.0
0   0.085    0.00
2    50    70   0.010   1.00     0.0     0.00   0.00     0.0     0.00   0.00     0.0
0   0.005    0.00
2    92    66   0.010   1.00     0.0     0.00   0.00     0.0     0.00   0.00     0.0
0   0.005    0.00
2   110    60   0.010   1.00     0.0     0.00   0.00     0.0     0.00   0.00     0.0
0   0.005    0.00
3    50    70   0.010   1.00     0.0     0.00   0.00     0.0     0.00   0.00     0.0
0   0.005    0.00
3    92    66   0.010   1.00     0.0     0.00   0.00     0.0     0.00   0.00     0.0
0   0.005    0.00
3   110    60   0.010   1.00     0.0     0.00   0.00     0.0     0.00   0.00     0.0
0   0.005    0.00
```

Related Topics

- CMS-Wave Files

CMS-Wave Simulation File

CMS-Wave simulation files (*.sim) contain a list of file names that are part of the simulation. This file is created by SMS when the simulation is exported. The file begins with a keyword that identifies this as a CMS-Wave simulation file, followed by the world origin and orientation of the grid.

The origin positions the grid in a projection (such as UTM or State Plane). It generally corresponds to the location of the corner of the grid offshore. The orientation is the CCW angle from East to the direction of the grid that goes from offshore to on shore. (For example, a coast that opens due West would have an orientation of 0.0, an coast line open to the South would have an orientaion around 270 (or -90) degrees.)

Each line begins with a key word identifying the file name. The order can be changed and only the DEP, OPTS and WAVE files are strictly required. The OPTS file includes the options that have been enabled for this simulation. If the OPTS file indicates that a file is needed, the line must appear in the SIM file. If a file is missing, the simulation will not perform as expected. If files are not needed, the presence of the line in the SIM file is ignored. For this reason, SMS writes all files to the SIM file every time.

Sample File

CMS-Wave 902500.0 4086600 135		/* Key word, grid origin, grid orientation*/
DEP	grid.dep	/* Name of input file containing depth values for each cell */
OPTS	grid.std	/* Name of input file containing model parameters */
SPEC	grid.eng	/* Name of input file containing energy density spectra (required in most cases) */
OBSE	grid.obs	/* Name of output file to save full spectra at specified monitoring locations */
NEST	grid.nst	/* Name of output file to save full spectra at specified nesting cells */
BREAK	grid.brk	/* Name of output file to save wave breaking indices or energy disipations at each cell. Follows the format of the DEP file */
SPGEN	grid.txt	/* Name of spectral parameter file, contains parameters used to generate spectra */

RADS	grid.rad	/* Name of output file to save wave radiation stress gradients at each cell. Follows the format of the CUR file */
STRUCT	grid.struct	/* Name of input file containing structure flags for each cell */
SELHTS	grid.out	/* Name of output file where CMS-Wave will write the bulk wave parameters for selected monitoring cells */
ETA	grid.eta	/* Name of optional input file containing water levels for each cell. Follows the format of the DEP file. This replaces the SUR card. Both are supported */
FRIC	grid.fric	/* Name of input file containing manning's n value for each cell. Follows the format of the DEP file */
FREF	grid.fref	/* Name of input file containing forward reflection value for each cell. Follows the format of the DEP file */
BREF	grid.bref	/* Name of input file containing reverse reflection value for each cell. Follows the format of the DEP file */
MUD	grid.mud	/* Name of input file containing mud(absorption) coefficient value for each cell. Follows the format of the DEP file */
WIND	grid.wind	/* Name of input file containing wind components for each cell. Follows the format of the CUR file */
WAVE	grid.wav	/* Name of output file for spatial wave conditions (height, direction) for each cell */
CURR	grid.cur	/* Name of optional input file containing current values (vx and vy) for each cell */

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Spectral Energy File

CMS-Wave energy files (*.eng) contain a the definition of a half plane spectral grid (5 degree directional bins and user specified frequency bins) and energy density datasets as boundary conditions for each wave condition to be simulated in a CMS-Wave analysis.

File Format

The first line of the file contains the number of frequency bins and directional bins. Typically, 20-30 frequency bins are used. The model requires 5 degree directional bins, so the number of directional bins must be set to 35.

Following the grid dimensions, the next lines of the file specify the frequencies for the model spectra, starting at the lowest frequency. There must be the specified number of values. They are read in free format and may occupy as many lines as needed. These frequencies should span the range where significant wave energy is contained in the spectrum. This can be estimated by inspecting the input spectrum or estimating the peak period expected using the wave growth curves int eh Coastal Engineering Manual. A rule of thumb is that the spectral peak should fall at about the lower one-third of the frequency range (e.g., if the peak frequency is 0.1 Hz, the range may be 0.01 to 0.3 Hz). Wave frequencies higher (or periods shorter) than the highest frequency bin or lower than the lowest frequency bin will not be resolved by the model. Typically, frequency increments are on the order of 0.01 Hz, but the increment need not be constant. West Coast applications will tend to require finer resolution focused at lower frequencies because of long wave periods. Gulf Coast or Gread lakes applications will tend to require coarser resolution covering a broader range of frequencies because of shorter wave periods.

Following the specification of the frequency and direction bins is a header line containing a spectrum identifier, wind information, peak frequency and water elevation correction. This line is read in free format.

After the header line, the file contains the energy densities in the units of meters squared/hertz/radian. The spectrum is read starting with the lowest frequency and reading all the direction s(-85 to 85), then reading energy densities for the next lowest frequency etc.

The file can contain multiple spectra.

Sample File

```
30 35
0.04000 0.05000 0.06000 0.07000 0.08000 0.09000 0.10000 0.11000 0.12000 0.13000
0.14000 0.15000 0.16000 0.17000 0.18000 0.19000 0.20000 0.21000 0.22000 0.23000
0.24000 0.25000 0.26000 0.27000 0.28000 0.29000 0.30000 0.31000 0.32000 0.33000
5 27.000000 0.000000 0.070000 0.000000
0.00000 0.00000 0.00000 0.00000 0.00000 0.00002 0.00006 0.00019 0.00048 0.00107 0.00209 0.00365 0.00576
.
.
.
```

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave Spectral Table File

Spectral Table Files (*.txt) contain the parameters for generated wave spectra. The ASCII file is created by SMS when the *Spectral Energy* dialog is used to create wave spectra. If wave spectra are imported from buoys or another source, this file will not exist. When this file exists, SMS can use the values in the table to populate the table in the *Spectral Energy* dialog to allow editing the spectra being used.

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

Sample File

```
ACE/vis drogue path file                       /* Title */
5                                              /* Number of Time Steps */
                                               /* Current Time Step and Number of Particles */
7200.0000              199
                                               /* xy values and id */
-0.766993322E+02 0.346589454E+02  1
```

```
-0.766986001E+02 0.346616775E+02  2

.

.
                                  /* Next Time Step and Number of Particles */
8000.0000              199

.

.
/* EOF */
```

The number of particles must be the same for each time step.

Related Topics

- CMS-Wave Files
- File Formats

CMS-Wave STD

CMS-Feature enhancements – June 15, 2010 Support the following mode parameters that appear on the first line of the std file (there are a total of 23 of them)

Sample of first line from *.std: 0 0 0 0 0 0 0 0 1 0 4.000000 0.005000 0.500000 0.300000 0 0 1 0 0 0 0 0 0

These parameters represent the following variables in the code:

- iprp
- icur
- ibreak
- irs
- kout
- ibnd
- iwet
- ibf
- iark
- iarkr
- akap
- bf
- ark
- arkr
- iwvbk
- nonln
- igrav
- irunup
- imud
- iwnd
- isolv
- ixmdf
- iproc

The first 15 parameters should already be supported in the SMS interface and are documented at: CMS-Wave Control File The new ones (16th to 23th) and their possible values include:

- nonln (nonlinear wave-wave effect):

  0 = none (default)

  1 = present
- igrav (infragravity wave effect):

  0 = none (default)

  1 = present
- irunup (run-up):

  0 = none (default)

  1 = automatic,

  2 = field file input (*.runup or runup.dat)
- imud (muddy bed):

  0 = field file input (*.mud or mud.dat, default)

  1 = none

- iwnd (wind field input):

0 = field file input (*.wind or wind.dat - same format as *.cur, default)

1 = none

- isolv (Matrix solver):

0 = ADI (default),

1 = Gauss-Seidel (allow multiple processors)

- ixmdf (ixmdf output format):

0 = ascii (default)

1 = output in xmdf format

2 = input/output in xmdf

- iproc (multiple processor):

0 = one-processor (same as 1, default), n = n processors (only for isolv = 1)

- isea (swell and local sea output):

0 = no additional outputs,

1 = output additional swell.wav (for swell) and sea.wav (local wave) component files (These have the same format as the total wave field output file *.wav)

2 = output the total.wav (idential to the default *.wav) (This is for the steering run that will merge all individual *swsteer.wav files.)

3 = output the total.wav, swell.wav and sea.wav.

The following new files need to be added in the *.std:

- *.fric
- *.fref
- *.bref
- *.mud,
- *.wind
- *.runup

These are the same as friction.dat, forward.dat, backward.dat, mud.dat, wind.dat and runup.dat

Related Topics

- CMS-Wave Files

CMS-Wave Structure File

CMS-Wave structure files (*.struct) contain a line defining the number of cells that should be treated as structures in the model simulation, and then a list of these cells with special features assigned to them. These are specified in the SMS interface by selecting the cell and assigning cell attributes (CMS-Wave menu or right click on selected cell). These cells can represent:

- a floating breakwater
- a bottom mound breakwater
- a beach segment and the land adjacent to it
- jetties or seawalls
- underwater features such as reefs or submerged structures

A trench, submerged mound or structure can be added to the bed as features without modifying the input depth file. Each feature cell is described by four parameters in a line in the structure file. A sample file and table of the parameters is defined below.

File Format

number of structure cells

one row for each structure cell (each row includes the I and J indices of a selected output cell, the structure type and the modifier value

Sample File

4 /* there are 5 structure cells in this sample file */

10 15 1 3 /* cell (10,15) will be modified to have a depth of 3 meters */

23 41 2 0.5 /* model will calculate runup and overwash at cell (23,41). It will be assigned an elevation 0f 0.5 meters above sea level */

15 7 3 1.5 /* model will treat cell (15,7) as a floating breakwater with a draft of 1.5 meters to calculate transmission of waves under the structure */

16 22 4 3 /* model will treat cell (16,22) as a breakwater/seawall/mound with an elevation of 3 meters above sealevel */

Parameter definitions

kstruc	Description	Modifier value (cstruc)
1	add alternative depth without modifying input depth file	specify the new structure depth (assumed land if no depth specified)
2	for calculation of wave runup and overwash on beach face or structure and adjacent land	specify the elevation above mean water level for the cell. Input depth used in no value specified. Has no impact if elevation < 0)
3	for calculation of transmitted wave under a floating breakwater	specify the floating breakwater draft (skipped if not provided or value is < 0.05 m)
4	for a vertical wall breakwater	specify breakwater/structure elevation (input depth value used in not provided; immersed if value < 0.0)
5	for a composite or rubble-mound breakwater	

Related Topics

- CMS-Wave Files
- File Formats

5.6. GenCade

GenCade

The GenCade model is a next generation combination of previous long-term planform evolution of a beach models GENESIS (GENEralized Model for SImulating Shoreline Change [138]) and Cascade [139] .

GenCade is a regional model for calculating coastal sediment transport, morphology change, and sand bypassing at inlets and engineered structures. GenCade is developed by the U.S. Army Engineer Research and Development Center (USACE-ERDC), Coastal and Hydraulics Laboratory (CHL). The developers maintain documentation for the model, comments on the interface and steps for a sample application .

The GenCade model can be added to a paid edition of SMS.

GenCade Coverage

The GenCade coverage allows defining the geographical location of the characteristics and structures of a GenCade simulation. One or more coverages can be mapped to the simulation.

Functionality

GenCade simulates shoreline change relative to regional morphologic constraints upon which these processes take place. The evolution of multiple interacting coastal projects and morphologic features and pathways, such as those associated with inlets and adjacent beaches may also be simulated. The model supports responses to imposed wave conditions, coastal structures, and other engineering activity (e.g., beach nourishment).

Typical longshore extents and time periods of modeled projects can be in the ranges of one to 100 km and one month to multiple decades, respectively, and almost arbitrary numbers and combinations of groins, detached breakwaters, seawalls, jetties, and beach fills can be represented. GenCade simulates shoreline change produced by spatial and temporal differences in longshore sand transport. Shoreline movement such as that produced by beach fills and river sediment discharges can also be represented. The main utility of the modeling system lies in simulating the response of the shoreline to structures sited in the nearshore. Shoreline change produced by cross-shore sediment transport as associated with storms and seasonal variations in wave climate cannot be simulated; support of cross-shore processes are being considered for future versions of the model.

Capabilities of GenCade:

- Almost arbitrary numbers and combinations for groins, jetties, detached breakwaters, beach fills, and seawalls
- Compound structures such as T-shaped, Y-shaped, and spur groins
- Bypassing of sand around and transmission through groins and jetties
- Diffraction at detached breakwaters, jetties, and groins
- Coverage of wide spatial extent
- Offshore input waves of arbitrary height, period, and direction
- Multiple wave trains (as from independent wave generation sources)
- Sand transport due to oblique wave incidence and longshore gradient in height
- Wave transmission at detached breakwaters

Practical Notes For Using GenCade

Shoreline change models are not applicable to simulating a randomly fluctuating beach system in which no trend in shoreline position is evident. In particular, GenCade is not applicable to calculating shoreline change in the following situations that involve beach change unrelated to coastal structures, boundary conditions, or spatial differences in wave-induced longshore sand transport:

- Beach change inside inlets or in areas dominated by tidal flow
- Beach change produced by wave-generated currents
- Storm-induced beach erosion in which cross-shore sediment transport processes are dominant
- Scour at structures

Limitations of GenCade:

- No wave reflection from structures
- No direct provision for changing tide level
- Basic limitations of shoreline change modeling theory

Case Studies / Sample Problems

- Tutorials for learning to use GenCade in SMS are under development.

Related Topics

- 1D Grid Module
- GenCade modeling process
- GenCade graphical interface

External Links

- US Army Corps of Engineers Coastal & Hydraulics Laboratory GENESIS website [140]
- GENESIS, Report 1, Technical Reference [141]
- GENESIS, Report 2, Workbook and System User's Manual [142]
- User's Guide to the Shoreline Modeling System [143]
- A History of GENESIS Updates [144]
- Mar 2002 ERDC/CHL CHETN-II-45 Wave Transmission at Detached Breakwaters for Shoreline Response Modeling [145]
- Mar 1990 CETN-II-21 Computer Program: Genesis Version 2 [146]
- Cascade User Guide: [147]
- Cascade Theory and Model Formulation: [148]
- Inlet Reservoir Model (sub-model within Cascade) [149]

GenCade Files

GenCade makes use of the following input and output files.

Input

A GenCade simulation requires the following input files:

- GenCade Control File (*.gen) – The simulation file which contains links to other input files as well as the definition of all the structures and events and other model parameters.

- Initial Shoreline File (*.shi) – Contains the initial shoreline position geometry.
- Wave File (*.wave) – There must be one wave file for each wave gage. The file contains the wave events for the associated gage for all time spanning the simulation.

The following are optional input files:

- Regional Contour File (*.shr) – Contains the regional contour geometry.
- Water Level File (*.wl) – Contains time-dependent wave transmission data.
- Variable Resolution File (*.shdx) – Contains the grid density values for a non-uniform grid.

Output

The following files are created by GenCade during a simulation:

- Print File (*.prt) – A summary printed output file that reports on the status of the run. It is for user review.
- Shoreline Position Output File (*.slo) – Contains the output shoreline positions through the simulation.
- Offshore Contour File (*.off) – Contains the offset from the original shoreline positions. This is the same as the positions from the SLO file minus the initial shoreline positions.
- Net Transport Rate File (*.qtr) – Contains the transient, spatially varied transport rates at the specified output times during the simulations.
- Mean Net Annual Transport Files (*.mql/*.mqr/*.mqn) – Contains the annual averaged transport rates to the left, to the right and net. This is a integration of the components that make up the QTR.
- Inlet Shoal Volume File (*.irv) – Contains the time varying volumes of each of the shoals related to the inlet. One IRV file will be created for each inlet in the simulation.

Related Topics

- GenCade

GenCade Modeling Process

A GenCade model is set up in a globally referenced projection (such as UTM or State Plane).

Required Input Data

The model requires the following data for input:

- Grid frame – This is a geometric object that defines the domain of the simulation. The modeling engine operates on a straight line (1D grid). The grid frame defines the origin, orientation and extent of the simulation region. The grid frame should be oriented to align to the principal shoreline direction. Generally the grid frame is positioned just to the landward side of the shoreline. When looking along the shoreline in a direction parallel (or close to parallel) to the direction of the grid frame, the ocean should be but on the left and the shore to the right. Positions in the model (such as shoreline values) are referenced to this datum. The graphical interface will generate a grid that lies along the grid frame. h.
- Initial shoreline definition – This is series of points ((x,y) or (lat,lon)) that define a reference position for the coastline at the beginning of a simulation (in the model projection). SMS stores these points as an arc in a GenCade coverage.
- Wave conditions: The model supports multiple wave inputs at locations throughout the domain. These are mapped to a single cell in the grid. Wave conditions include the wave parameters (height, period, direction) at each time step in the simulation. The direction of each wave condition must be provided to the model in local shore normal coordinates.
- Other structure definitions to represent the structures (seawalls, groins and inlets) in the domain.
- Dredging/nourishment and bypass events as warranted by the events during the period being simulated.

Directions

The model performs all calculations in a local direction defined by the grid. This direction can be visualized relative to several reference points. Two references are defined here:

- The model references the grid direction as an azimuth angle from North. An azimuth of zero (0.0) directs the local grid to go from South to North (land to the East). The azimuth is measured in a clockwise direction, so an azimuth of 90 degrees defines a grid that goes from West to East (land on the South).
- The SMS interface relys more on graphical positioning, since the grid frame is defined interactively and displayed with an origin. However, the system uses a Cartesian angle reference with the X axis (East) equal to 0 degrees. The direction for this convention follows the right hand rule (CCW = positive). Therefore, a Cartesian angle of zero corresponds to the West to East (land on the South) azimuth of 90 degrees. Similarly, a Cartesian angle of 90 degrees is the same as an azimuth of 0 degrees.

The following steps illustrate the standard modeling process.

1) Define a GenCade coverage.
2) Create a grid frame defining the domain extents.
3) Define refinement points as desired to control grid resolution.
4) Define feature points at the location of wave buoys. Read or enter the wave data at each of these points.
5) Define arcs representing the various structures in the domain. Assign attributes to these arcs.
6) Define arcs representing the events in the simulation. Assign attributes to these arcs.
7) Map the conceptual model to a numeric model.
8) In the 1D Grid Module, assign the model parameters in the *GenCade Model Control* dialog .
9) Save the simulation and run the model.
10) Read solution files to visualize results.

Related Topics

- GenCade

5.6.a. GenCade Graphical interface

GenCade Graphical Interface

SMS provides a graphical interface that is designed to visualize the projects being created, easily modify project parameters, and view the solutions produced by the GenCade model. The GenCade interface exists in the 1D Grid Module .

A GenCade simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model. To set up a simulation, the modeler can work directly with the 1D grid, but it is much more convenient to work with the conceptual model using a GenCade coverage.

Work with an existing simulation by selecting the 1D Grid Module . If a grid has already been created for a GenCade simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the 1D Grid module active and set the model to GenCade.

The interface consists of the conceptual model tools in the Map Module and the grid specific options in the 1D Grid Module .

Map Module – Conceptual Interface

In the map module, a GenCade coverage (or coverages) can be created that include the information defining the grid and structures in the desired simulation.

Grid Frame

The grid frame defines the orientation and extent that will be occupied by the 1D grid. After defining the shorelines and any necessary structures, the grid should be set up. Manually draw the grid frame using the **Create 1D Grid Frame** tool. The GenCade grid frame is purple and has an arrow at one end. If a person followed the grid from the end to the arrow, the water should always be to the left and the land should always be to the right.

For example, if the GenCade grid was oriented from north to south, the water would be to the east (left) and the land would be to the west (right). The grid can be modified by clicking the **Select 1-D Grid Frame** tool and double-clicking on the square in the center of the purple grid line.

Alternately, the grid options can be changed by selecting the grid frame and use the right-click **Properties** command. The *Grid Frame Properties* window will open, and the *Origin X, Origin Y, Angle,* and *I size* can be modified. The *I size* is the length of the grid. *Angle* refers to the sign convention in the conceptual model which is degrees counterclockwise from the x axis.

This is different from the GenCade model convention (degrees clockwise from north). Therefore, once the map is converted to a 1D grid, the *Azimuth* for the grid will be a different value. The cell size can be constant or variable.

Feature Points

In a GenCade coverage, a feature point can be assigned to be:

- A generic place holder
- A wave gage – The attributes of this point include a time series of wave data .
- A refine point – This point will allow controlling the grid density in the area of the point.

Feature Arcs

The arcs in a GenCade coverage can be assigned a variety of attributes including:

- Geometric object – shoreline and contours.
- Structure – jetty, inlet, sea wall, or breakwater and the associated parameters.
- Event – bypass or beach nourishment events and their time ranges and parameters.

The conceptual model is converted to a numeric model using the **Map → 1D grid** command. This command can overwrite an existing simulation or simply add additional structures to an existing simulation.

1D Grid Module – Numerical interface

The tools in the grid module allow selecting individual structures graphically and change their parameters. The menu allows bringing up tables of all structures of a specific type, view their parameters, and make edits as needed.

GenCade Menu

When the GenCade model is active, the *GenCade* menu becomes available. See GenCade Menu for more information.

GenCade Tools

GenCade makes use of the tools in the 1D Grid Module

Running the Model

The GenCade Files are written automatically with the SMS project file or can be saved separately using the *File* | **Save GenCade** or *File* | **Save As** menu commands. See GenCade Files for more information on the files used for the GenCade run.

GenCade can be launched from SMS using the *GenCade* | **Run GenCade** menu command.

Related Topics

- GenCade Model Control Dialog
- GenCade Result Visualization

GenCade Arc Attributes

The attributes that can be associated with for feature arcs in a GenCade coverage are specified through the *GenCade Arc Attributes* dialog. This dialog can be reached by selecting the arc and either right-clicking and selecting **Attributes** from the right-click menu or selecting **Attributes** command from the *Feature Objects* menu.

Attributes that can be specified for each arc include:

Arc Type

- *Generic* – Place holder in the conceptual model. Ignored by the simulation.
- *Initial Shoreline* – The geometry of this arc defines the starting shoreline position.
- *Reference Line* – This type is not currently used.
- *Regional Contour* – The geometry of this arc defines the regional geometric shape.
- *Breakwater* – Detached structure, generally offshore. In addition to location, specify depths at the ends of the structure and transmission attributes.
- *Seawall* – The geometry defined by this arc delineates the furthest landward point the shoreline can erode to in that region.
- *Groin* – This arc defines a shore perpendicular structure. Also the arc specifies the structure permeability and its diffractive attributes.
- *Inlet* – This arc actually defines the location of the mouth of the inlet. For each inlet, define an inlet name, initial and equilibrium volumes of the shoals associated with that inlet, bypass coefficient and defines dredging events that occur during the simulation. Several other arcs can define other components of the inlet.
- *Left Jetty of Inlet* – This arc defines a groin that is part of an inlet complex.
- *Right Jetty of Inlet* – This arc defines a groin that is part of an inlet complex.
- *Inlet Attachment Bar* – This arc defines the location of the attachment bar that will form related to an inlet complex.
- *Bypass Event* – This arc defines the beginning and ending points of a bypass event. The arcs also defines the time ranges and the bypass rates that will occur during the simulation between these points.
- *Beach Fill Event* – This arc defines the location of a beach nourishment event. Define with the arc the time ranges over which material will be added to the beach and the amount of berm width that will be added with each time range.

Related Topics

- GenCade
- Feature Objects Menu

GenCade Events

The GenCade simulation can also include timed events that impact the shoreline. These include dredging events at an inlet, beach fill (nourishment) events, and bypassing events.

Beach Fill

Parameters for characterizing beach fills include begin and end time, location, and width. To define a beach fill, proceed with the following steps:

1) Select the arc associated with the beach fill or select the menu option *GenCade* | **Edit Beach Fill** .
2) The GenCade *Beach Fill* dialog will appear allowing the entry of beach fill parameters.

Bypassing Event

Parameters for characterizing bypassing events include begin and end time, location, and bypassing rate. To define a bypassing event, proceed with the following steps:

1) Select the arc associated with the bypass or select the menu option *GenCade* | **Edit Bypassing** .
2) The GenCade *Bypassing* dialog will appear allowing the entry of bypassing parameters.

Dredging Events

Dredging events are created along inlets. See Dredging Events on the GenCade Structures article.

Related Topics

- GenCade

GenCade Menu

The *GenCade* Menu has the following commands:

Edit Grid...

Opens a *Grid Frame* dialog.

Edit Water Level...

Opens the *Water Level Events* table.

Edit Wave Data...

Opens the *Wave Gages* table.

Filter Wave Data...

(This tool was part of the GENESIS interface, but is currently not functional for GenCade)

Edit Breakwaters...

Opens the *Detached Breakwaters* table.

Edit Seawalls...

Opens the *Seawalls* table.

Edit Groins

Opens the *Groins* table.

Edit Inlets

Opens the *Inlets (Resvoir Model and Jetties)* table.

Edit Beach Fills...

Opens the *Beach Fill* table.

Edit Bypassing...

Opens the *Bypassing* table.

Model Control...

Can be used to set beach conditions, lateral boundary conditions and general simulation options. See GenCade Model Control Dialog for more information.

Model Check

Starts the model checker .

Run GenCade...

This command will start the GenCade Model.

Related Topics

- GenCade
- GenCade Events
- GenCade Structures
- Wave Gages

GenCade Model Control Dialog

The *Gencade Model Control* dialog is used to set beach conditions, lateral boundary conditions and general simulation options. This document highlights the more commonly used options. Refer to the GenCade web site for a more detailed description of how these parameters affect the model results.

Model Setup Tab

The following parameters are specified in the *model setup* tab:

- *Simulation*
  - *Title* – Title of simulation run.
  - *Full print output* – Path and name of printed output file.
- *Computational Time*
  - *Start Date* – Simulation start date.
  - *End Date* – Simulation end date.
  - *Time Step* – Model time step in hours.
  - *Recording Time Step* – Recorded model time step in hours.
- *Print Dates* – Dates to save simulated shoreline. Use the **Add** button to create dates and the **Remove** button to delete any unwanted dates.

Beach Setup Tab

The following parameters are specified in the *beach setup* tab:

- *Sand and Beach Data*
  - *Effective Grain Size* – Medium sediment grain size in millimeters.
  - *Average Berm Height* – In meters.
  - *Closure Depth* – In meters.
- *Longshore Sand Transport Calibration Coefficients*
  - *K1* – 0.1<K1<1
  - *K2* – 0.5K1<K2<1.5K1

Seaward BC Tab

The following parameters are specified in the *seaward boundary condition* tab:

- *Input Wave Adjustments* – This section contains the following options:
  - *Height Amplification Factor*
  - *Angle Amplification Factor*
  - *Angle Offset*
- *Wave Components to Apply* – options include "Primary (1)" or "Primary & Secondary".
- *Number of Cells in Offshore contour Smoothing Window* – Default value is 11, but it is suggested that this number range between 11 and 101.

Lateral BC Tab

The following parameters are specified in the *lateral boundary condition* tab for the left and right lateral boundary condition:

- *Type* – Determines one of three boundary types:
  - "Pinned" – Boundary will not move from the initial shoreline position.
  - "Moving" – Represents the amount of shoreline change at a boundary over a specified period of time.
    - *Shoreline Displacement Velocity* – Shoreline can be displaced over the "Simulation Period", "Day", or "Time Step".
  - "Gated" – Bounded with a groin. Requires that a groin exist and must be located in cell 1.
    - *Length of Groin from Shoreline to Seaward Tip*

Adaptive Time Steps Tab

The following parameters are specified in the adaptive time steps tab:

- *Use adaptive time steps* – If turned on, allows the option to use the default values or set custome values.
- *Use defaults* – If turned on, the default values will be used. If turned off, the following can be set:
  - *Threshold minimum*
  - *Threshold maximum*
  - *Number of days stable*
  - *Stability minimum*
  - *Stability maximum*

Related Topics

- GenCade

GenCade Result Visualization

The interface supports three types of plots to visualize the results of the model. These include:

- Shoreline plots on the grid. These plots are available in the system after reading in the shoreline output (*.slo) file. The offset file (*.off) contains the same data as a displacement quantity and can be read and viewed just like the shoreline output file. Select the desired dataset which is then plotted in the graphics window right over the initial shoreline. Display options also allow the display of the minimum and maximum shoreline positions during the simulation. When this file is read, SMS adds several datasets to the project explorer representing:
    - The shoreline position throughout the simulation time. This dataset includes a time step for each output time at each cell in the grid.
    - The change in shoreline position from the initial shoreline (since the beginning of the simulation). This dataset is also transient.
    - The average rate of change of the shoreline during the simulation (single value at each cell of the grid).
- Two dimensional plot of spatially variable datasets. In addition to the *.slo solution file, also instruct SMS to load the sediment transport rate (*.qtr) solution file and the mean transport rate files (*.mqr, *.mql and *.mqn). The sediment transport rate file contains the net transport computed for each cell in the grid at each time step. The mean transport rates contain annual average transport rates in the right, left, and net directions. This dataset, along with any of the other spatially variable datasets may be visualized in their own plot window. Display options allow selecting one or more of the datasets to be plotted. SMS can be instructed to plot the "active" dataset which results in the plot being updated as the active dataset is changed in the project explorer. In addition, multiple time steps may be selected (or the plot can use the "active" time step). SMS will assign a symbol to each dataset and cycle through line styles for each displayed time step. To create this plot:
    1) Load at least one solution file into the project.
    2) Click on the **Plot Wizard** tool.
    3) Select the *Shoreline* option and click the **Next** button.
    4) Choose the plot options to select the dataset and time steps to be plotted.
- Two dimensional plot of the Inlet Volume transport. The GenCade model outputs a solution file for each inlet defined in the domain. These solution files (*.irv files), contain sediment volumes and transport rates (in and out) for each of the structures/shoals related to the inlet including the ebb shoal, the flood shoal, and attachment and bypass bars both upstream and downstream of the inlet. The SMS package will search the directory for inlet solution files matching the specified inlet names and populate a list of functions that can be plotted for the selected inlet. Each function will be assigned a unique color to be plotted. Select a time range to plot the volume (or transport rates). To create this plot:
    1) Run a simulation that includes an inlet to generate at least one inlet solution file (*.irv).
    2) Click on the **Plot Wizard** tool.
    3) Select the *Inlet TS* option and click the **Next** button.
    4) Choose the plot options to select the inlet, functions and time range to be plotted.

It is possible to interact with the plot by dragging a zoom box over an area or right-clicking on the plot. Right-clicking invokes a menu to select the plot options, frame the data, set the plot attributes, or export the plot data in tabular or image form.

Related Topics

- GenCade

GenCade Structures

GenCade has the capability to model the structures listed below. The *GenCade* menu includes commands to view a table of the structures of a specific type included in a simulation. Some structures can also be selected interactively if they have a geographic attribute such as length.

Breakwaters

Parameters for characterizing breakwaters include start and end location, depth, and transmission coefficient. To define a breakwater, proceed with the following steps:

1) Select the **Create Breakwater** tool and click with the left mouse button to specify the start and end location.
2) Select the menu option *GenCade* | **Edit Breakwaters** . The GenCade *Detached Breakwaters* dialog will appear allowing the entry of breakwater parameters. At this point, verify that the parameters are correct. Further information concerning the selection of breakwater parameters is provided in the GenCade model documentation.

Seawalls

Parameters for characterizing seawalls include start and end location. To define a seawall, proceed with the following steps:

1) Select the **Create Seawall** tool and click with the left mouse button to specify the start and end location.
2) Select the menu option *GenCade* | **Edit Seawalls** . The GenCade *Seawalls* dialog will appear allowing the entry of seawall parameters. At this point, verify that the parameters are correct.

Groins

Parameters for characterizing groins include length, permeability, diffraction, and depth. To define a groin or jetty, proceed with the following steps:

1) Select the **Create Jetty/Groin** tool and click to specify the jetty or groin location. The groin or jetty will be created perpendicular to the grid.
2) Select the menu option *GenCade* | **Edit Groins** . The GenCade *Groins* dialog will appear allowing the entry of groin and jetty parameters. At this point, verify that the parameters are correct. Further information concerning the selection of groin and jetty parameters is provided in the GenCade model documentation.

Parameters that can be edited for groins include:

- Cell Index
- Length
- Permeability
- Diffracting
- Seaward Depth

Inlets

To create an inlet, create a feature arc, right-click on it, select **Attributes** and change the attribute to *Inlet* . Inlets have a name, a left and right bypass coefficient, cell positions, shoal volumes, jetties, and dredging options.

Cell Positions

Cell beginning and ending positions can specified for the inlet and its left and right bypass. These can be set by creating attachment arcs by the inlet before mapping to a 1D grid. If there are two inlets with one attachment arc in between, the attachment arc will be associated to the inlet closest to the arc.

Shoal Volumes

The initial and equilibrium shoal volumes can be specified for the ebb, flood, bypass (left and right) and attachment (left and right) shoals.

Jetties

The jetties to the left and right of the inlet can be given a length, permeability, and a seaward depth. Also, whether the jetty exists and is diffracting can also be specified.

Dredging Events

Dredging events can have a beginning and ending date, which shoal is to be mined, and the amount to be removed from the specified shoal between the beginning and ending date.

Related Topics

- GenCade

Wave Gages

Wave gages provide boundary condition information to a GenCade simulation. The wave height, period and direction influence the transport of material on the shoreline and the shoreline morphology. The GenCade model allows defining multiple wave gages, providing for spatially varied driving force.

The wave gage is created either as an *Attribute* on a feature point in the GenCade coverage, which will be mapped to a specific cell in the grid, or it may be specified directly on the grid. For each wave gage, specify a depth and wave events. Each wave event has a date and time, a H0, a period, and a direction. The model computes the wave conditions for each cell in the grid by interpolating the data from the neighboring wave gages.

Wave gages specified in a GenCade coverage will be displayed at the feature point in the coverage, and will appear when the 1D grid (not grid frame) is created on the associated cell in the grid.

If attributes of a wave gage are changed on a feature point, these changes do not impact the simulation unless the attributes are mapped to the grid again (**Map → 1D grid**). Wave gages can be edited using the **Edit Wave Data..** option from the *GenCade* menu after the grid is created and still take effect.

Wave Events

Wave events can be edited by clicking on the **Data...** button when editing the wave gage. This can be done for either a gage on a 1D grid, or a gage at a feature point.

The direction of a wave, for a given wave event, for a given wave gage may be specified in a meteorological, oceanographic, or Cartesian convention. Also, the wave direction can be specified in a shore normal convention as long as a 1D grid or 1D grid frame is present in the coverage which contains the wave gage. The default convention for wave gages is meteorological.

Wave Data for GenCade

Wave conditions, including significant wave height, peak period, and direction, drive the transport forces that impact coastal morphology. GenCade requires wave data for at least one location in the domain, and supports options for specifying wave data at multiple locations in the domain. Each wave gage includes:

- A location – (X,Y) coordinate as a feature point and Cell ID as an input to the model.
- A depth – The water depth at the gage.
- A time series of wave data events – Obtain these events either from a buoy or from a local wave transformation model.

Issuing the command to edit wave data invokes the *Wave Gages* dialog. This dialog lists the gauge cell IDs and the depths. It also includes a button for each gage to view/edit the wave data at that gage.

Clicking on the **Data ...** button for any of the gages brings up the *Wave Events* dialog. This dialog displays the time, height, period and direction (in shore normal coordinates) for the gage. The data is in a spread sheet, so it may be copied and pasted from other spread sheet data. There is also an **Import** button that can be used bring up the *Import Wizard* to bring in wave events data stored in a text file. The data is saved and passed to the model in the wave file.

Modify 1D Wave Line

Related Topics

- GenCade

5.7. STWAVE – Steady State Spectral Wave

STWAVE

STWAVE	
Model Info	
Model type	Model for nearshore wind-wave growth and propagation.
Developer	Jane Smith
Web site	STWAVE web site
Tutorials	General Section • Data Visualization • Observation Models Section • STWAVE Several sets of sample problems and case studies are available. These include: • Model Validation cases from the STWAVE website

STWAVE (STeady State Spectral WAVE) is a steady-state, finite difference, spectral model based on the wave action balance equation. STWAVE is written by the U.S. Army Corps of Engineers Waterways Experiment Station (USACE-WES).

The STWAVE model can be added to a paid edition of SMS.

Functionality

STWAVE simulates depth-induced wave refraction and shoaling, current-induced refraction and shoaling, depth- and steepness-induced wave breaking, diffraction, wave growth because of wind input, and wave-wave interaction and white capping that redistribute and dissipate energy in a growing wave field. The purpose of STWAVE is to provide an easy-to-apply, flexible, and robust model for nearshore wind-wave growth and propagation. Recent upgrades to the model include wind, surge and friction fields (spatially varied). Also, wind and surge fields can be temporally varied. The method of analysis used by the STWAVE code along with the file formats and input parameters are described in the STWAVE documentation. SMS supports both pre- and post-processing for STWAVE.

The new full-plane version of STWAVE is not a replacement for the half-plane version, but a supplement. The half-plane version will always have an advantage of substantially lower memory requirements (~ two orders of magnitude) and faster execution. The half-plane limitation is generally appropriate for nearshore coastal applications, with the exception of enclosed or semi-enclosed bays, estuaries, and lakes where seas and swells may oppose each other or there is no clear “offshore” direction. The full-plane version allows wave input on all boundaries and wave generation from all directions.

Saving STWAVE

When completing the *File* | **Save As...** command, the following files get saved in the *.sms

- *.mat referenced to new save location
- stw_grds.h5 referenced to new save location
- spec_grds.h5 referenced to new save location
- *.grd referenced to new save location
- *.ctl referenced to new save location
- *.sol referenced to a folder in the new save location

Using the Model / Practical Notes

- A grid for use with STWAVE is created and edited in SMS using the Map Module.
- The modeling parameters required by STWAVE are generated and applied to the mesh using commands grouped in the STWAVE menu.
- Post processing of solution data generated by STWAVE is done using the generic visualization tools of SMS.
- Wind can be entered in the STWAVE model control as either a constant value or by specifying an existing Cartesian Grid dataset.
- STWAVE requires metric units. All data in SMS needs to be in metric units before running STWAVE.
- Water depths are defined as positive numbers and land elevations are negative numbers.

Graphical Interface

The STWAVE Graphical Interface contains tools to create and edit a STWAVE simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to STWAVE. If a grid has already been created for a STWAVE simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to STWAVE. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian grid menus and tools augmented by the STWAVE Menu . See STWAVE Graphical Interface for more information.

STWAVE Files

Here are tables of some of the available input and output files for STWAVE

- For more information on these files see page 16 of the manual .

SMS Input Files

Name	Description
GRD	Grid File
H5	Spectral Grid HDF5 Wave File
H5	STWAVE Grid HDF5 Depth File
H5	Scatter HDF5 Velocity File
DIS	Discretization File
MAP	Boundary Map File
SMS	SMS Project File

SMS Generated STWAVE Input Files

Name	Description
DEP	Cell Depth File
ENG	Spectral Dataset Energy File

STWAVE Output Files

Name	Description
912	File
CMPCT	Full-Plane Mode Compact Output File
Log.OUT	Log Output File
Nest.OUT	Nesting Application Energy Spectra Grid File
Obse.OUT	Observation Output File
Spatial.OUT	Spatial Output HDF5 File
Selh.OUT	Local Paramater Summary Spatial Dataset
SIM	Simulation File
TP.OUT	Peak Wave Period File
Wave.OUT	Wave Height, Period, Direction File

External Links

- STWAVE 6.0 User's Manual Sep 2011
- Aug 2007 ERDC/CHL CHETN-I-76 Modeling Nearshore Waves for Hurricane Katrina [150]
- Aug 2007 ERDC/CHL CHETN-I-75 Full-Plane STWAVE with Bottom Friction: II. Model Overview [151]
- Sep 2006 9th International Workshop On Wave Hindcasting and Forecasting Jane McKee Smith Modeling Nearshore Waves For Hurricane Katrina [152]
- Mar 2006 ERDC/CHL CHETN-I-71 Full Plane STWAVE: SMS Graphical Interface [153]

- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D (now know as CMS-Flow) and STWAVE [154]
- Jun 2002 ERDC/CHL CHETN-I-66 Grid Nesting with STWAVE [155]
- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [156]

  Please see this forum post for an explanation of ADCIRC and STWAVE steering
- Sep 2001 ERDC/CHL CHETN-I-64 Modeling Nearshor Wave Transformation with STWAVE [157]

Grid Nesting

Grid nesting refers to the ability to take output at specified locations from one grid to use for boundary conditions on another. The source grid as the parent grid and the grid using the boundary conditions as the child grid. This approach is often used in order to cover a large area with a coarse grid with a more refined grid in a specific area of interest.

Within SMS, set up nesting for STWAVE using a STWAVE parent grid or a WAM parent grid. The functionalities are quite similar with some minor changes specific to each type of setup.

Nesting STWAVE → STWAVE

Nesting can be done with full or half plane grids but the parent and child grids must be consistent (SMS can't nest a half-plane STWAVE grid within a full-plane grid). Specify the nesting options by clicking *STWAVE* | **Generate Nesting Points** command.

Requirements

The following conditions must be met to create nested grids:

1) The cartesian grids being used must be composed of square cells
2) The offshore boundary of the child grid should be entirely contained in the parent grid.

Nesting Dialog

The *Nesting* dialog specifies the parent and child grids with the following options:

- *Parent grid* – Clicking the **Select** button will bring up a *Select Tree Item* dialog where the parent grid can be designated.

- *Child grid* – Clicking the **Select** button will bring up a *Select Tree Item* dialog where the child grid can be designated.
- *Child grid edge*
- *Spacing Options*
  - *Space every*
  - *Number of sites per grid boundary*

When doing STWAVE → STWAVE nesting, some of the parameters in the child grid must match those in the parent grid. These include whether or not the model is using time steps, whether the model is using a reference time, whether the model is full or half plane, and the identifiers for each of STWAVE snaps (time steps or cases). These options will be disabled in SMS once having chosen to do nesting.

Older Options

The *Nesting Options* dialog is used in SMS 11.2 and older.

- **Select Nesting Grid –** This command indicates that the current grid should obtain input spectral values from a parent grid. By selecting this option the current grid is nested in the grid selected in the combo box below.
- **Type of Boundary Interpolation** – Choose to use either linear boundary interpolation or morphic boundary interpolation. The morphic interpolation method was developed to preserve the shape of the directional distribution. It is appropriate for climatic wave transformation studies where a parametric spectral shape is applied based on wave parameters.

Nesting WAM → STWAVE

Even though the output spectra from WAM is always full-plane, a full or half plane STWAVE grid may be nested within a WAM grid. Set the nesting options by clicking on the options button in the

dialog. Since WAM models are always run at specific date/times, child STWAVE models always use time steps with reference times.

Options

- **WAM Simulation** – The WAM simulation which will provide the input spectra.
- **WAM Grid** – The WAM grid within the simulation to extract spectra.
- **Start time –** The first time to run STWAVE. This time must be an output time specified in the WAM grid options.
- **End time –** The last time to run STWAVE. This time must be an output time specified in the WAM grid options.
- **Interval –** The snap interval must be set so each snap will hit a WAM output time. For example if WAM data was being output every hour, the interval may be 1 hour, 2 hours, or 3 hours but not 1.5 hours.
- **Type of Boundary Interpolation** – Choose to use either linear boundary interpolation or morphic boundary interpolation. The morphic interpolation method was developed to preserve the shape of the directional distribution. It is appropriate for climatic wave transformation studies where a parametric spectral shape is applied based on wave parameters.

The start time, end time, and interval will determine the number of snaps used by STWAVE and the times associated with each.

Spectral Sites

Regardless of the type of nesting that is going to take place, spectra need to be provided to STWAVE along the boundaries with input spectra (see Spectral Events Dialog). Both WAM and STWAVE support the ability to output computed spectra at specified locations. These locations can be specified manually using cell attributes but preferably are automatically generated. To automatically generate these output locations, click on the spectral sites button in the model control dialog (the nesting options should have already be set). Be sure that if using a full-plane model, the sides which will have spectra input have already been identified. The dialog that comes up will allow choosing between different options to control the number of nesting cells along the offshore boundary. The more nesting cells there are, the better the project can capture differences along the boundary. However, more nesting cells require more disk space and time to read and write the files.

STWAVE Graphical Interface

The STWAVE graphical interface includes tools to assit with creating, editing, and debugging a STWAVE model. The STWAVE interface exists in the Cartesian Grid Module .

Model Control

The *STWAVE Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The STWAVE files are written automatically with the SMS project file or can be saved separately using the *File* | **Save STWAVE** or *File* | **Save As** menu commands. See STWAVE Files for more information on the files used for the STWAVE run.

STWAVE can be launched from SMS using the *STWAVE* | **Run STWAVE** menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *STWAVE* | **Model Check** menu command.

STWAVE Menu

See STWAVE Menu for more information.

Visualizing Results

The solution files for STWAVE are XMDF files. These files can be opened in the normal SMS interface. To view the observation or nesting spectral output simply open the files (outside of the *Spectral Energy* dialog), open the *Spectral Energy* dialog, and select the spectral grid. This will open the spectral grid for viewing in the *Spectral Energy* dialog. Make sure to set the original grid back before leaving the dialog to ensure the model does not change.

Related Topics

- Create a 2D Cartesian grid

STWAVE Menu

The items in the *STWAVE* menu in the Cartesian Grid Module are described below:

STWAVE Menu Commands

The *STWAVE* menu has the following commands:

- **Assign Cell Attributes** – Opens the *STWAVE Cell Attributes* dialog.
- **Generate Nesting Points** – Opens the *Nesting* dialog.
- **Model Check** – Starts the SMS model checker.

- **Model Control** – Opens the *STWAVE Model Control* dialog.
- **Run STWAVE** – Launches the finite element analysis engine.

Obsolete Menu Commands

The following commands are no longer available in current versions of SMS.

- **Spectral Energy** – Opens the *Spectral Energy* dialog.

Assign Cell Attributes Command

The *STWAVE Cell Attributes* dialog is used to assign a cell type to selected Cells. It is opened using the menu command *STWAVE* | **Assign Cell Attributes** . The following cell types can be assigned:

Default

Cells are no longer assigned as Land or Ocean types but are assigned as Default where their elevation values are used to simulate whether flow exists or not. If no elevation data is available for part of the grid area, it is important to specify an extrapolation value of a negative number. Extrapolation values can be specified in the *Interpolation* dialog while doing **Map → 2D Grid** .

Monitoring Station

Spectral output is generated for cells designated as monitoring station cells.

Nesting output

Spectral output is generated for nesting output cells cells to be used as input for a nested child grid .

Displaying Cell Attributes

The symbols and colors used to identify the attributes assigned to cells can be changed in the STWAVE *Display Options* .

Run STWAVE Command

After a simulation has been read or saved, SMS can launch the finite element analysis engine. To run STWAVE :

1) Select *STWAVE* | **Run STWAVE** . A dialog appears showing the engine that will execute.
2) If the executable program is the wrong version, or if given the message that it was not found, then click the file browser icon and choose the correct version of the program that should run.
3) Click the **OK** button or press the *ENTER* key.

A window will appear which displays various information as STWAVE runs. Normally it is not necessary to type anything in this window. If the window prompts for file names, an updated version of the STWAVE model executable is needed. When STWAVE has finished, a prompt will ask to press the *ENTER* key. If the window goes away before doing this, STWAVE encountered a problem and crashed.

Related Topics

- STWAVE

STWAVE Model Control

The **Model Control…** command in the *STWAVE* Menu opens the *STWAVE Model Control* dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

Grid Definition

The grid definition section of the model control reports defining characteristics of the grid including:

- Cell size
- Number of rows and columns
- X, Y origin
- Angle of rotation

Boundary Condition Source

The boundary condition for STWAVE consists of one or more energy spectra entering on one or more open edges of the grid. Traditionally, specify a single spectra that is assigned to all of the cells on the offshore edge. In this situation specify one spectra for each wave case or time snap being simulated. Alternately, the input spectra can be interpolated to the cells on the offshore edge(s) from a parent STWAVE or WAM grid.

- **Source combo box** – This combo box lists the types of spectral sources. Choose from "Spectral Coverage", "STWAVE run", and "WAM run". If the option is "Spectral Coverage", provide the spectra in a spectral coverage as well as a definition for the spectral grid to be used when exporting STWAVE simulation files. The grid definition can be specified by selecting the **Spectral Grid...** button. Both of the other options indicating that spectra will be coming from a parent grid and the associated options and spectral sites buttons become enabled. If an option is selected that is not valid (i.e. choosing an "STWAVE Run" when only a single grid exists in the project so not parent grid exists), SMS generates a message indicating the option is invalid and resets the source.
- **Options...** – This button brings up a dialog which allows specifying information for using output from a STWAVE or WAM parent grid as spectral input for this grid. When a parent STWAVE grid is desired, specify which grid is the parent and what type of spectral interpolation should be used.
- **Spectral sites...** – This button invokes a dialog that creates nesting sites in the parent grid. Spectral output sites for nesting may also be specified explicitly by selecting cells in the parent grid and assigning attributes for those cells. See grid nesting for more information.

Settings

- **Boundary Control** – Opens the *Spectral Events* dialog used to specify the STWAVE Boundary Conditions.
- **Output Control** – Opens the *STWAVE Output Control* dialog. (see below)
- **Iterations Control** – Opens the STWAVE *Iterations Control* dialog. (see below)
- **Full Plane** – Consult the STWAVE model documentation for more information.
- **Half Plane** – Consult the STWAVE model documentation for more information.
- **Depth Type** –
- **Source Terms** – Specifies whether STWAVE should generate waves using wind and the input spectrum or whether to use the wave spectrum.
- **Current Interaction** – Specifies whether currents will be used as input to the model.
- **Bottom Friction** – Specifies which friction type is to be used, if any.
- **Surge Fields** – Specifies whether surge fields are being used, or a constant tidal offset.
- **Wind Fields** – Specifies whether wind fields (spatially varied wind) are being used, or a constant wind magnitude and direction.
- **Number of Iterations** – Number of iterations to run. (Full Plane Only).
- **Select buttons** – The select buttons by next to the current, bottom friction, surge, and wind options are used to specify a dataset on the grid to use as the desired dataset. These buttons open up the *Select Dataset* dialog. These datasets must be created prior to entering the model control dialog.

- **Ice Fields** – Specifies whether an ice dataset is being used. This is a scalar dataset that defines the percent of ice in each grid cell.
  - **Ice Threshold** – If an ice dataset is being used, this value is needed to specify to threshold concentration level. Any cells with a percentage of ice higher than this threshold will be considered inactive by STWAVE.

STWAVE Output Control

The *STWAVE Output Control* dialog is used to specify the solutions files STWAVE should output.

The following datasets are always written:

- **Height**
- **Period**
- **Direction**
- **Wave Vector** (magnitude of height with wave direction)
- **1/fma** – Period that represents the inverse of the spectral peak frequency redefined for local wind growth cases

The options include:

- *Radiation stresses* – If this is on, radiation stresses will be calculated and output into datasets
- *Breaking* – This will output a dataset representing the wave breaking. For full-plane, this can be no indices (off) or write indices which gives a value of 1 where breaking occurs and 0.0 otherwise. Half-plane has the full-plane options and has an additional option to calculate energy dissipation. This will give a dataset of energy dissipation.
- *Output type* – SMS supports the ability to write ASCII files, XMDF files or both. ASCII files refer to STWAVE version 6 global dataset files and XMDF files are datasets written using the XMDF library built upon HDF5. XMDF is the recommended file format because they are much faster to read/write. XMDF is only available on windows platforms.

STWAVE Interations Control

The STWAVE *Iterations Control* dialog is reached through the *STWAVE Model Control* dialog. Here the number, stop value, and stop percent of iterations can be set. The dialog contains the following options:

Iterations Parameters

- Maximum number of initial iterations
- Initial iteration stop value
- Initial iteration stop percent
- Maximum number of final iterations
- Final iteration stop value
- Final iteration stop percent

Using Datasets as Input

Datasets may be used as input to STWAVE to represent spatially varied currents, bottom friction (either manning's or JONSWAP), surge, or wind. Bottom friction cannot vary from snap to snap. The other dataset types can have different values for each snap.

Datasets are chosen within the model control by clicking on the **Choose** button to the right of the dataset choice. The dataset values for each timestep will be the values for the time that the time step occurs. The time steps for the dataset chosen do not need to match the time steps specified for the simulation. The dataset values passed to STWAVE can be interpolated from a dataset with values at different time steps than is being used by the model.

If the simulation is not using time steps, the datasets are chosen in the *Spectral Events* dialog . In this situation, a steady state dataset is chosen for each snap.

Related Topics

STWAVE Menu

5.8. WAM – Wave Prediction Model

WAM

The global ocean WAve Model (WAM) is a third generation wave prediction model. WAM predicts directional spectra as well as wave properties such as significant wave height, mean wave direction and frequency, swell wave height and mean direction, and wind stress fields corrected by including the wave induced stress and the drag coefficient at each grid point at chosen output times.

The model is continually updated to incorporate the latest results of research. The verification has been carried out in three areas National Oceanic and Atmospheric Administration (NOAA) moored buoys are available on the global Telecommunications System (GTS). It is hoped that the buoys chosen will allow the identification of both successes and failures in WAM model physics and will minimize shortcomings due to sub-grid scale effect.

The WAM model can be added to a paid edition of SMS.

Features

The present version of WAM makes the following assumptions:

- Time dependent wave action balance equation.
- Wave growth based on sea surface roughness and wind characteristics.
- Nonlinear wave and wave interaction by Discrete Interaction Approximation (DIA).
- Free form of spectral shape.
- High dissipation rate to short waves.

Graphical Interface

The WAM Graphical Interface contains tools to create and edit a WAM simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to WAM. If a grid has already been created for a WAM simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian Grid Module active and set the model to WAM. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian Grid Module menus and tools augmented by the *WAM* Menu. See WAM Graphical Interface for more information.

External Links

- Influence of Marsh Restoration and Degradation on Storm Surge and Waves
- Dynamics and modelling of ocean waves By G. J. Komen, L. Cavaleri, M. Donelan, S. Hasselmann, P. A. E. M. Janssen

WAM Map to Raster Utility

The *Map to Raster* utility has been updated to allow for conversions between big endian to little endian. Most machines are little endian, therefore use the utility as before. However, if the InOut/Map files were created from a machine that is big endian, convert to little endian. To convert: copy the InOut/Map files to a little endian machine. Then from the command line run map_to_rast.exe passing "big_endian" as an argument. It will look something like this: c:\>map_to_rast.exe big_endian. Other command line arguments can also be used outlined below:

- native

```
Specifies that unformatted data should not be converted.
```

- big_endian

```
Specifies that the format will be big endian for integer data and big
endian IEEE floating-point for real and complex data.
```

- cray

```
Specifies that the format will be big endian for integer data and CRAY*
floating-point for real and complex data.
```

- fdx (Linux, Mac OS X)

```
Specifies that the format will be little endian for integer data, and VAX
processor floating-point format F_floating, D_floating, and X_floating for
real and complex data.
```

- fgx (Linux, Mac OS X)

```
Specifies that the format will be little endian for integer data, and VAX
processor floating-point format F_floating, G_floating, and X_floating for
real and complex data.
```

- ibm

```
Specifies that the format will be big endian for integer data and IBM*
System\370 floating-point format for real and complex data.
```

- little_endian

```
Specifies that the format will be little endian for integer data and
little endian IEEE floating-point for real and complex data.
```

- vaxd

```
Specifies that the format will be little endian for integer data, and
VAX* processor floating-point format F_floating, D_floating, and H_floating
for real and complex data.
```

- vaxg

```
Specifies that the format will be little endian for integer data, and VAX
processor floating-point format F_floating, G_floating, and H_floating for
real and complex data.
```

Related Topics

- WAM

5.8.a. WAM Graphical Interface

WAM Graphical Interface

The WAM interface supports the wave model WAM .

WAM models are built by creating WAM grids, WAM simulations (right-click in blank area of project explorer), and dragging the WAM grids you want to use into each WAM simulation. There are options specific to each grid in a simulation as well as the simulation as a whole (model control).

WAM has the ability to use nested grids to improve resolution in an area of interest.

WAM Menus

The WAM Model makes use of a variety of menus depending on the active item.

WAM Grid Menu

For grids created from a WAM coverage in the Map module, the *WAM* menu will appear when the grid is active. The following option are under the *WAM* menu:

- **Assign Cell Attributes** – Brings up a *Cell Attributes* dialog where a spectra site can be assigned to the cell.
- **Model Check** – Starts the *Model Checker* .
- **Model Control** – Brings up the *Grid Options* dialog for WAM.

WAM Grid Right-Click Menu

Right-clicking on a WAM grid item in the Project Explorer contains the standard right-click menu commands as well as the following additional commands:

- **Options** – Brings up the *Grid Options* dialog.
- **WAM to STWAVE** – Brings up the *WAM to STWAVE Options* dialog.

WAM Simulation Menu

The WAM simulation menu is accessed by right-clicking on the WAM simulation in the Project Explorer. It has the following options:

- **Delete** – Removes the simulation from the project.
- **Duplicate** – Creates a copy of the selected simulation.
- **Rename** – Will allow the name of the simulation to be changed.
- **Export WAM Files** – Generates the WAM files necessary for the model run.
- **Launch WAM** – Will start the WAM model wrapper .
- **Save Project, Export, and Launch WAM** – Performs the process of saving the WAM project, exporting the model files, and running the WAM model all at once.
- **Assign Cell Attributes** – Brings up the *Cell Attributes* dialog.
- **Model Check** – Runs the WAM model check process.
- **Model Control** – Brings up the WAM Simulation Model Control .
- **Read Spectra Files**

WAM Simulation

A new WAM simulation is created by right-clicking in the Project Explorer and selecting **WAM** from the *New Simulation* menu. The WAM grid is then dragged under the new simulation to create a link. If there are multiple grids in the same project (for example, nested grid and parent grid), all grids should be linked to the simulation.

Multiple simulations can also be created.

Model Control

Once a simulation is created and the grid(s) is linked, the grid options can be edited in the *Simulation Model Control* which can be accessed by right-clicking on the simulation. Once all information are entered, WAM files can be exported, the project can be saved and WAM can be launched.

See WAM Simulation Model Control for more information.

Model Wrapper

Once all of the simulation information is ready, the WAM files can be exported by right-clicking on the simulation. After the WAM files are exported, then the model wrapper can be launched by right-clicking on the simulation and choosing **Launch WAM** . Both steps can be done with the **Save Project, Export, and Launch WAM** option, also under the right-click menu.

SMS will export all WAM files to a folder with the same name as the simulation under the WAM folder in the same directory as the saved SMS project. All wind dataset files will be in a folder called *WindInput* under the simulation folder. All WAM files, except the wind dataset, for each grid in the simulation will be in a folder with the same name as the grid under the simulation folder.

After all the files for a WAM simulation have been exported, the WAM executables can be launched. There are three WAM executables which are preproc, chief and map_to_rast. These must be executed in order for a given grid.

Preproc creates the Grid_Info and Preproc_prot files. WAM_Preproc reviews the grid and creates two files: "Grid_info" and "Preproc prot". The prot file is an ASCII dump of the model process. The Grid_info file is used in the next phase of the simulation.

Chief is the main process of WAM, and it creates the boundary value output files, the integrated parameter files, the restart files, the spectral output files and the WAM_Prot file. WAM_Chief takes the bulk of the run time. The process does not output any diagnostics as it runs, so the model wrapper cannot pass along progress information as the model runs. This process creates a prot file, an ASCII dump of the model process that can be reviewed in an editor. It also outputs three series of solution files. Each of these includes a particular type of output for a specified simulation interval.

The three types of output file are:

- IntOut### – The interval output file. It contains the spatially varied datasets computed by WAM.
- SpectraOut#### – The spectral output file. It contains the spectra at each spectral site, at the output frequency specified for spectral output.
- Restart### – A series of files which contain information to restart the simulation at a specific point if needed.

Map_to_rast creates the HDF5 file of the datasets that SMS can read. Map_To_Raster is a utility that converts the data in the IntOut### solution files into an HDF5 format so that SMS can read them. This allows for post processing of the datasets. SMS writes a script, fort.10, that instructs this utility to name the solution file wam_output.h5.

SMS will automatically run all grids in a simulation, even nested grids. The model wrapper will run a process for a coarse grids before running the same process for one of the grids nested inside of it.

Nested Grids

- Used to make a finer grid of a particular area that will use input information from the coarse grid.
- To create a nested grid, make a grid frame around the area of the coarse grid to which you wish to make a nested grid. Then, before the nested grid is mapped, check the *Fine Grid* checkbox and select the coarse grid from the *Coarse Grid* combobox. This will change the values for the newly mapped grid, such as the origin and increment, to correspond to cells in the coarse grid.
- The dataset for the wind of the coarse grid may be used for the fine grid. Other datasets, such as for ice and currents, for the fine grid will need to be made in the same manner as the coarse grid, if they are to be used.
- The nested(fine) grid will have to be added to the simulation as well as the parent(coarse) grid.

WAM Wet and Dry Cells

Wet/dry status for WAM cells is determined solely upon the z values (represented as a depth). If the depth is positive, the cell is wet. If the depth is negative, the cell is dry.

Only wet cells may be used as spectra sites.

To display wet/dry cells: Select the *WAM grid Display→Display Options* then turn on *land and ocean* cell. For best results also turn off *contours* .

The following operations may change the wet/dry status of a WAM cell since they are changing the depth values.

1) Select 1 or more WAM cells and then change the Z value found at the top.
2) Select and right-click a WAM cell. Choose **Interpolate Bathymetry...**
3) Enter a "single value" on the left side and press **Ok**
4) When converting a WAM map coverage to a WAM grid **Map→2DGrid** whatever value is in the *Depth Options* section will be converted.

WAM Cell Attributes

The *Cell Attribute* dialog allows specifying a spectra site name for the selected cell.

Spectra Sites

Since the spectra data of every cell of the grid at each time step would take an unreasonable amount of disk space, WAM writes spectra data at user specified cells. These cells are referred to as spectra sites and can be specified within the SMS interface. The spectra output can also be visualized on these cells.

Spectra sites can be made at any ocean cell on a Cartesian grid. To make a spectra site, select a cell on the desired Cartesian grid and change the cell attribute to Spectra Site and give it a name. If more than one cell is selected, then the name the spectra sites will receive is the name given concatenated with a number.

After WAM has run, and the spectra files have been read back into SMS, the spectral energy graph of the spectra sites may be viewed by right-clicking on the selected cell where the site is and selecting **Spectral Energy** . The spectra files may be read back into SMS by selecting the grid, and choosing **Read Spectral Files** from the *WAM* menu. The spectra files should be within the grid's folder within the simulation folder.

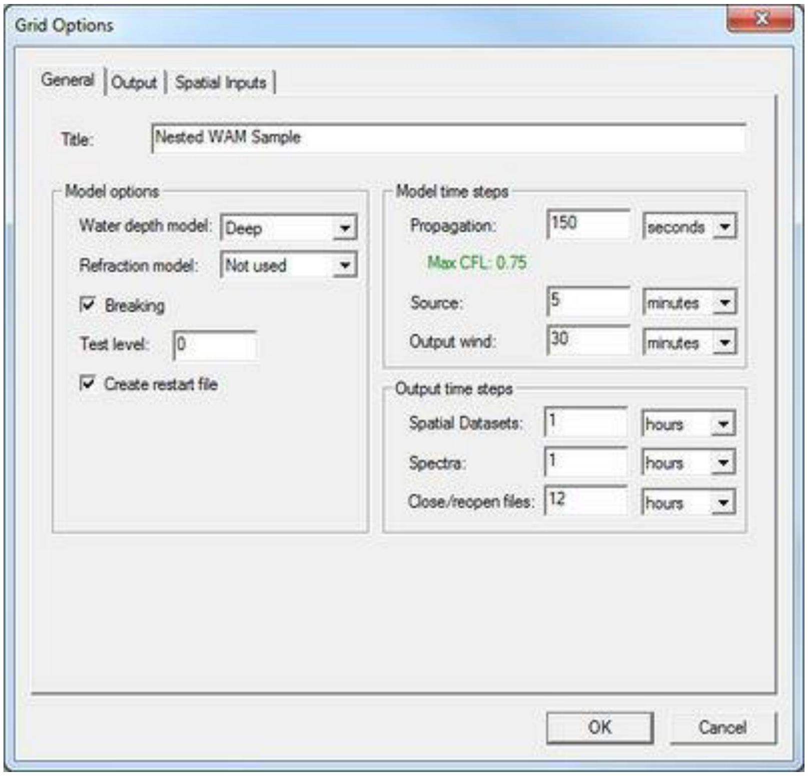
Related Topics

- WAM

WAM Grid Options

The *Grid Options* dialog is accessed by clicking on **Model Control** command in the *WAM* menu. The *Grid Options* dialog contains three tabs which are as follows:

General Tab



Model options

- *Water depth model* – Choosing a shallow depth model specifies the use of the full dispersion equation. Dispersion equation is where is radial frequency, is acceleration due to gravity, is the wave number, and is the water depth. Computes all phase and group celerity where there is depth dependence.
- *Refraction model* – If running in deep water, specify "Not used." It is not recommend to use if grid resolution is greater than 3-min (0.05 deg) on a spherical grid.
- *Breaking* – It is recommended that breaking be used when the water depth model is "Shallow." This will implement limited breaking within the WAM model.
- *Test level* – A control for output diagnostics and is mostly used for model debugging.
- *Create restart file* – Causes WAM to create a restart file to continue analysis at a future time.

Model time steps

- *Propagation* – The *Propagation* time step is controlled by the CFL condition. The CFL condition requires that the numerical solution cannot move faster than the group speed of the waves. It is recommended that the other model time steps and output time steps be multiples of the propagation time step.
- The CFL criteria must be less than 1.0 for WAM to run successfully. The estimated CFL value is shown below the time step. The value shows in green if the settings meet the criteria or red if not. The CFL computation is dependent upon whether deep or shallow water depth model is being used as well as the refraction model. Currently the CFL estimation in SMS does not account for currents but this is used within WAM. Therefore, the CFL estimate may look okay in SMS but fail when running WAM. If this occurs further reduce the propogation time step or change some of the other options.

- *Source* – This is the time step for the source term integration time step. Increasing this value can greatly decrease run times. However, it should be used with caution. It is recommended that the source term time step be no greater than 10 times the propagation time step.
- *Output wind* – This time step controls how frequently the WAM simulation updates the forcing terms. WAM interpolates between these intervals to attain a smaller forcing time step.

Output time steps

- *Spatial Datasets* – This is the frequency of output that will be generated that can be loaded into SMS as datasets. The WAM file it creates is the "integrated parameter map file."
- *Spectra* – This is the frequency of output for spectra data at defined observation cells.
- *Close/Reopen files* – When writing files on a computer, often data being written is stored temporarily in RAM before being written to disk. If a file is in this state and WAM crashes, the final data may not be complete. Closing and reopening the output files forces the file system to complete writing the data to file.

Output Tab

The *Output* tab consists of different Model options. Selecting each option will determine the dataset generated in the solution. Options include:

- Wind speed at 10m
- Wind direction
- Friction velocity
- Drag coefficient
- Significant wave height
- Wave peak period
- Wave mean period
- Wave TM1 period
- Wave TM2 period
- Wave direction
- Directional spread
- Normalized wave stress
- Sea significant wave height
- Sea peak period
- Sea mean period
- Sea TM1 period
- Sea TM2 period
- Sea direction
- Swell significant wave height
- Swell peak period
- Swell mean period
- Swell TM1 period
- Swell TM2 period
- Swell direction
- Swell directional spread
- Spectra of total sea
- Sea spectra
- Swell spectra

Sea waves and swell waves are defined as:

- Sea waves: waves generated by the wind blowing at the time, and in the recent past, in the area of observation.
- Swell waves: waves which have traveled into the area of observation after having been generated by previous winds in other areas. These waves may travel thousands of kilometers from their origin before dying away. There may be swell present even if the wind is calm and there are no 'sea' waves.

Spatial Inputs Tab

WAM uses spatial input for ice cover, currents, and wind. The *Spatial Inputs* section of the *WAM Model Control* is used to manage these datasets.

Ice Cover

The ice cover dataset is an editable scalar dataset.

- "Dataset" – After creating the dataset using the *Editable Datasets* dialog accessed from the *Model Control* , the dataset will be displayed in the project explorer in a folder named *Editable Datasets* . Only one ice cover dataset may exist at a time.

- "File" – Rather than using a dataset, specify the file name for ice.

Currents

The currents dataset is specified to be any vector dataset on the current grid.

- "Dataset" – The dataset and time step must both be selected. The selection may be removed by clicking on the **Clear** button. This deselects the dataset as the specified currents dataset, but it does not remove the dataset from the SMS project.
- "File" – Rather than using a dataset, specify the file name for currents.

Wind

The winds dataset can be set to any vector dataset that belongs to the current grid or another grid whose angle is zero and whose extents cover the current grid.

- "Dataset" – Nested grids may use the wind dataset of the parent grid. The time steps in the dataset used must match those used with the WAM simulation.
- "File" – PBL can be used to create an input wind file for WAM by selecting "WAM input" in the *PBL Model Control* . The fort.12 file created by PBL may be selected as the wind data. Note: This option is only useful if there is access to the "Planetary Boundary Layer" model which isn't available through SMS.

Related Topics

- WAM

WAM Simulation Model Control

The *WAM Simulation Model Control* can be accessed by right-clicking on the *Simulation* and selecting **Model Control** from the menu.

General Tab

The *General* tab has two sections: *Frequency and direction grid* and *Simulation run times* .

Frequency and direction grid

- *Number of frequencies* – Defines the extent of the spectral grid. Each frequency is 10% larger than the previous frequency. With the default of 25 frequencies, the maximum frequency is 0.4114 which corresponds to approximately a 2.5 second wave.
- *Number directions* – Determines the size of the directional bin in the spectra. The default of 24 corresponds to a 15 degree bin.
- *Lowest frequency* – Defines all the frequencies in the grid. Each frequency is defined by the frequency before it, so this is an important parameter.

Starting Frequency Band

This is a very important parameter in WAM. Based on historical testing of the source terms (see Komen et al., 1994) there was a prescribed frequency range to tune the Discrete Interaction Approximation and the other source terms defined in the energy balance equation. This work used a starting frequency band of 0.04177248 Hz. Also the definition of the frequency band range is dictated by:

$$f(n+1) = 1.1 * f(n)$$

where

$$n = 1, NFRE - 1$$

($NFRE$ is number of frequency bands)

For many application this starting frequency may be too high for the case of a large-oceanic scale domain like the Pacific Ocean where there is the potential for energy carrying frequencies lower than the designated threshold (i.e. 25-sec peak spectral wave periods, where wave period is the inverse of the frequency). Also, for small-scale fully enclosed bodies of water (i.e. the Great Lakes, reservoirs etc.) the range of wave frequencies is much higher, where the lowest frequency may be on the order of 0.0833-Hz (or 12-sec). The initial frequency value can be changed, upward or downward, as long as the range of frequency values can be defined from the 0.0417728-Hz standard. For the example of the Pacific Ocean implementation, a starting value of 0.03138428-Hz is used, where f(4) = 0.0417728-Hz as defined by the above equation. Setting a higher starting point for a Great Lakes simulation and knowledge of the wave climate an initial value could be 0.050545-Hz. Standard practices use conservatism allowing for energy to be transferred. Also, it is also advantageous to set the first frequency band high enough to minimize the computational time, where the propagation of wave energy is dictated by the group speed of the first frequency band. The lower the frequency, the shorter the stable propagation time step. The number of direction bands is nominally set to 24 or a 15-deg directional resolution of the wave spectra. Selection of a higher directional resolution (or increasing the number of directional bins) will linearly increase computational time because the directional loop is the outer-most loop in all of WAM’s calculations.

Simulation run times

The starting and ending time of the simulation must be entered in this section. The run times must be less or equal to the time available for wind data. Time is given in MM/DD/YYYY HH:MM:SS AM/PM format.

- *Start* – Indicates when the simulation is to begin.
- *End* – Indicates when to simulation is to stop. Can be used to reduce the run time.

Spectra Tab

The parameters in the *Spectra* tab with the exception of Fetch are Jonswap parameters.

- *Run type* – Can be set as "Cold Start" or "Hot Start". The "Hot Start" option requires a hot start file generated from a previous run.
- Phillips' parameter
- Peak/Max freq
- Overshoot factor
- Wave direction
- Left peak width
- Right peak width
- Fetch
  - Specified
  - 0.5 of the latitude increment

Related Topics

- WAM

WAM Spectra from STWAVE Grids

Spectra Sites

Since the spectra data of every cell of the grid at each timestep would take an unreasonable amount of disk space, WAM writes spectra data at user specified cells. These cells are referred to as spectra sites and can be specified within the SMS interface. The spectra output can also be visualized on these cells.

Spectra sites can be made at any ocean cell on a Cartesian grid. To make a spectra site, select a cell on the desired Cartesian grid and change the cell attribute to Spectra Site and give it a name. If more than one cell is selected, then the name the spectra sites will receive is the name given concatenated with a number.

After WAM has run, and the spectra files have been read back into SMS, the spectral energy graph of the spectra sites may be viewed by right-clicking on the selected cell where the site is and selecting **Spectral Energy** . The spectra files may be read back into SMS by selecting the grid, and choosing **Read Spectral Files** from the *WAM* menu. The spectra files should be within the grid's folder within the simulation folder.

WAM Spectra From STWAVE Grids

WAM can be used to provide boundary condition information to a spectral wave model such as STWAVE. **WAM→STWAVE** has a defined and support mechanism for converting WAM spectral output into STWAVE input.

The first part of this process is to define the locations where WAM will generate output for the STWAVE BC. Two or more WAM cells may be used along a STWAVE boundary. The STWAVE grids must exist and the boundaries types specified (which side(s) to apply spectra.

To create spectral sites along the STWAVE boundaries:

1) Verify/Modify the boundary types of a STWAVE Grid.
    1) Select a STWAVE grid.
    2) Select menu item **STWAVE → Model Control** then press the **Boundary Condition...** button.
    3) Select the appropriate boundary type for each edge.
    4) Repeat for each STWAVE grid.
2) Select and right-click on a WAM grid from the tree item on the left of SMS.
3) Select **Create spectral sites...** to bring up the *WAM spectral spectral sites from STWAVE* dialog.
4) Select the grids the project should to generate sites for.
5) Select a spacing option on the right hand side and the number of points/sites.
6) Press **Ok** .

Each STWAVE boundary cell will have a candidate WAM corner (where WAM spectra are output). A candidate corner is the nearest WAM corner to that STWAVE boundary cell. Multiple STWAVE cells may have the same candidate WAM corner. Duplicate WAM corners are automatically removed.

Spacing can be done by either spacing along candidate corners or by specifying an exact number of sites/corners along each boundary.

If spacing by candidate point(s) is chosen:

- Spacing by every 1 candidate points will keep all candidate corners and make them spectra sites.
- Spacing by every 2 candidate points will keep every other corner and make those spectra sites.

Regardless of the option selected at least two corners will be used on each STAVE boundary.

Related Topics

- WAM

6. Riverine and Estuarine Models

6.1. ADH – Adaptive Hydraulics Modeling

ADH

"ADH is a state-of-the-art ADaptive Hydraulics Modeling system developed by the Coastal and Hydraulics Laboratory (CHL), ERDC, USACE , and is capable of handling both saturated and unsaturated groundwater, overland flow, three-dimensional Navier-Stokes flow, and two- or three-dimensional shallow water problems." [158]

Information about the ADH model can be found at the ADaptive Hydraulics Modeling HomePage .

The ADH model can be added to a paid edition of SMS.

Graphical Interface

Dialogs

SMS provides the capability to view and edit model parameters via dialog boxes specific to the ADH interface. The following are descriptions of important windows accessible from the

menu and right-click menus.

ADH Menu

- Assign BC – When a node or arc is selected, brings up a dialog where boundary condition parameters can be assigned.
- Assign Bed Layers – Brings up a dialog to assign bed layer specifications.
- Assign Consolidations
- Material Properties – Brings up a dialog where material properties are defined.
- Hot Start Initial Conditions – Brings up a dialog to define the initial condition dataset to start a simulation.
- Time Series Data – Brings up a *Time Series* dialog.
- Model Check – Launches the model check process.

- Model Control – Brings up the *Model Control* dialog where model run parameters are set.
- Sediment Library Control – Brings up a dialog where sediment properties can be defined.
- Run Model – Launches the model check, pre-ADH model wrapper, and ADH model wrapper.
- Read ADH Solution – Opens ADH solution files.

Files

An ADH simulation is defined using three main files: mesh geometry (*.3dm), boundary conditions (*.bc), and hot start conditions (*.hot). ADH expects a 3-dimensional mesh geometry since it can solve 3D models, but SMS will only interface the 2D capabilities and will therefore only read and write 2D entities in the *.3dm file. A generic SMS mesh geometry file (*.2dm) can be used by ADH if the extension is changed to *.3dm. To facilitate handling the ADH files, SMS creates a simulation file (*.sim) which lists the individual files. Opening the *.sim will load the ADH files in SMS.

Each file is a card based text file that can be viewed and edited by a text editor (i.e. Notepad).

- Mesh geometry file (*.2dm/*.3dm)
- Boundary condition file (*.bc)
- Hot start file (*.hot)
- Boat definition file (*.bt)

Functionality

ADH can address a range of modeling application, including: [159]

- Design of approach channel improvements
- Guardwall porting design
- Design of a fish passage structure through an existing spillway
- Velocity field calculations during different stages of construction
- Overall effects of sedimentation due to the proposed work.
- Evaluating the effects of multiple projects constructed within a river reach to support objectives for both navigation and ecosystem restoration.
- Salinity intrusion analysis in conjunction with system changes.
- Impacts of navigation channel design changes on ship passage by supplying hydrodynamic results.
- Effects of vessel traffic on hydrodynamics, salinity, and sedimentation with the use of simulating vessel movement in the model.
- Flood inundation by use of dynamic and continuous wetting/drying fronts.

Using the Model / Practical Notes

- Sediment Transport and Bed Layers
- Time Series
- Wind Stations

ADH String Structures

ADH uses string structures to assign properties to entities. The following table describes the relation between the string structures and SMS structures.

ADH to SMS (Reading)

ADH String Structure		SMS Structure
NDS with DB card	and only 1 node ID	Node with boundary condition (BC)
	and 2 or more node IDs	Nodestring with BC
EGS with NB card (and possibly FLX card)		Nodestring with BC (and flux output)
MDS with FLX card		Nodestring with flux output
MTS with MP cards		Material with properties

SMS to ADH (Writing)			
SMS Structure			**ADH String Structure**
Node with BC (Dirichlet)			NDS (only 1 node ID) with DB card
Nodestring	with Natural BC	with flux output	EGS with NB and FLX cards
			EGS with NB card
	with Dirichlet BC	with flux output	NDS (2 or more node IDs) with DB card, and MDS with FLX card
			NDS (2 or more node IDs) with DB card
	with flux output		MDS with FLX card
Material with properties			MTS with MP cards

Case Studies / Sample Problems

The following ADH simulation files have been provided by CHL to test ADH and the SMS interface capabilities. Additional test cases will be posted here as the interface can successfully read, write, and run the ADH model from within SMS.

SMS 10.1 Development version using ADH rev #3669 dated August 2007:

- **Bayou_Sorrel** – Uses the NB OVL and NB OTW boundary condition cards. Includes two materials on a complex mesh.
- **Pool5** – Uses the NB OVL and NB OTW boundary condition cards. Includes two inflows with one outflow and four materials on a complex mesh.
- **nb dis** – Tests the NB DIS boundary card on a straight flume.
- **op bt** – Tests the OP BT card which includes the vessel movement library. This test case contains a boat definition file (*.bt).
- **db lde** – Tests the DB LDE pressure lid card on a straight flume.
- **db ldh** – Tests the DB LDH pressure lid card on a straight flume.
- **db lid** – Tests the DB LID pressure lid card on a straight flume.

External Links

- ADH Website
- ADH Documentation List
- ADH Manual
- ADH Model Troubleshooting Flowchart

Related Topics

- SMS Models

ADH Bed Layers Assignment

To open this window, global bed layers must be specified in *Sediment Library Control* on the *Global Material Properties* tab and at least one node must be selected. After a selection is made, select the *ADH* | **Assign Bed Layers...** menu item or use the right-click menu.

Dialog Description

The following controls specify the bed layer specifications to be assigned to the currently selected nodes to overwrite the global specifications. Overwriting is specified on a layer by layer basis at the selected nodes. This window is expandable by dragging the window edges to increase visibility of the spreadsheet data.

Bed Layers (untitled group box)

Spreadsheet contains the defined bed layers as columns and bed layer specifications as rows. The bed layers begin with Layer 1 and proceeds by ascending order by ID to the right. The specification rows are as follows:

- Edit differing specifications – Contains Edit check boxes and only appears when multiple nodes are selected and the bed layers specified at the nodes conflict. A check box is only placed in the columns of layers with conflicts (denoted by asterisks appended to the layer ID). The entire layer column will be disabled unless Edit is checked. If it is checked when accepting changes (exiting the window by clicking **OK**) the specifications for the bed layer will be applied to all selected nodes. Bed Layers that are not edited (remain unchecked) will not be changed.

- *Overwrite global specification* – Contains *Overwrite* check boxes for each bed layer to specify whether the layer will be overwritten. If checked, the parameter cells of the column are enabled for editing, else the global specification values of the bed layer are displayed as read-only. The check box is disabled if a layer conflict exists and Edit is unchecked.
- *Sediment distribution* – Title cell with each defined sediment listed below in its own row. The row contains **Normalize** buttons for each bed layer. Clicking on the button will normalize the distribution of each sediment such that the total distribution equals 100%. The button is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if *Overwrite* is unchecked or checked, respectively.
  - *Sediment (name)* – Consists of this sediment's distribution (as a percentage) for each layer. A sediment row is provided for all defined sediments. The value must be a positive real number and correlates to the MP SBN card. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if *Overwrite* is unchecked or checked, respectively.
  - *Total* – Displays the current total specified distribution of all sediments for each layer. The row updates automatically as specifications change and cannot be edited. Each layer's total sediment distribution must equal 100%. The cell is disabled if a layer conflict exists and Edit is unchecked.
- *Thickness* – Consists of the each bed layer's thickness measured in meters. The value must be a positive real number and correlates to the MP SBN card. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if *Overwrite* is unchecked or checked, respectively.
- *Bulk Density* – Consists of each bed layer's bulk density measured in kilograms per cubic meter. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if *Overwrite* is unchecked or checked, respectively.
- *Erosion crit. shear* – Consists of each bed layer's critical shear stress for erosion measured in newtons per square meter. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if *Overwrite* is unchecked or checked, respectively.
- *Erosion rate constant* – Consists of each bed layer's unitless erosion rate constant. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if *Overwrite* is unchecked or checked, respectively.
- *Erosion rate exponent* – Consists of each bed layer's unitless erosion rate exponent. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if *Overwrite* is unchecked or checked, respectively.

Layer 1 is the deepest (or bottom) layer; the highest numbered layer is exposed to the stream flow. The layer IDs cannot be edited and bed layering order is fixed to ascending layer ID order (See Sediment Transport and Bed Layers).

Asterisks (*) denote bed layers with specifications that differ between the selected nodes. This note only appears when multiple nodes are selected and the bed layers specified at the nodes conflict. Asterisks are appended to the layer ID of differing layers in the title row across the top of the spreadsheet.

Related Topics

- ADH
- Sediment Transport and Bed Layers

ADH Boat Definition File Cards

The following lists ADH *.bt cards that are recognized by SMS. For a description of card specifics, please use the ADH Quick Reference .

Boat Properties

	Read	Edit	Write	Notes
BLEN	Allowed	Allowed	Allowed	
BOAT	Allowed	Allowed	Allowed	
BWID	Allowed	Allowed	Allowed	
CBOW	Allowed	Allowed	Allowed	
CSTR	Allowed	Allowed	Allowed	
DRFT	Allowed	Allowed	Allowed	
ENDD	Allowed	Allowed	Allowed	
FDEF	Allowed	Allowed	Allowed	
PBOW	Allowed	Allowed	Allowed	
PROP	Allowed	Allowed	Allowed	
PSTR	Allowed	Allowed	Allowed	
SDEF	Allowed	Allowed	Allowed	

Related Topics

- ADH

ADH Extract WSE

SMS now has the ability to extract water surface elevation to a nodestring boundary condition from an existing dataset. This is done by right-clicking on the nodestring and selecting *Boundary Condition* | **Extract Water Surface Elevation...** and selecting the dataset to extract from in the *Extract Boundary Condition* dialog. SMS will break up the existing nodestring into segments and extract the water surface elevation at the midpoint of each segment. The nodestrings are then assigned the Tailwater (outflow) boundary condition and the extracted data is set as the time series. This data can be edited in the *ADH Boundary Condition Assignment* dialog.

Extract Boundary Condition Dialog

The dialog has the following options:

- *Regional model time options* – If extracting from a transient dataset, the time steps used can be set.
  - *Start time* – The first time step to use in the extraction.
  - *End time* – The last time step to use in the extraction.
- *Boundary condition time options* – If extracting from a transient dataset, the time rate can be set.
  - *Start time* – Sets the time rate for the time series assigned to the boundary condition.
- *Regional scalar model solution* – In the data tree, available datasets are shown. Select the dataset to be used for extraction.

Related Topics

- ADH
- ADH Boundary Condition Assignment

ADH Hot Start File

The hot start file is mandatory and contains initial conditions for each node of a mesh. ADH allows for the specification of depth and velocity hydrodynamics, and sediment concentration and displacement datasets. The model requires at least depth initial conditions for hydrodynamics, and non-specified conditions will be defaulted to values of zeros (0.0).

ADH Hot Start (*.hot) File

The *.hot file format is actually the same as the ASCII dataset Files (*.dat) but necessitates the special extension since it should only contain following specifically named datasets:

Hot Start Datasets

Name	Initial Condition
"ioh" or "IOH"	Depth
"iov" or "IOV"	Velocity
"ioc" or "IOC"	Sediment concentration (SMS currently does not support this)
"iod" or "IOD"	Sediment displacement (SMS currently does not support this)
Name	**Initial Condition**
"ioh" or "IOH"	Depth
"iov" or "IOV"	Velocity

SMS will recognize and read in a hot start file if the dataset definition matches the current mesh definition (number of nodes and elements) and the ADH is the active model (other models also use a *.hot extension). Valid datasets with the above names are added to a locked "ADH Hot Start" folder in the Project Explorer. The folder and datasets can not be deleted or renamed from the tree. See Hot Start Initial Conditions for interface information.

Related Topics

- ADH

ADH Hot Start Initial Conditions

This window can be accessed by the *ADH* | **Hot Start Initial Conditions** menu item. ADH requires that at least an initial depth dataset is defined to start a simulation. ADH can not start with a completely dry domain.

The initial condition datasets will be generated and appear in the "ADH Hot Start" folder of the Project Explorer. This folder and the datasets are locked for editing except from this dialog.

Dialog Description

Depth

Constant water surface / Constant depth / Use defined dataset radio group specifies the method for defining the initial depth dataset. The *Constant water surface* control group includes the *Elevation* real number edit field which only allows elevations greater than the maximum mesh elevation. This produces a fully flooded domain with a flat horizontal water surface. The *Constant depth* control group similarly includes the *Depth* positive real number edit field. The depth value is applied to every node to create a water surface which is offset from the mesh geometry. The *Use defined dataset* control group allows for the selection of an existing scalar dataset located in the Project Explorer via **Select Dataset...** button. The name and time step information (if transient) of the selected dataset will appear below this button.

Velocity

Specify velocity check box specifies whether an initial velocity dataset will be included and if checked, enables the *Constant velocity / Use defined dataset* method specification radio group. The *Constant velocity* control group includes *X* and *Y* velocity component real number edit fields. The composite velocity vector will be applied to every node in the domain. The *Use defined dataset* control group allows for the selection of an existing vector dataset located in the Project Explorer via **Select Dataset...** button. The name and time step information (if transient) of the selected dataset will appear below this button.

Related Topics

- ADH

ADH Material Properties

To open this window, select the *ADH* | **Material Properties...** menu item.

Dialog Description

Materials (Untitled group box)

Name text displays the name of the currently selected material in the material list.

ID text displays the numeric ID of the currently selected material in the material list.

List box contains the existing globally defined materials.

ADH Material Properties (Tab control)

This tab control contains the following three tabs:

Properties

Manning's roughness real number edit field limited to a value of 0.0 to 0.1. This correlates to a FR MNG card applied to a material.

Kinematic eddy viscosity / Estimated eddy viscosity radio group specifies which method to use and correlates to the MP EVS and MP EEV cards, respectively. The *Kinematic eddy viscosity* control group includes the eddy viscosity tensor edit fields of *EVxx* , *EVyy* , and *EVxy* . The *EVxy* is specified once, and copied to the second term. All edit fields allow positive real number input. The *Estimated eddy viscosity* includes the *Weighting factor* real number field. This factor is used with other model conditions to calculate the eddy viscosity during the simulation.

Include coriolis force check box and the associated *Latitude* edit field will include the MP COR when checked. The edit field allows real number input between -90.0 and 90.0.

Exclude from computations (remove from domain) check box correlates to a OFF card applied to a material.

Meshing and Boundary Conditions

Mesh Refinement

Maximum number of refinement levels positive integer edit field correlates to the MP ML card.

Error tolerance for refinement positive real number edit field correlates to the MP SRT card.

Boundary Condition Assignment

Apply rain or evaporation check box with the associated **Flow (per unit area) curve** button, *Series time units* combo box, and *Specify curve fir tolerance* check box and positive integer edit field defines the rain/evaporation series. Clicking on the curve button will open the XY Series Editor . This control group correlates to the NB OVL boundary condition card with a XY1 series definition card.

Transport Constituents

List box contains the existing transport constituents.

Molecular diffusion rate edit field displays the parameter value for the currently selected transport constituent. This data pertains to the MP DF card.

Error tolerance for refinement edit field displays the parameter value for the currently selected transport constituent. This data pertains to the MP TRT card.

Miscellaneous (Outside of any group)

General Material Properties... button opens the *Materials Data* window. Any changes made to global materials will be reflected in the materials list box.

Related Topics

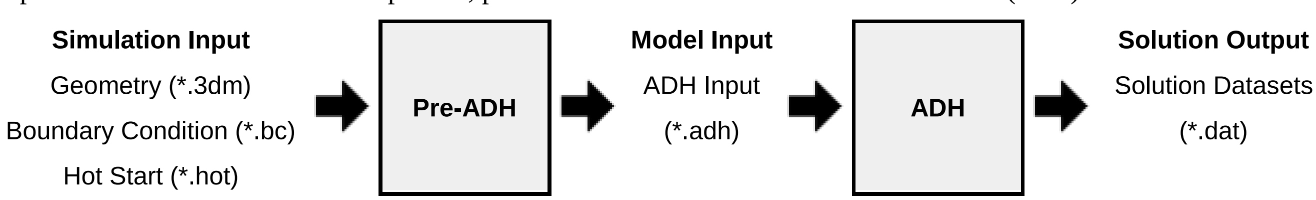
- ADH

ADH Run Model

To access the ADH model wrapper, select the *ADH* | **Run Model** menu item.

If the simulation has been changed and not yet saved, a prompt will appear to do so before running the model. A prompt will also appear asking to locate the Pre-ADH and ADH executables prior to running each if SMS does not know the location or find the executable. Model executable locations can be managed in on the *File Locations* tab of the *Preferences* dialog , accessed from the *Edit* | **Preferences...** menu item.

Pre-ADH takes the simulation input files (*.3dm, *.bc, and *.hot), performs some checks and creates the *.adh model input file. ADH reads the model input file, performs calculations and writes solution files (*.dat).



If Pre-ADH is not successful (the *.adh file is not written) then SMS will not allow ADH to be executed. Upon completion of ADH, the solution files can be automatically loaded into SMS or opened manually.

Dialog Description

Progress

Status line is blank for Pre-ADH and provides time step computation information for ADH.

Progress bar and text shows the percentage of completion.

Run Time displays the time since starting the executable. This is updated when the progress is updated.

Output

Pre-ADH Output or *ADH Output* field displays the last 1000 lines of commentary given by the running executable.

Output Text To File allows all commentary provided by the executable to be saved to a text file (*.txt). This button is enabled upon completion of the executable.

Load Solution check box is available only for ADH and is enabled upon successful completion of ADH. Check this option to automatically load the solution datasets into SMS.

Abort / **Run ADH** / **Exit** button controls the progress of the model wrapper.

Related Topics

ADH

ADH Time Series

ADH includes several model specific time series (curve) groups to organize model curves by type. The curves of these groups can be viewed and edited in the *Time Series* editor and include time series attributes specific to ADH. The following table describes the groups:

ADH Curve Groups

Name	Description	Specification Use	Restrictions
Depth	Time offset with depth measurement (L)	Flow or pressure boundary condition (DB OVH or DB LDH card)	
Discharge	Time offset with flow rate (L^3/T)	Flow boundary condition (NB DIS card)	
Draft	Time offset with draft measurement (L)	Pressure boundary condition (DB LID card)	
Elevation	Time offset with elevation	Pressure boundary condition (DB LDE	

	value (L)	card)	
Flow (per unit length)	Time offset with unit flow rate (L^2/T)	Flow boundary condition (NB OVL card in conjunction with an ADH edge string)	
Meteorologic	Time offset with unit flow rate (L/T)	Flow boundary condition (NB OVL card in conjunction with an ADH material string)	
Time Step Size	Time offset with time step size (T)	Simulation condition (TC IDT card)	Unavailable in steady state
Transport	Time offset with concentration measurement (PP)	Transport boundary condition (NB TRN or DB TRN card)	Unavailable in steady state
Velocity	Time offset with velocity components (L/T) (three columns)	Flow boundary condition (DB OVH or DB OVL card)	
Water Surface Elevation	Time offset with elevation value (L)	Flow boundary condition (NB OTW card)	
	(Units: T = time, L = length, PP = unitless parts per)		

Curves with similar value types (i.e. pressure boundary conditions) have been separated into individual groups to avoid the unintentional application.

Specifying and Selecting Time Series Data

In the *ADH* menu, selecting the **Time Series Data...** option will open the *Time Series* editor with the above mentioned available curve groups. This is provided to quickly view or specify all simulation time series data from one window. However, specifying a time series data in the editor accessed from this menu item does not associate the time series with a boundary or simulation condition. Various windows of the ADH interface (i.e. *Boundary Condition* tab of the *Material Properties* window) include a **curve** button which opens the *Time Series* editor with only the condition's curve group available. Association of the selected curve with a condition occurs when the editor's **OK** button is selected. These buttons will display the name and plot of the selected time series, unless in a spreadsheet (i.e. *Transport Constituents* tab of the *Boundary Condition Assignment* window).

Steady State Simulation

If setting up a steady state simulation, the *Time Series* editor is still used to specify and associate time series. There is no restriction on the time series data in the interface (the number of rows is not limited), however SMS will write a steady state series consisting only of the first condition value(s) listed. The written series is created using the simulation start and end times (zero offset and the simulation length from the Time tab in Model Control, respectively) with the same condition value(s). To avoid confusion, it is suggested that only one row should be specified in the editor (with a zero time offset) for all steady state series. SMS will validate that a time series selected to be associated with a boundary condition is appropriate steady state data.

Boundary Condition File

When saving the ADH Boundary condition file (*.bc), only the time series that are actually used by the simulation (associated with conditions) are written out. These are converted to the units expected by ADH (i.e. mph to ft/s) as necessary and written as XY1 cards with XYT cards if curve fit tolerance is specified (apart of time series attributes). Unused curves can be saved by exporting them from the Time Series editor. A written series that is not defined for the full simulation duration is automatically appended with the simulation end time and the last specified condition value(s). A time series in the file can appear different than as specified in the Time Series editor because of unit conversions and validation.

After reading a XY1 card from a file, SMS will assign the data to a curve group based on the type of card(s) that references it in the file. If multiple types of cards reference the same time series, then the data will be assigned to multiple curve group as separate and unique time series.

Related Topics

- ADH

ADH Time Series Attributes

This window is accessed from the *Time Series* window by clicking on the **Attributes...** button. The current curve group must be an ADH model specific curve group (i.e. ADH Discharge) to open this attributes window. The attributes are only associated with the selected curve.

A curve fit tolerance (XYT card) can be specified with each curve to be used when ADH interpolates values from a curve (while time adapting).

Dialog Description

Specify curve fit tolerance check box and the associated positive integer edit field specify whether a tolerance is included and its value. The edit field is enabled only if the check box is checked.

Related Topics

- ADH

ADH Wind Stations

Wind station data can be included in an ADH simulation to apply wind stress on the water surface. This data is in addition to and should not affect the mesh geometry, so stations are specified on a feature coverage in the Map module. The Spatial Data coverage is used because it allows the specification of generic time series data at locations represented by feature points. ADH wind station data is specified in SMS as wind speed data (NOTE: ADH currently expects wind stress data, but is planned to accept wind speed data.)

Creating Wind Stations

To create a wind station for ADH, first create a new Spatial Data coverage and give it a descriptive name. Using the **Create Feature Point** tool , add a point at the location of the wind station. Right-click on the feature point with the **Select Feature Point or Node** tool , point at the *Add* option, and select *Time Series* . This will open the *Time Series* editor. Select a velocity curve group (either "Velocity – Components" or "Velocity – Mag. & Dir.") and click on the **New...** button. Specify a name, curve data, and reference time. When finished, clicking the **OK** button adds the selected curve to the selected feature point's data. Right-clicking on the feature point again, this menu will display the added time series and provide options to view, edit, or delete this data.

After wind stations have been created on a coverage, the ADH simulation must be told to include this information on the *Model Parameters* page of *Model Control* . Since a specific coverage is chosen, create multiple coverages and quickly switch wind conditions as necessary. The velocity time series data of a coverage is not specific to ADH, and consequently do not include ADH time series attributes which contains curve fit tolerance. Therefore, a general curve fit tolerance can be specified in *Model Parameters* to be attributed to every velocity time series in the selected coverage.

Steady State Simulation

If setting up a steady state ADH simulation, a spatial data coverage is still used to specify wind stations and their time series. However, SMS will write a steady state velocity series consisting only of the first condition values listed regardless of the reference time. The written series is created using the simulation start and end times (zero offset and the simulation length from the *Time* tab in *Model Control* , respectively) with the same condition values. To avoid confusion, it is suggested that only one row should be specified in the editor (zero offset value with any reference time) for all steady state velocity series. The user is responsible for specifying appropriate steady state data.

Boundary Condition File

When saving the ADH Boundary condition file (*.bc), SMS will check to whether wind station data is included and, if specified, that the selected coverage is still valid (exists). SMS will then proceed to iterate through every feature point in the selected coverage, formatting series data and writing XYC and XY2 cards (with XYT cards if curve fit tolerance is specified). Since the general velocity types (Velocity – Components and Velocity – Mag. & Dir. curve groups) include reference time, part of the series formatting includes extracting data and re-referencing it to the simulation start time. If the data does not cover the simulation start or end time, it will be appended with the time and the first or last specified condition values, respectively. The extracted series will start at the simulation start time and include all remaining data. Formatting also includes converting velocity data to component form (magnitude and direction to x,y components) and units expected by ADH (i.e. mph to ft/s) as necessary. A velocity time series in the file can appear different than as specified in the Time Series editor because of form and unit conversions, and validation. Any feature point without velocity data will be ignored along with all other feature objects the coverage may contain. If a feature point contains multiple velocity time series, then only the data of first time series found will be written. Coverages not included in the simulation can be saved separately if desired.

As SMS reads an ADH boundary condition file, wind station data will be added to a new spatial data coverage with the default name of "ADH Wind Station". This coverage will be automatically selected in Model Paramters. If the file contains any curve fit tolerances for wind station velocity time series, the minimum tolerance (the values may be different) will be used for the entire coverage.

Realted Topics

ADH

6.1.a. ADH Boundary Condition

ADH Boundary Condition

To open this window, at least one node or nodestring must be selected. After a selection is made, select the *ADH* | **Assign BC...** menu item or use the right-click menu.

ADH boundary conditions consist of two specification types (Dirichlet and Natural) and three condition types (flow, pressure, and transport). Dirichlet data is applied on the domain to individual nodes or to groups of nodes (defined with nodestrings). Natural data (flux) is applied through edges of the domain defined by nodestrings and includes a friction specification. A node or nodestring can be assigned only one condition option from each of the three condition types (two flow conditions cannot be assigned to a single node).

Boundary Conditions

Flow Condition Options	Specification Type	Information
None	---	No condition specified
Flow (per unit length)	Natural	Specified by a Flow (per unit length) time series ; NB OVL card with a friction card
Sidewall (no through flow)	Natural	Creates a vertical wall with the specified shear stress; just a friction card
Subcritical inflow	Dirichlet	Specified by a Velocity time series ; DB OVL card
Supercritical inflow	Dirichlet	Specified by a Depth time series and a Velocity time series ; DB OVH card
Tailwater (outflow)	Natural	Specified by a Water Surface Elevation time series ; NB OTW card

		with a friction card
Total discharge	Natural	Specified by a Discharge time series ; NB DIS card with a friction card
Pressure Condition Options	**Specification Type**	**Information**
None	---	No condition specified
Lid (depth underneath)	Dirichlet	Specified by a Depth time series ; i.e. a structure (ex. culvert, sluice gate, etc.) defined using bed elevation as datum; DB LDH card
Lid (draft)	Dirichlet	Specified by a Draft time series ; i.e. a floating object (ex. boat, dock, etc.) with draft; DB LID card
Lid (elevation)	Dirichlet	Specified an Elevation time series ; i.e. a structure (ex. culvert, sluice gate, etc.) defined using the elevation datum; DB LDE card
Transport Condition Options	**Specification Type**	**Information**
None	---	No condition specified
Dirichlet	Dirichlet	Specified by a Transport time series ; DB TRN card
Equilibrium	Dirichlet	Specified by initial concentration (in ppm); This condition can only be assigned to sediments and specified in simulations without cohesive sediments; EQ TRN card
Natural	Natural	Specified a Transport time series ; NB TRN card
Weir Options	**Specification Type**	**Information**
None	---	No condition specified
Weir		Nodestrings are specified as upstream, upstream edge, downstream or downstream edge. See the WER and WRS card

Natural Flow Condition Friction Options

Friction Options	Information
Manning's n	Specified by a roughness value; FR MNG card
Equivalent roughness height	Specified by a roughness height (in ft or m); FR ERH card
Submerged aquatic vegetation	Specified by an undeflected stem height (in ft or m); FR SAV card
Un-submerged rigid vegetation	Specified by a roughness height (in ft or m), averaged stem diameter (in ft or m), and average stem density (in stems/ft^2 or stems/m^2); FR URV card

All mesh boundary edges that do not have a boundary condition will be treated as a frictionless sidewall with no through flow.

All boundary conditions listed above can only be assigned to the mesh boundary, expect the pressure conditions which can also be assigned to the mesh interior.

When changing a condition type on any tab, the existing condition information is cleared from the window such that the information must be respecified upon returning to that type.

Dialog Description

The following controls specify the boundary condition to be assigned to the currently selected entities. This window is expandable by dragging the window edges to increase visibility of the spreadsheet data on the *Transport tab* .

Boundary Condition Type (Tab Control)

This tab control can contain the following three tabs:

Flow

This tab is only available if all of the currently selected nodes or nodestrings are on the mesh boundary.

Boundary type combo box contains the conditions available for flow (listed above) for the selected entities.

Series data curve button selects the time series to be associated with the selected boundary type. The text label above this curve button will change with the boundary type to denote the type of input. Clicking on the curve button will open the Time Series editor (limited to the curve group of the boundary type). A second curve button will appear for boundary types that require two conditions with time series data.

Friction combo box specifies the type of bed shear stress to be applied to the boundary edge and is available only for Natural boundary types. The friction options are:

- *Manning's n* provides the *Manning's n* roughness real number edit field limited to a value of 0.0 to 0.3.
- *Equivalent roughness height* provides the *Roughness height* positive real number edit field.
- *Submerged aquatic vegetation* provides the *Undeflected stem height* positive real number edit field.
- *Un-submerged rigid vegetation* provides the *Roughness height*, *Average stem diameter* and *Average stem* density positive real number edit fields.

Pressure

This tab is always available.

Boundary type combo box contains the conditions available for pressure (listed above) for the selected entities.

Series data curve button selects the time series to be associated with the selected boundary type. The text label above this curve button will change with the boundary type to denote the type of input. Clicking on the curve button will open the Time Series editor (limited to the curve group of the boundary type).

Transport Constituents

This tab is only available if all of the currently selected nodes or nodestrings are on the mesh boundary and constituents have been included in Transport Constituents .

Boundary type combo box contains the conditions available for transport (listed above) for the selected entities.

Spreadsheet contains the following information specific to each transport type:

- *None* clears the display of any information in the spreadsheet.
- *Dirichlet* provides each defined constituent (first column) with a *Include* check box (second column), *Specify Series* ... curve button (third column), and a series name cell (fourth column). To include the constituent in the specification of the transport condition, check the *Include* control and specify a concentration series by clicking on the *Specify Series* ... button which will open the Time Series editor (limited to the Transport curve group). When a series has been defined, the name will be listed in the series name cell. A series can be selected by multiple constituents. The concentration series button and name cells are only enabled if the constituent is included in the transport source.
- *Equilibrium* provides each defined sediment (first column) with a Include check box (second column) and a positive real number *Initial concentration* cell (third column). To include the sediment in the specification of the transport condition, check the *Include* control and specify a concentration (in parts per million). If the sediment is not to be present at the beginning of the simulation, specify a concentration of zero. The *Initial concentration* cell is only enabled if the sediment is included in the transport source.
- *Natural* provides the same as *Dirichlet.*

Output (untitled group box)

Output calculated flow across this entity check box is available only for nodestrings and correlates to the FLX card.

Related Topics

ADH

ADH Boundary Condition File Cards

The following lists ADH *.bc cards that are (or will be) recognized by SMS and their capability (reading, editing values and writing) currently available within the interface. For a description of card specifics, please use the ADH Quick Reference . SMS will ignore any card that is not listed below when reading a *.bc file.

Constituent Properties

See ADH Transport Constituents for interface information.						
		Read	Edit	Write		Notes
CN	CLA	X	X	X		SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	CON	X	X	X		SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	SAL					
CN	SLT	X	X	X		SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	SND	X	X	X		SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	TMP					
CN	VOR	X	X	X		

Friction Controls

See Boundary Condition Assignment for interface information.						
		Read	Edit	Write		Notes
FR	ERH					
FR	MNG	X	X	X		A natural boundary condition without a specified friction card will have a roughness value of zero.
FR	SAV					
FR	URV					

Iteration Parameters

See Model Control Model Parameters for interface information.						
		Read	Edit	Write		Notes
IP	FLI	X	X	X		This card and the IP MIT card are mutually exclusive.
IP	FNI	X	X	X		This card and the IP NIT card are mutually exclusive.
IP	ITL	X	X	X		
IP	MIT	X	X	X		This card and the IP FLI card are mutually exclusive.
IP	NIT	X	X	X		This card and the IP FNI card are mutually exclusive.
IP	NTL	X	X	X		
IP	RTL					
IP	SST					

Material Properties

See Model Control Global Material Parameters and Material Properties for interface information.						
		Read	Edit	Write		Notes
MP	CBA					
MP	COR	X	X	X		Material specific.
MP	DF					Material and transport constituent specific.
MP	DTL	X	X	X		Global property.
MP	EEV	X	X	X		Material specific.
MP	EVS	X	X	X		Material specific.
MP	G	X	X	X		Global property.
MP	ML	X	X	X		Material specific.
MP	MU	X	X	X		Global property.
MP	MUC	X	X	X		The ADH units system flag currently keys off of the current horizontal coordinate system.

MP	NBL					
MP	RHO	X	X	X		Global property.
MP	SBA					
MP	SBM					
MP	SBN					
MP	SRT	X	X	X		Material specific.
MP	TRT					Material and transport constituent specific.

Operation Parameters

See Model Control Model Parameters for interface information.

		Read	Edit	Write		Notes
OP	BLK	X	X	X		
OP	BT	X	X	X		
OP	BTS					
OP	INC	X	X	X		
OP	PRE	X	X	X		
OP	SW2	X	---	X		Always written by SMS (not editable), denotes a 2D Shallow Water ADH model
OP	TPG					
OP	TRN					
OP	TEM					

Output Controls

See Model Control Output for interface information.

		Read	Edit	Write		Notes
END		X	---	X		Always written by SMS (not editable), denotes the end of the file.
FLX		X	X	X		See Boundary Condition Assignment for additional interface information.
OC		X	X	X		

Solution Controls

See Boundary Condition Assignment for nodal and nodestring boundary condition interface information.

		Read	Edit	Write		Notes
DB	LDE	X	X	X		
DB	LDH	X	X	X		
DB	LID	X	X	X		
DB	OVH	X	X	X		
DB	OVL	X	X	X		

DB	TRN					
EQ	TRN					
NB	DIS	X	X	X		
NB	OTW	X	X	X		
NB	OVL	X	X	X		See the Boundary Condition Assignment section of Material Properties for additional interface information.
NB	TRN					
OB	OF	X	---	---		This card is obsolete as the functionality is now handled internally in ADH.
OFF		X	X	X		This card can only be applied to a material string (MTS). See the Properties section of Material Properties for interface information.

String Structures

See Practical Notes for string structure information.						
		Read	Edit	Write		Notes
EGS		X	X	X		Use the Select Nodestring tool, and assign boundary conditions .
FCS						
MDS		X	X	X		Use the Select Nodestring tool, and assign boundary conditions for flow output.
MTS		X	X	X		Add and delete materials in the Materials Data window. Specify material properties , and using the Select Element tool, assign material type from the Elements | Assign Material Type... menu command.
NDS		X	X	X		Use the Select Node or Select Nodestring tool, and assign boundary conditions .

Time Controls

See Model Control Time for interface information.						
		Read	Edit	Write		Notes
TC	IAC	X	X	X		
TC	IDT	X	X	X		
TC	NDP	X	X	X		
TC	SDI	X	X	X		
TC	STD	X	X	X		
TC	STH					
TC	TF	X	X	X		
TC	T0	X	X	X		SMS writes a "global time" character string in quotes preceding the regular card fields. The date/time information is used to relate the simulation with other data loaded in SMS concurrently.

Time Series

		Read	Edit	Write		Notes
XY1		X	X	X		Define series (curves) in various places. An output times series using the auto-build specifier can be read in and created, however, the auto-build format is not currently used when writing out. SMS writes a series name character string in quotes preceding the regular card fields.
XYC						
XYT		X	X	X		Available where series (curves) are defined as "curve fit tolerance".

Related Topics

ADH

6.1.b. ADH Model Control

ADH Model Control

The *ADH Model Control* dialog contains the graphical prompts for specifying model options. The dialog is accessed through the *ADH* | **Model Control...** menu item.

Dialog Description

Model control contains the following pages:

- Model Parameters
- Iterations
- Time
- Output
- Global Material Properties
- Advanced (Cards)
- Solver

ADH Model Control Output

This page window is accessed from the Model Control window by clicking on the *Output* tab.

ADH will write solution data at startup and at each time step specified in a XY1 series definition and referred to by the OC card. Flux data will be included with the solution data if output flow strings are specified (see assigning boundary conditions).

ADH Model Control Advanced (Cards)

Model options not available in the interface can be specified in the ADH model control under the advanced tab. Each row in the spreadsheet will be saved as a different line in the *.bc file. The cards can be inserted and deleted using the insert and delete rows.

- The lines will be written out exactly as typed.
- The lines will be written out in the order in which they appear in the dialog.
- Any lines encountered in a *.bc file that is not recognized will appear in this list.

- Lines beginning with a "!" character are treated as comment lines and will be skipped over during a read.
- Comment lines are automatically generated and will not be preserved.
- All of these advanced cards will be grouped together at the bottom of the file regardless of where they appeared during reading.

Dialog Description

The following controls specify how often solution data will be written by ADH and the number of flux boundaries present.

Output Times

List box contains the times to be contained in the output series. Multiple selection is enabled for deleting times.

Add by specifying a range / Add individual output time radio group specifies time information to be added. The *Add by specifying a range* control group includes Start at positive integer/real number field and units combo box, End at positive integer/real number field, and *Increment* positive integer/real number field and units combo box. The *End* at time units are the same as specified for the *Start* at time. The Add individual output time control includes a positive integer/real number field and units combo box. Each edit field allows for positive integer seconds or positive real number minutes, hours, etc., with six decimal places. All times are rounded to the nearest whole second (a decimal minute is rounded to the nearest decimal minute with a whole second equivalent) before being added.

Add button includes the time information from the above radio group in the times list box. Duplicate times will be automatically removed from the list.

Delete button removes the selected time(s) from the list box.

Delete All buttons clears the entire times list without regard to the current selection.

View output times in combo box specifies the time unit the list box will display the output times in.

Specify curve fit tolerance check box and positive integer edit field specifies the series tolerance when checked.

Flow

Number of nodestrings specified to output flow calculated across themselves displays the total number of ADH flux boundaries included in the simulation.

Related Topics

ADH

ADH Model Control Model Parameters

This window is part of the *ADH model control* dialog (*ADH* | **Model Control...** command). It is accessed from the *Model Control* window by clicking on the *Model Parameters* tab.

Model Parameters

The model parameters window is divided into three sections including:

- Wind Stations
- Density effects
- Bendway correction

Each of these sections are described below.

Density effects

The ADH model includes the ability to simulate the effects of salinity and temperature on flow. This section includes two check boxes to activate these options.

Salinity (CN SAL card)

The *Include salinity* check box with the associated reference concentration allows turning on the simulation of salinity and define a positive non-zero real number for the background or reference concentration . The concentration is specified in parts per thousand.

Temperature (CN TMP card)

The *Include temperature* check box with the associated *Reference temperature* allows turning on the inclusion of temperature effects in the simulation of specify a positive non-zero real number which correlates to the CN TMP card reference temperature. The temperature is measured in degrees.

Bendway Correction (CN VOR card)

The SW2 option in ADH simulates two-dimensional flow. However, some three dimensional effects can be approximated. This includes a method for correcting for the effects of vorticity around bends. Selecting the *Include vorticity* check box will cause SMS to add this card and activate this feature. It also adds the OP TRN card since vorticity is simulated in ADH as a transport constituent.

When the feature is active, it's necessary to specify three coefficients. These include the As and Ds empirical coefficients determined by integrating against measured values (defaults of 5.0 and 0.5) and a normalization factor.

The *Override empirical coefficients* toggle activates the edit fields for the other coefficients. If the option is unchecked, the coefficients will be reset to the default values. Zero values for both coefficients will assume the default values. *Currently it is not recommended to override the empirical coefficients.*

Wind Stations (XYC cards)

Wind can be included in a number of ways in an ADH simulation. These include on a material by material basis described in a different section. The *Include wind stations (defined on spatial data coverage)* check box specifies that wind shear stress should be included in the simulation, computed from a set of spatially defined wind stations. This check box is only available if at least one spatial data coverage exits. In the spatial data coverage define a set of wind gages (defined as feature points) with time series of wind data associated.

If more than one spatial data coverage is defined, the associated **Select...** button undims and a single set of wind data can be selected for the current simulation. Clicking on the **Select...** button opens the *Select Coverage* window which lists all available spatial data coverages.

SMS will write an XYC card for each wind station. This card defines the curve containing the wind data and the location of the station.

Specify curve fit tolerance check box and the associated positive integer edit field specify whether a tolerance for all wind time series is included and its value. These controls are enabled only if a coverage is selected (the tolerance check box must also checked to enable the edit field).

For more information see Wind Stations .

Related Topics

- ADH Model Control

ADH Model Control Iterations

This window is part of the *ADH model control* dialog (*ADH* | **Model Control...** command). It is accessed from the *Model Control* window by clicking on the *Iterations* tab.

Iterations

The iterations tab allows specifying the level of precision for the conservation of mass and momentum as ADH performs its calculations.

ADH solves conservation statements concerning water volume, momentum, and constituent mass. ADH is written in conservative form and thus can be regarded as both a finite element method and also a finite volume method. As such ADH can be shown to represent a sum of fluxes around the edge of an element to be in balance with the mass or volume change within the element. In the case of the momentum equation it will be a sum of fluxes and forces balancing the momentum change within an element.

ADH computes the water levels and velocities at each computation point in the domain for each time step in the simulation in an iterative process. At the end of each iteration, calculations are made to determine how much the solution is changing and what the errors in conservation are. If the errors are small, and the solution is not changing, the process is said to be converged and calculations can proceed to the next time step. If the solution is changing and/or the errors in conservation are large, ADH can attempt another iteration.

There are several iteration parameter controls in ADH. These are described in detail in the ADH users manual . The Solver options tab controls the linear iterations. This dialog controls the non-linear iteration parameters.

At the end of each iteration ADH computes the conservation properties for the current approximations of water depth and velocity. If the errors are less than specified tolerance, the solver moves on to the next time step. If the errors are greater than the specified tolerance, but still reasonable, and the maximum number of iterations specified on the IP NIT card has not been reached, ADH will attempt another iteration. If the errors have become unreasonable (divergence), ADH will exit the loop with a failure notification. The controls in this dialog allow defining how ADH should determine convergence and how to proceed when convergence is not reached.

Maximum number of iterations per time step (IP NIT and IP FNI cards)

In this edit field specify the maximum number of iterations that ADH will try for a single time step. By examining the output for successive iterations, determine if the process is converging in general (trending towards zero), or oscillating. If the trend is towards convergence, this maximum number of iterations can be increased. As the value increases, ADH can spend more time processing individual time steps which may decrease the overall efficiency of the solver. For this reason, do not simply increase this parameter without verifying that it will be beneficial.

SMS writes either the IP NIT or IP FNI card based on the setting of the If tolerance is not satisfied radio group.

- IP NIT card is written if the Reduce time step option is selected. (Recommended)
- IP FNI card is written if the Accept solution option is selected.

The IP FNI card instructs ADH to continue with the solution as if it had converged. This is not recommended.

When the *Reduce time step* option is selected and the solution has diverged or the maximum number of iterations have been attempted, ADH will reduce the time step size to ¼ of the previous and continue the calculations.

When the *Accept solution* option is selected, and the maximum number of iterations have been attempted, ADH proceeds as if convergence has been reached to the next time step. (This option is not recommended.)

NTL/ITL/Both combo box

NTL (IP NTL card)

If the NTL option is selected, ADH will check for non-linear convergence on the "maximum residual norm". An initial estimate for an appropriate tolerance value for a problem can be computed as:

This is the maximum allowance. It will likely be desirable to set the tolerance higher than this estimate so the simulation will progress.

ITL (IP ITL card)

If the *ITL* option is selected, ADH will check for non-linear convergence on the "maximum increment norm" or the change in the solution (velocity, depth and concentration).

Both

If the *Both* option is selected, SMS will write both cards to the BC file and ADH will check for convergence on both terms. Both must be satisfied for the solution to proceed.

Related Topics

ADH

ADH Model Control Time

This page window is accessed from the *Model Control* window by clicking on the *Time* tab.

ADH is set up as a transient conditions model, however, a steady state simulation can be invoked which utilizes an iterative process to converge to a solution. A steady state solution is accepted if the end time of the model is reached or the iteration tolerance requirement (see Model Parameters) is met. It's required to ensure that all series data are acceptable (constant linear series) if the steady state method is used. ADH also includes a quasi-unsteady method which strings multiple steady state simulation together to form a step function style hydrograph.

Due to the nature of a steady state simulation, constituent transport and bed layers cannot be included in the simulation. If any constituent (and bed layer) is specified in a dynamic simulation and then the simulation is changed to steady state, SMS will clear the constituent data (and bed layer specification). A message will warn of this effect when the steady state simulation type is selected and constituent data (and a bed layer) exists. Another message will state that SMS has cleared the data (if steady state is still selected) when switching to another tab or exiting Model Control by selecting OK.

Since series data specified for a dynamic simulation is usually inappropriate for a steady state simulation, a change in simulation type from dynamic to steady state will force SMS to clear all previously specified series data when changes are accepted in Model Control. This data is usually associated with the boundary conditions of materials, nodes, and nodestrings. A message will warn of this effect when the simulation type is changed and series data exists. Another message will state that SMS has cleared the data (if the simulation type remains changed) upon accepting the changes (exiting Model Control).

Time

The following controls specify the timing parameters for the ADH model run.

Simulation

Dynamic / Steady state / Quasi-unsteady radio group specifies the simulation type.

- *Dynamic* option provides the following:
    - *Start* date and time field and the *Delay model start* check box with positive real number and units combo box correlates to the TC T0 card.
    - *End* date and time field (with the defined start) specifies the simulation run length on the TC TF card.
    - *Duration* positive real number edit field and units combo box also specifies the simulation run length. This field and the End field will update as the other is changed.
- *Steady state* option provides the following:
    - *Duration* positive real number edit field and units combo box to specify the simulation length.
- *Quasi-unsteady* option is currently unavailable.

Apply adaptive time control check box includes the TC IAC card or TC NDP card when checked or unchecked, respectively.

Time Step Control

Time step size curve button with the associated *Series time units* combo box, and *Specify curve fit tolerance* check box and positive integer edit field defines the data to be associated with the TC IDT card and its XY1 series definition card. Clicking on the curve button will open the XY Series Editor. This series must be defined for the entire model run length, so include or exceed the time *Model run time* listed in the *Simulation* group. This control group is available only when the *Dynamic* simulation type is selected.

Initial time step positive real number edit field and units combo box specifies the time step value of the TC STD card. This control group is available only when the *Steady state* simulation type is selected.

Specify extra sediment transport time steps check box and positive integer edit field while include the TC SDI card when checked. If sediment transport is included in the simulation and this control is not checked, then the model will calculate sediment according to the time increment specified by the *Time step size* series.

Dynamic simulation type option, in the Simulation group, provides the following:

- *Time step* size curve button selects the time series data to be associated with the TC IDT card. Clicking on the curve button will open the Time Series editor (limited to the Time Step Size curve group). The selected series should be defined for the entire model run length, so include or exceed the time Model run time listed in the Simulation group.
- *SNumber* of sediment transport time steps per hydrodynamic time step positive non-zero integer edit field correlates to the TC SDI card. This field is only enabled if sediments have been specified on the Transport Constituents tab of Model Control. This applies only to sediments and not any other constituents. If sediment transport is not included in the simulation, then the TC SDI card is not included in the file.
- *Max sediment time steps* value is associated with the TC SDI card. This allows specifying smaller sediment transport time steps that what is used for the hydrodynamic time step. Sediment constituents must exists otherwise this option will be grayed out and the card will not be written.
- *Automatic time step* option, provides the following:
- Specifies the time step data for the TC ATF card. Pressing the 'calculate' button calculates the time step based on the formula: $= x / \sqrt{g*h}$

Where:

x = mesh element length (currently the distance between the 1st and 2nd node in the 1st element)

g = gravitational acceleration

h = min water depth

If an initial dataset doesn't exist, the automatic time step option is disabled and the TC ATF card is not written.

Steady State simulation type option, in the Simulation group, provides the following:

- *Initial time step* positive real number edit field and units combo box specifies the time step value of the TC STD card. This control group is available only when the Steady state simulation type is selected.

Related Topics

- ADH Model Control

ADH Model Control Output

This window is part of the *ADH model control* dialog (*ADH* | **Model Control...** command). It is accessed from the *Model Control* window by clicking on the *Output* tab

ADH will write solution data at startup and at each time step specified in a XY1 series definition and referred to by the OC card. Flux data will be included with the solution data if output flow strings are specified (see assigning boundary conditions).

Output Control

The following controls specify how often solution data will be written by ADH and the number of flux boundaries present.

Output Times

List box contains the times to be contained in the output series. Multiple selection is enabled for deleting times.

Add by specifying a range / Add individual output time radio group specifies time information to be added. The *Add by specifying a range* control group includes *Start at* positive integer/real number field and units combo box, *End at* positive integer/real number field, and *Increment* positive integer/real number field and units combo box. The *End at* time units are the same as specified for the *Start at* time. The *Add individual output time* control includes a positive integer/real number field and units combo box. Each edit field allows for positive integer seconds or positive real number minutes, hours, etc with six decimal places. All times are rounded to the nearest whole second (a decimal minute is rounded to the nearest decimal minute with a whole second equivalent) before being added.

Add button includes the time information from the above radio group in the times list box. Duplicate times will be automatically removed from the list.

Delete button removes the selected time(s) from the list box.

Delete All buttons clears the entire times list without regard to the current selection.

View output times in combo box specifies the time unit the list box will display the output times in.

Specify curve fit tolerance check box and positive integer edit field specifies the series tolerance when checked.

Flow

Number of nodestrings specified to output flow calculated across themselves displays the total number of ADH flux boundaries included in the simulation.

Geometry

Output adapted mesh files check box specifies whether geometry files will be saved as ADH adapts the mesh. The geometry will be outputted at the same interval specified in the list box for the solution data. This correlates to the PC ADP card.

Related Topics

- ADH
- ADH Model Control

ADH Model Control Global Material Properties

This window is part of the *ADH model control* dialog (*ADH* | **Model Control...** command). It is accessed from the *Model Control* window by clicking on the *Global Material Properties* tab.

ADH contains parameters that apply to all materials and parameters that are specific to (and must be defined for) each material. The parameters found within this tab are of the former type.

The bed layers specified here are applied to all mesh nodes and can only be overwritten at the nodal level in the *Bed Layers Assignment* window. When bed layers are created or deleted here, nodal bed layers are defaulted or deleted, respectively.

Global Material Properties

Parameters (Untitled group box)

Enable wetting/drying limits check box and the associated *Dry depth* and *Wet depth* positive real number edit fields correlate to the optional MP DTL card. If the water depth of a node is below the *Dry depth* , then the node is completely dry and is not included in the ADH shallow water equations. If the water depth is above the wet depth, then the node is included in the equations. A water depth which falls within these two depths will include the node in the equation with a restricting factor.

Constant

Uniform background viscosity positive real number edit field correlates to the MP MU card.

Gravitation acceleration positive real number edit field correlates to the MP G card.

Density positive real number edit field correlates to the MP RHO card.

Wetting and Drying

Enable shock capturing/stability procedures below depth is check box and the associated wetting and drying process. When this toggle is selected, ADH performs extra calculations for all cells with depth values less than the specified minimum to stabilize the process. As the specified depth value increases, extra calculations also increase and model performance decreases. When this option is selected, the *Depth* edit field is enabled by SMS. In this field specify a positive real number correlating to the optional MP DTL card.

Sediment Bed layers

Number of bed layers positive integer edit field correlates to the MP NBL card. The value specified is the number of bed layer column provided in the spreadsheet. If the value is increased, the new bed layer columns are appended; if decreased, the existing bed layer columns are removed. The effected bed layers are always the highest numbered layers (at the end of the spreadsheet). This is enabled only if sediments have been defined on the *Transport Constituents* tab.

Layer 1 is the deepest (or bottom) layer; the highest numbered layer is exposed to the stream flow. The layer IDs cannot be edited and bed layering order is fixed to ascending layer ID order (See Sediment Transport and Bed Layers).

Edit spreadsheet column toolbar is enabled when a cell selection exists in the spreadsheet, and provides:

- Insert New Column Before tool inserts a single new bed layer column before (to the left of) the current selection.
- Delete Column(s) tool removes all of the bed layer columns of the currently selected cells.

Spreadsheet contains the defined bed layers as columns and bed layer specifications as rows. The bed layers begin with Layer 1 and proceeds by ascending order by ID to the right. This spreadsheet will be empty if no sediments have been defined on the *Transport Constituents* tab or zero bed layers are specified. The specification rows are as follows:

- Sediment distribution – Title cell with each defined sediment listed below in its own row. The row contains Normalize buttons for each bed layer. Clicking on the button will normalize the distribution of each sediment such that the total distribution equals 100%.
- Sediment (name) – Consists of this sediment's distribution (as a percentage) for each layer. A sediment row is provided for all defined sediments. The value must be a positive real number and correlates to the MP SBA card.
- Total – Displays the current total specified distribution of all sediments for each layer. The row updates automatically as specifications change and cannot be edited. Each layer's total sediment distribution must equal 100%.
- Thickness – Consists of each bed layer's thickness measured in meters. The value must be a positive real number and correlates to the MP SBA card.
- Bulk Density – Consists of each bed layer's bulk density measured in kilograms per cubic meter. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.
- Erosion crit. shear – Consists of each bed layer's critical shear stress for erosion measured in newtons per square meter. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.
- Erosion rate constant – Consists of each bed layer's unitless erosion rate constant. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.
- Erosion rate exponent – Consists of each bed layer's unitless erosion rate exponent. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.

Related Topics

- ADH Model Control

ADH Model Control Advanced

This window is part of the *ADH model control* dialog (*ADH* | **Model Control...** command). It is accessed from the *Model Control* window by clicking on the *Advanced* tab.

Advanced Cards

SMS includes support for many of the features in ADH, however, this list of features is a dynamic set. The user base of ADH includes a development team that is constantly experimenting with new options.

This window allows SMS to maintain the options that are not otherwise supported by the interface. For example, when SMS reads the BC file for an ADH simulation, and encounters a card that it does not recognize, that card is stored, verbatim, to a list of "Advanced Cards". That list is displayed in this window.

Enter cards in this dialog to experiment with other new features in the model. As those features are added to the list of supported features, they would automatically be moved to their own location in the model control the next time SMS reads the BC file.

Unsupported or advanced cards are written at the end of the BC file, just before the END card. This relies on the attribute of ADH that does not require any order dependence in the BC file.

Related Topics

- ADH
- ADH Model Control

ADH Model Control Solver

This window is part of the *ADH Model Control* dialog (*ADH* | **Model Control...** command). It is accessed from the *ADH Model Control* window by clicking on the *Solver* tab.

(As noted at the top of this dialog, this is an advanced feature. It is recommended to only use these options when working with the ADH development team.)

Operation

The operation section contains controls to specify general operation parameters for ADH. SMS automatically writes the OP SW2 card to all ADH boundary condition files because SMS interfaces with the 2D shallow water problems simulated by ADH. The other operational parameters specified in this section include:

- *Increment memory allocation block size*

  The memory used when running the ADH model is allocation in blocks. The size of the block (OP INC card) can be specified here. The default value is 40 units. If the specified number is too small, running ADH may be slowed as the model will search for additional memory. If the number is too large, the model run will use excess memory not needed to complete the run.

- *Iterative solver pre-conditioner*

  The *Iterative solver pre-conditioner* parameters apply to parallel processing applications. If running ADH in single processor mode, this card has no impact.

  If running ADH in parallel mode, the edit field defines how many blocks per processor are to be used in the preconditioning which subdivides the problem for each processor. Provides the following options: "One-level Additive Schwarz", "Two-level Additive Schwarz", and "Two-level Hybrid". After selecting a preconditioning scheme, specify how many blocks per processor are to be used in the preconditioner by completing the *Block specification* field.

- *Include vessel stress effects*

  This toggle instructs SMS to write the OP BTS card which includes vessel stress effects check in the simulation when checked. This calculates and outputs bed shear stresses due to vessels in dyn/cms. This currently requires use of metric units. If enabling this option, make sure there are vessel coverages in the simulation and that each boat has propellers defined.

Linear Iterations By Solver

This window includes the options to control linear iterations in ADH. The controls are similar to the controls for non-linear iterations included on the Iterations tab.

The distinction between linear and non-linear iterations are described in some detain in the ADH users manual.

Maximum number of iterations (IP MIT and IP FLI cards)

In this edit field specify the maximum number of linear iterations that ADH will try for each non-linear iteration.

SMS writes either the IP MIT or IP FLI card based on the setting of the *If internal linear tolerance is not satisfied* radio group.

- IP MIT card is written if the *Stop the solution* option is selected. (Recommended)
- IP FLI card is written if the *Proceed to the next non-linear iteration* option is selected.

The IP FLI card instructs ADH to continue with the solution as if it had converged. This is not recommended.

When the *Stop the solution* option is selected and the solution has not converged at the maximum number of iterations have been attempted, ADH will treat the solution as diverged and the non-linear iterations will be aborted.

Related Topics

- ADH
- ADH Model Control

6.1.c. ADH Library Control

ADH Sediment Library Control

The *ADH Sediment Library Control* contains the graphical prompts for specifying model options. The dialog is accessed through the *ADH* | **Sediment Library Control** menu item.

To make use of the *ADH Sediment Library Control* , it is required that the ADH model be in metric units. Otherwise, the model will encounter an error.

Dialog Description

Consolidation

See ADH Consolidation for more information.

Transport Constituents

See ADH Transport Constituents for more information.

Global Materials Sed Props

The *Global Materials Sediment Properties* tab sets the number of of bed layers. Once the *Number of bed layers* field has been completed, the **Sediment Bed Layers...** button will become active. Clicking this button will bring up the *Sediment Bed Layers* dialog.

See ADH Sediment Transport and Bed Layers for more information.

Sediment Bed Layers Dialog

The *Sediment Bed Layers* dialog permits changing bed layer values. The following values can be viewed or changed:

- *Layer* – Gives the layer name. Changes to this field will not be saved. Layers are ordered with the lowest layer number at the bottom.

- **Distribution** – This button will open the *Distribution* dialog. In the *Distribution* dialog, the material percentages in the bed layer can be adjusted.
- *Thickness (m)* – The thickness of the bed layer given in meters.
- *Bulk Denisity* – Measured in kilograms per cubic meters.
- *Erosion crit. shear* – Critical shear stress for erosion is used in the erosion flux equation. Measured in Newtons per square meters.
- *Erosion rate constant* – Used in the erosion flux equation.
- *Erosion rate exponent* – Used in the erosion flux equation.

Sediment Properties

This tab has five options.

- *Non-cohesive sediment entrainment*

  Values of Option include: "Garcia-Parker", "Wright-Parker", "Van Rijn" or "C2Shore". When this option is selected it will write out the SP NSE card.

- *Non-cohesive bedload entrainment*

  Values of Option include: "Van Rijn", "Meyer Peter Mueller", "Meyer Peter Mueller w/Wong Parker Correction", or "C2Shore". When this option is selected it will write out the SP NBE card.

- *Non-cohesive hiding factor*

  Values of Option include: "Karim Holly Yang" or "Egiazaroff". When this option is selected it will write out the SP HID card.

- *Cohesive settling velocity*

  Values of Option include: "Free settling" or "Hwang and Mehta". When this option is selected it will write out the SP CSV card.

- *Bed shear stress due to wind wave*

  Values of Option include: "No applied wind-wave stress", "Grand and Madsen", or "Teeter". When this option is selected it will write out the SP WWS card.

CS2SHORE Sediment

This tab has ten options.

- *Median sediment grain size* with card SP C2SD50
- *Breaking efficiency* with card SP C2SEB
- *Bottom dissipation efficiency* with card SP C2SEF
- *Bed porosity* with card SP C2SBP
- *Sediment fall velocity* with card SP C2SWF
- *Sediment specific gravity* with card SP C2SSG
- *Suspended load parameter* with card SP C2SSL
- *Wave-related bed load parameter* with card SP C2SBLW
- *Current-related bed load parameter* with card SP C2SBLC
- *Wave friction factor* with card SP C2SFW

Related Topics

- ADH
- SMS:ADH Model Control

ADH Consolidation

This window is part of the *ADH model control* dialog (*ADH* | **Sediment Library Control...** command). It is accessed from the *ADH Sediment Library Control* window by clicking on the *Consolidation* tab.

Consolidation

This window consists of a table defining the properties of cohesive sediments included in an ADH simulation.

The sediment classes must be specified before entering these time-based parameters for each sediment. The check box labeled Use time-based consolidation at the bottom of the page tells ADH that consolidation should be computed.

The spreadsheet contains the parameters and values specific to the currently selected constituent in the list. Constituent parameters are:

- *Time* (sec) – Measured in seconds.
- *Bulk density* (clay/silt type only) – Measured in kilograms per cubic meters.
- *Erosion crit. shear* (clay/silt type only) – Critical shear stress for erosion is used in the erosion flux equation. Measured in Newtons per square meters.
- *Erosion rate constant* (clay/silt type only) – Used in the erosion flux equation.
- *Erosion rate exponent* (clay/silt type only) – Used in the erosion flux equation.

All value fields are restricted to positive non-zero real numbers unless stated otherwise.

Related Topics

- Consol
- ADH Sediment Library Control

ADH Transport Constituents

This window is part of the *ADH Sediment Library Control* dialog (*ADH* | **Sediment Library Control...** command). It is accessed from the *ADH Sediment Library Control* window by clicking on the *Transport Constituents* tab.

ADH transport constituents consist of two types: regular and sediment. Transport is only available for a normal (dynamic) simulation (specified on the *Time* tab of *Model Control*).

Constituents

Transport Constituents

- General – CN CON
- Salinity – CN SAL
- Temperature – CN TMP
- Vorticity – CN VOR

Sediment Transport Constituents

- Sand or gravel – CN SND
- Clay or silt – CN CLA

Non-sediment Transport Constituents

- Salinity – CN SAL (This is a currently unavailable option)
- Temperature – CN TMP (This is a currently unavailable option)
- Vorticity – CN VOR

Dialog Description

Sediment

List box contains type and name of sediments to be included in the model simulation.

New button inserts a defaulted general sediment into the sediment list.

Delete button removes the currently selected sediment from the list.

Spreadsheet contains the parameters and values specific to the currently selected sediment in the list. Available parameters will be updated when the sediment type is changed. The sediment list box will update when the sediment name or type change.

Bendway Correction

Include vorticity check box and the associated *Normalization factor* , *As coefficient* , and *Ds coefficient* positive real number edit fields correlates to the CN VOR card.

Generic and Sediment

List box contains type and name of user-defined constituents to be included in the model simulation.

New button inserts a defaulted generic constituent into the user-defined constituent list.

Delete button removes the currently selected constituent from the list.

Spreadsheet contains the parameters and values specific to the currently selected constituent in the list. Constituent parameters are:

- Constituent type – Options are: Generic, Sand or gravel sediment, and Clay or silt sediment
- Name – Within SMS, the constituent will be referenced by this name. Names must be unique.
- Characteristic concentration – The characteristic concentration of the constituent is measured in micromass per unit mass or parts per million.
- Grain diameter (sand/gravel and clay/silt types) – Measured in millimeters (this is converted to meters for ADH).
- Specific gravity (sand/gravel and clay/silt types) – Value must be greater than or equal to 1.0.
- Grain porosity (sand/gravel type only)
- Bulk density (clay/silt type only) – Measured in kilograms per cubic meters.
- Erosion critical shear (clay/silt type only) – Critical shear stress for erosion is used in the erosion flux equation. Measured in Newtons per square meters.
- Erosion rate constant (clay/silt type only) – Used in the erosion flux equation.
- Deposition critical shear (clay/silt type only) – Critical shear stress for deposition is measured in Newtons per square meters.
- Settling velocity (clay/silt type only) – Measured in meters per second.

The constituent list box will update when the constituent name or type changes. If the type of a constituent is changed, the name and characteristic concentration are retained, but all other values are cleared. All value fields are restricted to positive non-zero real numbers unless stated otherwise.

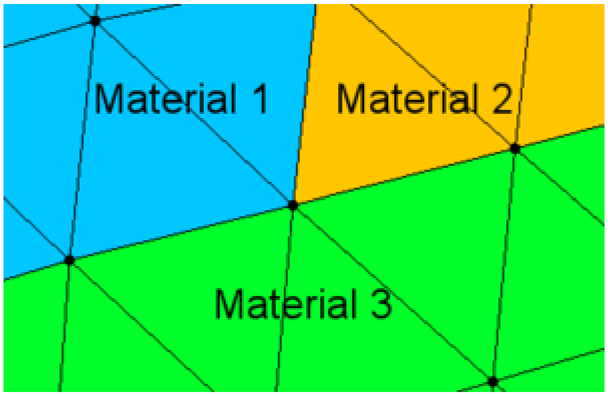
Related Topics

ADH Model Control

ADH Sediment Transport and Bed Layers

Sediments properties defined on the CN CON, CN SND, and CN CLA cards describe the properties of the sediment particle when it is first deposited.

Bed layers specified for a simulation are previously deposited layers defined by the MP SBA, MP SBM, and MP SBN cards. While running ADH, the previously deposited layers are aggraded and degraded and the solution datasets provide the resulting bed profile conditions during the simulation. For sediment transport with cohesive sediments, bed layers also have cohesive layer attributes defined by the MP CBA, MP CBM, and MP CBN cards to account for settlement and compaction of the previously deposited layer.



ADH provides the MP SBM and MP CBM cards to apply a bed layer specifications to nodes (which overwrites the global specifications) by specifying a material, however conflicts can arise since material type is an attribute of an element and not a node. As depicted in the image, a node can define the boundary between multiple materials and if each material has an associated bed layer specification, the central node can have three different specifications. ADH will assign material bed layer specifications by applying them to nodes and overwritting prevoius specifications in the order the cards are read from the *.bc file. Therefore, if MP SBM (or MP CBM) cards for material 1, material 2, and material 3 are read in that order, the central node (in the picture) will finish with the specification for material 3. SMS will read the *.bc file by the same process, however, bed layer specifications by material type will not be allowed in the interface (in Material Properties) and not written to file since materials are not currently prioritizable. Bed layers can be specified to nodes using a material type by selecting the *Edit* | **Select by Material Type...** menu item with the **Select Node** tool active, but the material type is only used to filter the selection and is not associated to the actual node or bed layer specification. SMS will only write MP SBA, MP CBA, MP SBN, and MP CBN cards to file.

Bed layers are in continuous ascending ordered by ID starting with 1 as the deepest specified layer in the profile and the highest numbered layer at the top, exposed to the stream flow. ADH automatically adds a solid boundary, consisting completely of an extra immobile (non-erodable) sediment, below Layer 1 which is not presented in the interface but will be included in solution files.

ADH Bed Layering Order
Stream Flow
Layer X
...
Layer 3
Layer 2
Layer 1
Solid Boundary

Bed layers specifications can be edited using the following:

- *ADH* | **Sediment Library Control...** menu item opens the *ADH Sediment Library Control* window where global bed layers are specified on the *Global Material Properties* tab.
- *ADH* | **Assign Bed Layers...** menu item opens the *ADH Bed Layers Assignment* window for specifying bed layers at the nodal level. Available only with a current node selection.
- Using the **Select Node** tool right-click menu *Bed Layers...* | **Assign...** command opens the *ADH Bed Layers Assignment* window for specifying bed layers at the nodal level. Available only with a current node selection.

- 'Using the **Select Node** tool right-click menu *Bed Layers...* | **Delete...** command deletes the nodal specification which overwrites the global. Available only when the current node selection includes at least one nodal bed layer specification.
- Using the **Select Node** tool right-click menu *Bed Layers...* | **Find Equivalent** command selects all nodes of the mesh that are equivalent to a bed layer specification of the current node selection.

Related Topics

- ADH
- ADH Sediment Library Control
- ADH Bed Layers Assignment

6.2. Finite Element Surface Water Modeling System (FESWMS)

FESWMS

FESWMS	
Model Info	
Model type	Two-dimensional finite element surface water computer program that can compute the direction of flow and water surface elevation in a horizontal plane. Also has the ability to model hydraulic structures commonly used by hydraulic engineers.
Developer	David C. Froehlich, Ph.D., P.E.
Web site	FESWMS page
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • FESWMS • FESWMS Steering • FESWMS Weirs

The Finite Element Surface Water Modeling System (FESWMS) consists of multiple modules used to simulate surface-water flow in a two-dimensional horizontal plane. The SMS includes an interface for the FST2DH (Flow and Sediment Transport) module of FESWMS. FESWMS is sponsored by the Federal Highway Administration. David C. Froehlich, Ph.D., P.E. originally developed FESWMS for the United States Department of Transportation Federal Highway Administration (FHWA) and the United States Geological Survey (USGS) . The FHWA has continued to maintain and sponsor development of subsequent versions, which continue to incorporate features specifically designed for modeling highway structures in complex hydraulic environments.

The FESWMS model can be added to a paid edition of SMS.

Functionality

FST2DH is the FESWMS program that performs the two-dimensional hydraulic computations in surface-water bodies. FST2DH can perform either steady state or dynamic flow modeling and provides analysis of highway crossings and structures, including bridges, culverts, weirs, roadway embankments, and drop-inlet spillways. FST2DH simulates the movement of water and non-cohesive sediments in rivers, estuaries, and coastal waters by applying the finite element method to solve steady-state or time-dependent systems of equations that describe two-dimensional depth-averaged surface-water flow and sediment transport. [4]

4 Sept. 2002 - User's Manual for FESWMS Flo2DH - Publication No. FHWA-RD-03-053

Using the Model / Practical Notes

- FESWMS Spindown Steering (Incremental Loading)
- The representation of the wind direction in the FESWMS Manual is incorrect. The corrected image is shown in the *FESWMS Model Control* dialog of the SMS interface.
- FESWMS known model issues
- Support Forum FESWMS with Sediment Transport Tips User Post
- Transform the mesh so it is close to the origin (0,0). Large x, y coordinates can be problematic for FESWMS.
- Verify mesh does not have incorrect elevation values (often due to extrapolation). Try looking at the mesh in a rotated view.

General Steps to build a FESWMS Model

Various sequences of steps can be used to create and edit a two-dimensional model in SMS. A suggested methodology is provided below.

1) Import background data consisting of:
    1) Elevation data covering the area to be modeled. See Scatter Module
    2) Open and register background image(s) if desired. See Images
2) Create a FESWMS coverage and define the conceptual model that will be used to generate the finite element mesh. See Map Module
    1) Define the physical boundaries of the model
    2) Build feature polygons within the model domain. See Build Polygons in the Feature Objects menu.
    3) Assign mesh types to the feature polygons and adjust the density and spacing of vertices (i.e., redistribute vertices) as necessary to achieve appropriate element shapes and sizes
3) Creata an area properties coverage and assign material types. This is an optional step. The FEWSMS coverage can also be used to define material regions. See Map Module
4) Generate a finite element mesh from the conceptual model. See Map Module
5) Check the final quality of the finite element mesh and edit the mesh as needed. See 2D Mesh Module
    1) Designate element types (i.e., linear or quadratic; triangular or quadrilateral)
    2) Review the mesh quality
6) Specify FESWMS element attributes (i.e., material properties) such as Manning's or Chezy roughness factors and eddy viscosity to the material types. See 2D Mesh Module
7) Create nodestrings and assign model boundary conditions not included in the conceptual model. See 2D Mesh Module
8) Renumber the finite element mesh.
9) Specify initial conditions and model control parameters. See FESWMS Model Control
10) Run the model.
11) View model results. If necessary, troubleshoot error/warning messages generated during the model run and revise the model accordingly.

Graphical Interface

SMS provides a graphical interface that is designed to visualize the projects being created, easily modify project parameters, and view the solutions produced by the FESWMS model. See FESWMS Graphical Interface for more information.

The FESWMS Graphical Interface contains tools to create and edit an FESWMS simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to FESWMS. If a mesh has already been created for a FESWMS simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to FESWMS. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the *FESWMS* Menu . See FESWMS Graphical Interface for more information.

Saving FESWMS

When completing a *File* | **Save As...** command the following files get saved in the *.sms.

- *.fpr referenced to new save location
- *.mat referenced to new save location
- *.flo referenced to original location unless reran then moves to saved location.

External Links

- FESWMS FST2DH User's Manual PDF (Revised Oct 2003)
- U.S. Department of Transportation Federal Highway Administration – Hydraulics Engineering Software Page [160]
- Ipson, Mark K. (2006). Analysis of the Sediment Transport Capabilities of FESWMS FST2DH. Thesis, Brigham Young University. [161]

Related Topics

- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- FESWMS Spindown
- FESWMS Sediment Transport
- FESWMS Known Model Issues

FESWMS Arc Attributes Dialog

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

- *None* – Leaves the arc type as unassigned.
- *Boundary Conditions* – Options button opens the *FESWMS Nodestring Boundary Conditions* dialog.
- *Continuity Check/Flux* –

Related Topics

- Feature Objects Menu

FESWMS BC Nodestrings

Generic 2D Mesh Boundary conditions are generally defined on feature arcs in the conceptual model or nodestrings on the 2D mesh. Boundary conditions constrain the water surface elevation and/or flow at the model boundary.

General Boundary Options

Constant vs Dynamic

FESWMS boundary conditions can be constant or dynamic. If a constant boundary condition is used with a dynamic simulation the boundary condition does not change during the simulation. It is not possible to have a dynamic boundary condition with a steady state simulation.

Total Flow String

The nodestring that is the total node flow string identifies the flowrate that will be considered 100% of the flow in the FESWMS printed output file (*.prt). Only one nodestring may be the total flow nodestring and it will be the last one identified as the total flow nodestring.

Boundary Condition Types

Specified Flow / WSE

This type of boundary constrains the flowrate and/or water surface elevation to fixed values. Both the flowrate and water surface elevation should only be specified for supercritical inflow boundary conditions.

A flow boundary condition can be a normal boundary or a weakly reflecting boundary. A weakly reflecting boundary can be used to allow tidal variation to affect the boundary. A weakly reflecting boundary simulates a fictitious river without frictional resistance upstream of the boundary.

Generally a fixed water surface elevation boundary is used for the downstream boundary condition of the model. The water surface elevation can vary from one end of the boundary to the end by using the vary along string option.

Supercritical Outflow

For supercritical boundaries, both the flowrate and water surface elevation is defined on the inflow boundary. Even though no data is associated with supercritical outflow boundaries, these boundaries must be identified so FESWMS will allow flow through the boundary.

Computed WSE

This is used to provide a rating curve (water surface vs flow) relationship to compute the downstream boundary condition. FESWMS can compute the rating curve using the friction slope option. Strangely enough FESWMS requires providing flow information to use this option.

Storm/Tide WSE

This is used to tell FESWMS that the boundary water surface elevation will be determined from the storm associated with the simulation.

Related Topics

- FESWMS Arc Attributes Dialog

FESWMS Executable Known Issues

There are several different builds of the FESWMS executable (fst2dh.exe). Unfortunately there are some known issues and likely some unknown issues as well. This page is intended to help identify which executable to use for which type of project.

FESWMS development is sponsored by the Federal Highway Administration. Aquaveo does not have access to the FESWMS source code and cannot make changes to fix the issues found in the FESWMS numerical model. Aquaveo can fix problems in the SMS interface, used for pre and post processing of FESWMS files.

This list may not contain all know issues, so please feel free to add to it.

Sediment Transport

Aquaveo employees have successfully run test cases using the following options:

- Equilibrium inflow
- Clear water inflow

Attempts to create test cases using the other options have been unsuccessful [5] .

5 Ipson, Mark K. (2006). Analysis of the Sediment Transport Capabilities of FESWMS FST2DH. Thesis, Brigham Young University. [162]

Version 3.3.2

This is the version of fst2dh.exe that is currently distributed with SMS 10.0

- Does not work with Piers.

Version 3.3.2 can be downloaded at ftp://pubftp.aquaveo.com/download/fst2dh332.exe .

Version 3.3.3

- Fixed the issue with Piers but does not work with transient models. It is unknown what is wrong specifically other than models that run using 3.3.2 do not always work with 3.3.3. The problem seems to be that the model doesn't work with transient models.

Version 3.3.3 can be downloaded at ftp://pubftp.aquaveo.com/download/fst2dh333.exe .

Related Topics

- FESWMS

FESWMS Files

Input and Output files for FESWMS are listed below. Note that FESWMS FST2DH will not accept filenames with spaces (e.g. "CAD Mesh.dat").

Input Files

- FST2DH project file (*.fpr)
- Control data (*.dat)
- Mesh data (*.msh – *.net for SMS)
- Flow input data (*.flo)
- Sediment input data (*.sed – *.sdi for SMS)
- Boundary condition data (*.bcs)
- Wind data (*.wnd)
- Wave data (*.wve)
- Time-dependent data (*.tim)

Output Files

- Report data (*.rpt – *.prt for SMS)
- Flow output data (*.flo)
- Sediment output data (*.sed)
- Restart-recovery data (*.rsr)
- Upper coefficient matrix (*.upp)
- Lower coefficient matrix (*.low)
- Scalar output data (*.scl)
- Vector output data (*.vec)
- Profile output data (*.pro)
- Run status data (*.sta)

Related Topics

- FESWMS
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- FESWMS Spindown

FESWMS Graphical Interface

The FESWMS Graphical Interface includes tools to assist with creating, editing and debugging a FESWMS model. The FESWMS interface exists in the Mesh Module .

Model Control

The *FESWMS Model Control* dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. Boundary conditions in FESWMS include flows in/out of the model domain or known water surface elevations. In FESWMS boundary conditions are generally defined on nodestrings but may also be defined on nodes . The default boundary condition is a closed boundary (no flow). See FESWMS BC Nodestrings and FESWMS Point Attributes Dialog for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

Hydraulic Structures

FESWMS was designed for use around highways and includes support for several different types of structures including weirs, culverts, drop inlets, gates and piers. See FESWMS Hydraulic Structures for more information.

Running the Model

The FESWMS Files are written automatically with the SMS project file or can be saved separately using the *File* | **Save FESWMS** or *File* | **Save As** menu commands. See FESWMS Files for more information on the files used for the FESWMS run.

FESWMS can be launched from SMS using the *FESWMS* | **Run FSTD2H** menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *FESWMS* | **Model Check** menu command.

FESWMS Menu

See FESWMS Menu for more information.

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog

- FESWMS Spindown
- Total Flow Nodestring

FESWMS Hydraulic Structures

FESWMS has the capability to model hydraulic structures including: bridges, roadway embankments, culverts, weirs, and drop-inlet spillways. In the finite element network, bridges and roadway embankments are represented with a collection of two-dimensional elements, which overlay the plan view of these structures. However, since culverts, weirs, and drop-inlet spillways are difficult to characterize with elements, these structures are modeled with either one or two node points, with these nodes representing points of inflow and outflow. An exception to this modeling technique occurs when a culvert spans a large channel or is large in comparison to the size of the defined floodplain elements; in this instance, consider modeling the culvert with two-dimensional elements. Units entered into dialogs are determined by the planar and vertical units specified in the SMS display projection.

Culvert

Small culverts are modeled with inlet and outlet node points in FESWMS unless they are located on the model boundary, in which case they are represented with one node point. Parameters for characterizing small culverts include barrel geometry (shape, length, span, slope, and number of barrels), inlet configuration, barrel roughness, and tailwater depth. FESWMS employs the FHWA culvert routing routines as presented in *Hydraulic Design Series 5 (HDS 5), Hydraulic Design of Highway Culverts* [163] [164] and discharge through a culvert is determined by the energy heads at the nodes where the culvert is placed. To define a culvert with two mesh nodes, proceed with the following steps:

1) Select the culvert's inlet and outlet node by choosing the **Select Mesh Node** tool and clicking on the nodes of interest while pressing the *Shift* key.
2) Select the menu option *FESWMS* | **Culvert** . The *FESWMS Culvert* dialog will appear allowing the entry of culvert parameters. Note that when the Flap-gate option is selected, the model only permits flow to travel in the downstream direction. At this point, verify that the nodes identified as "upstream" and "downstream" in the *FESWMS Culvert* dialog represent the culvert inlet and outlet, respectively.
3) Culvert material, shape, and inlet type may be designated by selecting the appropriate drop-down menus and clicking on the desired values. Additionally, the *FESWMS Culvert* dialog offers help in determining the culvert entrance loss coefficient (ke) and the Mannings roughess coefficient of the culvert barrel (nc); simply select the **Help** button next to these two parameters for information. To enter inlet control flow coefficients other than the default values provided (not recommended for novice users), click on the *Override defaults* box net to the appropriate parameters and enter a new value. Further information concerning the selection of culvert parameters is provided in HDS 5.

Weir

FESWMS weirs can be used to model flows over topping bridge decks, embankments, guide banks (spur dikes), and other structures. As with culverts, weirs located in the interior of the finite element network are described by two node points, one on the upstream side and one on the downstream side of the weir. Weir flow is typically modeled by dividing the bridge deck, embankment, or other structure into a series of weir segments. Each segment is described by the appropriate number of nodes (either one or two), a discharge coefficient, submergence criteria, and the length and crest elevation of the segment (all necessary as input to the broad-crested weir-flow equation). To define a weir segment, perform the following steps:

1) Select one or two nodes (depending on if the weir is on the boundary or in the interior of the model) by clicking on the **Select Mesh Node** tool and choosing the node or nodes of interest in defining the weir. If defining a "two-node" weir, hold the *Shift* key to select both the upstream and downstream node.
2) Select the menu option *FESWMS* | **Weir** to open the *FESWMS Weir* dialog.
3) Adjust the default parameters and enter the user-defined values as necessary. The "paved roadway" option is typically selected from the Weir type drop-down menu for bridge decks. As with culverts, when the "Flap-gate" option is selected, the model only permits flow to travel in the downstream direction. Also, the Switch button can be used to change the flow direction if the nodes identified as "upstream" and "downstream" do not correspond to the actual flow direction over the modeled weir.

When defining a series of weir segments to model a bridge deck or other structure, one can create nodestrings on the upstream and downstream sides of the structure. After creating the nodestrings, select each nodestring using the **Select Nodestring** tool in turn while holding the *Shift* key, click on the menu option *FESWMS* | **Weir** , and assign the appropriate crest elevation and other parameters. Further information on weir flow and the broad-crested weir equation can be found in the FESWMS Users Manual [165] and FHWA publication Hydraulic Engineering Circular (HEC) 22, Urban Drainage Design Manual [166] [167] .

Drop-Inlet Spillways

Drop-inlet spillways are sometimes used as flow conveyance structures in reservoirs, and the discharge through this type of spillway is limited by one of three flow conditions (depending on the existing water surface at the inlet and the configuration of the structure):

1) Weir flow
2) Orifice flow
3) Full pipe flow

Flow through a drop-inlet spillway in the interior of a finite element network is modeled by designating an inflow (entrance) node, an out-flow (exit) node, and a set of parameters that describe the structure. If the structure is located on one of the model boundaries, only an entrace node is specified (as flow is assumed to exit the model domain).

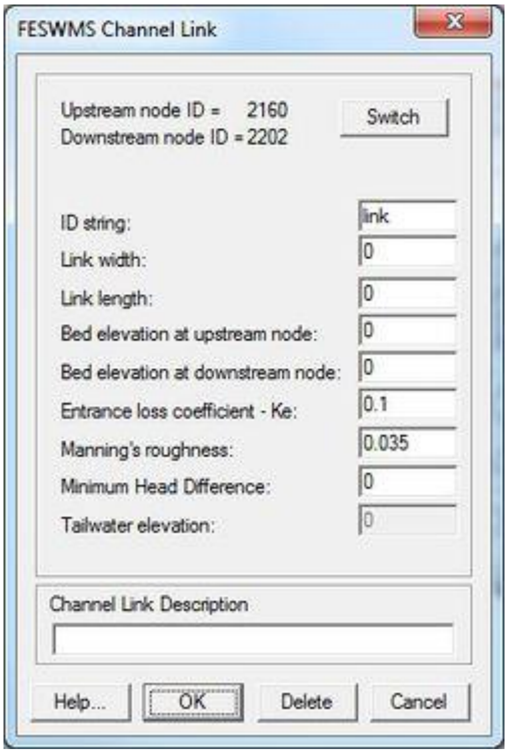
To specify a drop-inlet spillway in the FESWMS model:

1) Select one or two nodes (depending on if the drop-inlet spillway is on the boundary or in the interior of the model) by clicking on the **Select Mesh Node** tool and selecting the nodes of interst.

2) Select the menu option *FESWMS* | **Drop Inlet** . The *Drop Inlet Definition* dialog will open, allowing the entry of the inlet characteristics including geometry and discharge coefficients.

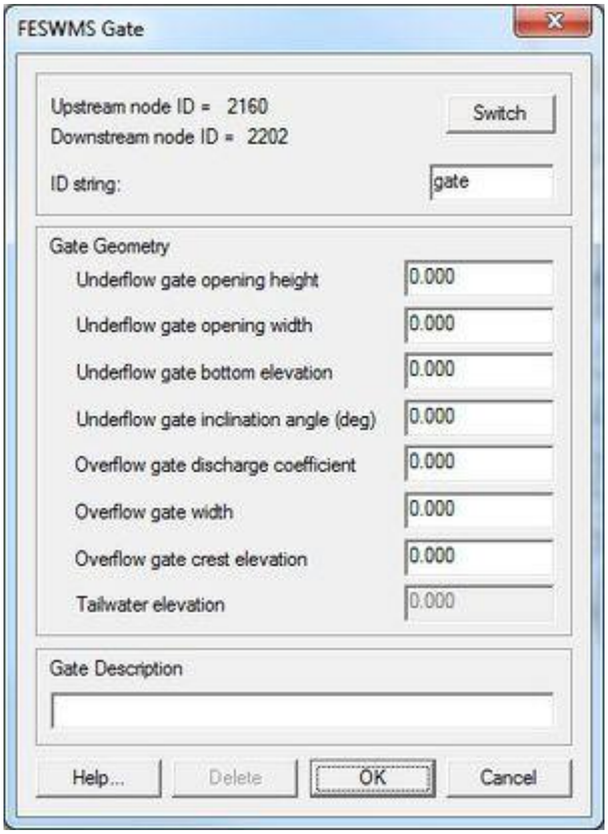
Help with selection of weir and orifice parameters is provided in the FHWA publication Hydraulic Engineering Circular (HEC) 22, Urban Drainage Design Manual [168] [169] .

Channel Link



- Switch
- ID String
- Link width
- Link length
- Bed elevation at upstream node
- Bed elevation at downstream node
- Entrance loss coefficient - Ke
- Manning's roughness
- Minimum Head Difference
- Tailwater elevation

Gate



- Switch
- ID string
- Gate Geometry
  - Underflow gate opening height
  - Underflow gate opening width
  - Underflow gate bottom elevation
  - Underflow gate inclination angle (deg)
  - Overflow gate discharge coefficient
  - Overflow gate crest elevation
  - Overflow gate crest elevation
  - Tailwater elevation

Bridges & Roadway Embankments

Unlike culverts, weirs, and drop-inlet spillways, bridges and roadways will usually be modeled with a collection of two-dimensional elements. Bridge modeling considerations include:

1) Deck roughness
2) Weir flow over decking
3) Pressure flow through the bridge opening
4) Flow around bridge piers
5) Embankment elevations

Once bridge elements are created (in the Map Module during network construction) such that they conform to the two-dimensional plan view of the bridge deck, further treatment of the bridge and associated embankments continues as described below:

Deck Roughness

To define the roughness of the bridge deck elements in the Mesh Module, proceed as follows:

1) Click on the **Select Element** tool and highlight the elements representing the bridge deck while holding the *Shift* key.
2) Select the menu option *FESWMS* | **Material Properties** and click on the *Roughness Parameters* tab of the *FESWMS Material Properties* dialog.
3) To define a material property group for the bridge elements (if not previously defined during network construction), select the **General material properties** button, click on the *New* button in the *Materials Data* window, and enter the property name for the bridge elements. Once finished, click **OK** .

4) in the *FESWMS Material Properties* dialog, highlight the name of the bridge elements in the left-hand window and enter the Manning's n in the *Deck Roughness* edit field. Click **OK** .

A similar procedure can be used to define roughness values for the roadway, including embankments.

Weir Flow Over Decking

Weir flow over decking can be modeled by segmenting the bridge and approaches and establishing a weir for each segment as described previously.

Pressure Flow Through The Bridge Opening

In order for pressure flow to be calculated, the property group representing the bridge deck needs to be coded for potential pressure flow and a bridge deck ceiling elevation (also known as the low chord or low steel elevation) needs to be specified for each element node in the deck as follows:

1) To indicate potential pressure flow, first select *FESWMS* | **Material Properties** and under the *Roughness Parameters* tab highlight the bridge properties group and select the *Potential Pressure* flow box.
2) Next, code in the ceiling elevation for each element node in the deck by using the **Select Mesh Node** tool and highlighting the element nodes corresponding to the bridge deck by holding the *Shift* key and clicking on each node to be assigned the ceiling elevation. An alternative means of selecting all bridge nodes is to choose the **Select Mesh Node** tool, click on the menu option *Edit* | **Select by Material Type** , choose the *Bridge* material type, and click on *Select* . Once the appropriate bridge deck nodes have been highlighted, select *FESWMS* | **Local Parameters** and click on the **Ceiling Options** button. In the *Specify ceiling for selected items* box, enter the ceiling or low chord elevation of the selected nodes and click **OK** .

Bridge and Embankment Elevation Adjustments

To adjust bridge and embankment elevations:

1) Focus in on the area surrounding the embankment or bridge nodes of interest using the **Zoom** tool.
2) Select the corner nodes of the roadway element using the **Select Mesh Node** tool.
3) Adjust the elevations of the nodes to the desired values by overwriting the z-coordinate value in the Z node coordinate display edit field in the SMS window. Note that adjusting the corner nodes automatically adjusts the center and midside nodes.
4) Repeat these steps until all bridge and roadway elements have been set to the desired elevations.

Piers

Piers may be modeled in SMS by either of two methods:

1) Creating elements of the proper geometric sizes and elevations in the network to mimic piers
2) Utilizing the tools provided in SMS including the *FESWMS Pier Definition* dialog

To create a pier in the Mesh Module using the *FESWMS Pier Definition* dialog:

1) Select the **Create Pier** tool and click on the approximate position of the pier in the finite element mesh.
2) Choose the **Select Pier** tool, select the pier of interest, and choose the menu option *FESWMS* | **Pier** . The *FESWMS Pier Definition* dialog will open. Pier parameters used by FESWMS describe the pier location, geometry, and flow resistance. The pier angle is the orientation of the long axis of the pier measure counterclockwise from the x-axis.
3) Adjust the pier parameters, including coordinate location, as necessary by filling in the various edit fields under *Pier Attributes* . Note that pier names can be changed by clicking on the name displayed in the *FESWMS Pier Definition* dialog.

4) Various parameters for pier scour computations can also be specified by selecting the **Global Options** button. For more detailed information on pier parameters and scour calcuations, refer to FHWA publication Hydraulic Engineering Circular (HEC) 18, Evaluating Scour at Bridges [170] [171] [172] [173] .

Modeling Tips

- When dividing a weir into segments on an element basis, it is recommended to assign 2/3 of the length to the midside node of an element, and 1/6 of the length to each of the adjacent vertex nodes.
- More than two rows of elements may be required to accurately model pressure flow through a river crossing.
- A smaller, more highly refined finite element mesh should be used at and around piers, abutments, and other areas of curvilinear/complex flow patterns to improve the accuracy of model results.

External Links

- Hydraulic Design Series 5 (HDS 5), Hydraulic Design of Highway Culverts [174] [175]
- Hydraulic Engineering Circular (HEC) 18, Evaluating Scour at Bridges [176] [177] [178] [179] .
- User's Manual for FESWMS Flo2DH [180]
- Urban Drainage Design Manual Second Edition [181] [182]

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Model Control Dialog
- FESWMS Spindown

FESWMS Material Properties

FESWMS material properties include roughness, turbulence, and wind/wave parameters. The *Materials Properties* dialog is accessed through the *FESWMS* menu.

Roughness Parameters

The roughness helps determine the energy losses as water flows over elements. Each material includes roughness information.

- Manning *n* values '(*n1* , *n2* , *Depth 1* , *Depth 2*)

The primary roughness property is the manning *n* value associated with the element. The manning *n* value can vary with depth by specifying *n* values at two depths. The n1 value is used below depth1. The n2 value is used above depth2. Between depth 1 and depth 2 the *n* value is linearly interpolated.

- *Wall roughness*

The wall roughness is used on the edge of the model domain. Wall roughness is ignored unless the model is using semi-slip boundaries.

- *Soil Liners*

FESWMS can be used with materials representing a soil liner. To use a liner, turn on the linear critical shear stress and set the value. When using a liner n1, n2, depth1, and depth2 are ignored.

- *Pressure flow*

It is required to toggle on "potential pressure flow" with all materials that are assigned to elements with a ceiling elevation. If pressure flow is enabled, the deck roughness is the manning value for the bridge deck. Otherwise deck roughness is ignored.

- *Chezy*

Chezy values are an alternative to using Manning n values for roughness. It's necessary to turn on chezy in the model parameters to use this value.

- *Bed critical shear stress*

The bed critical shear stress is used to compute clear water scour.

Turbulence Parameters

Turbulence parameters are used to control the energy lost in turbulence.

- *Vo*

Base eddy viscosity in Length^2/second

- *Cu1* , *Cu2*

Turbulence model coefficients used to modify base eddy viscosity

- *Eddy diffusivity*

Used to modify base eddy viscosity and is related to the amount of curves in the channel

- *Storativity depth*

The global storativity depth (specified in the model parameters) can be overriden on a material level by entering a non-zero value.

Wind/Wave Parameters

- *Wind Shear Reduction Factor*
- *Water Wave Height*
- *Wave Period*

Related Topics

- FESWMS Menu

FESWMS Menu

The following menu commands are available in the *FESWMS* Menu:

Assign BC (Boundary Condition)

Opens either the *FESWMS Nodestring Boundary Conditions* dialog or the *FESWMS Nodal Boundary Conditions* dialog. A node or nodestring must be selected for this command to be available.

Local Parameters

Brings up the *FESWMS Local Parameters* dialog. Requires that a node be selected.

Initial Conditions

Brings up the *Initial Conditions Data* dialog.

Weir

Opens the *FESWMS Weir* dialog. Requires a selected nodestring be active.

Culvert

Brings up the *FESWMS Culvert* dialog. Requires that the inlet and outlet nodes are currently selected.

Drop Inlet

Brings up the *Drop Inlet Definition* dialog. Requires that a node is currently selected.

Channel Link

Brings ups the *FESWMS Channel Link* dialog.

Gate

Opens the *FESWMS Gate* dialog.

Pier

Brings ups the *FESWMS Pier Definition* dialog.

FLUX String

Option to set currently selected nodestring to Total Flow string.

Material Properties

Opens the *FESWMS Material Properties* dialog. This dialog is different than the general *Material Properties* dialog in SMS.

Model Check

Runs the model check. If there are errors, then the *Model Check* dialog will appear.

Model Control

Brings up the *FESWMS Model Control* dialog.

Run FST2DH

Starts the FST2DH model wrapper.

Local Parameters Dialog

Available when a node is selected. Allows setting **Ceiling Options** or **Sediment Options** for the selected node.

Initial Conditions Dialog

This dialog allowed writing out a FESWMS initial conditions (*.ini) file.

File Name – This section has an option for saving out an initial conditions (*.ini) file. The initial time step can also be set.

Velocity

- *Constant* – Allows setting a constant initial water surface elevation.
- *Dataset* – Using the **Select** button brings up Select Dataset dialog where velocity dataset can be designated.

Water Surface Elevation

- *Constant* – Allows setting a constant initial water surface elevation.
- *Dataset* – Using the **Select** button brings up Select Dataset dialog where velocity dataset can be designated.

Run FST2DH

This command will first run the *Model Checker* which will search for errors in the model set up. If found, the *Model Checker* dialog will display the errors found and give recommendation on how to fix the errors.

If there are no errors, or SMS is told to ignore the errors, the *FESWMS* model wrapper will start. The model wrapper will display the progress of the model run. Clicking the **Abort** button will end the model run. Once the model run has finished, the **Abort** button will change to an **Exit** button. Selecting the *Load solution* option will open the solution file in SMS when the **Exit** button is clicked.

Related Topics

- FESWMS

FESWMS Model Control Dialog

The *FESWMS Model Control* dialog is used to set general simulation options. This document highlights the more commonly used options. Refer to the FESWMS manual [183] for a more detailed description of how these parameters affect the model results.

General Tab

The controls in the *General* tab include model description, FESWMS version, run controls, input options, and output options.

- Network stamp and BC descriptor

  These are comments that are added at the top of the FESWMS input files. The network file (*.net) includes only the network stamp but the bc file includes both the network stamp and the bc descriptor.

- Slip conditions

  This controls how boundaries are handled by FESWMS. Slip boundaries allows water to move freely along the boundary. No slip fixes the velocity at fixed boundaries to 0.0. Semi-slip allows water to move along boundaries but reduces the velocities.

- Run Type

  A simulation can compute hydrodynamics, sediment, or both hydrodynamics and sediment (semi-coupled). It is not recommended to use the sediment only option. Use the semi-coupled option instead.

- Solution Type

  This controls whether the simulation will be treated as steady state or dynamic (time dependent). A steady state run requires boundary conditions that do not change and computes a solution that does not change through time.

- Input Options

This controls the files that will be used in FESWMS.

An INI (initial conditions) file is used to provide an initial solution for FESWMS. The initial conditions file format is identical to the FESWMS solution file format and a FESWMS solution may be used for an initial conditions file. Initial conditions files can also be created from inside of SMS. INI files are used as hot starts in the spin down process.

- Output Options

The scalar dataset file and vector dataset files are additional output files written in the EMRL dataset format. Regardless of whether this file is used or not, the datasets water surface elevation, depth, and velocities are written to the *.flo solution file. The dataset files will have the same base name as the other project files and have the extensions *.scl (scalar) and *.vec (vector). The data written to the dataset file can be specified by clicking on the options button. Available scalar datasets include: Depth-averaged velocity magnitude, Unit flow rate magnitude, Froude number, Mechanical energy head elevation, Bed shear stress magnitude, Vorticity magnitude, Wind Speed, Depth of general scour, Scoured bed elevation, Sediment concentration, Sediment volumetric transport rate, Sediment volumetric transport capacity, and Sediment transport rate deficit. Available vector datasets include: Depth-averaged flow velocities, Unit flow rate, Bed shear stress, Tropical cyclone wind velocities, Sediment volumetric transport rates, and Sediment volumetric transport capacities.

Timing Tab

- Relaxation factor

Affects how fast and reliable FESWMS will find a solution. Reducing this value may help FESWMS achieve a solution but may increase runtime.

- Iterations

This is the maximum number of iterations to perform. There will be fewer iterations if the model reaches the convergence parameters before reaching this number of iterations (see parameters tab for convergence parameters).

- Starting time

The time to start a dynamic simulation in hours.

- Run time

The length of the simulation in hours.

- Time step size

The length of time for each time step in hours. FESWMS is an implicit model so the time step size is not dependent upon the courant number.

Parameters Tab

- General Parameters
  - Water surface elevation

    This is the initial water surface elevation for a cold start simulation. This number should always be larger than the elevation of the highest node in the mesh.

  - Unit flow convergence

    The maximum change of unit flow between iterations that will be considered a acceptable for a converged solution.

  - Water depth convergence

    The maximum change of water depth between iterations that will be considered acceptable for a converged solution.

  - Storativity depth

Storativity allows an element to remain active as long as the water surface elevation is above the highest node value minus the storativity depth. Specifying a non-zero storativity value generally increases the stability of the model.

- Element wetting/drying

  Unless this is checked, FESWMS will not allow elements to wet and dry during a simulation.

- Depth tolerance for drying

  Indicates the required depth to re-wet a dry node. If this value is zero, any depth above the node will re-wet a dry element. Using a value greater than zero helps prevent wet/dry excessive wet/dry oscillations that can lead to instabilities.

- Reference Point Location
  - Latitude
  - Longitude
  - Angle
  - X-Coordinate
  - Y-Coordinate
- Default Weir Paremeters
  - Default tailwater
  - Default minimum head
- Default Culvert Parameters
  - Tailwater elevation
  - Minimum head difference
- Default Channel Link Parameters
  - Tailwater elevation
  - Minimum head difference
- Default Gate Parameters
  - Tailwater elevation
  - Minimum head difference
- FST2DH Task
  - Check Network
  - Resequence

Print Tab

This tab controls the information that is written to the printed output file (*.prt).

- Extras
  - Wide column format
  - ECHO to screen
  - Sound beeps after run
    - Number beeps
- Output Print
  - ECHO input
  - Elements and nodes
  - Initial conditions

    - Element assembly sequence
    - DOF array
    - Nodal scalar data
    - Nodal vector data
    - Pier scour data
    - Clear water scour
    - Include nodal unit flow rates
- Iteration Print Code
    - Print diagnostics every:
- Reports
    - Weirs
    - Drop Inlets
    - Piers
    - Links
    - Culverts

Sediment Transport

This tab defines the parameters used for sediment transport including bed configuration. See FESWMS Sediment Control for more information.

Wind/Storm Conditions Tab

- Condition

  Wind conditions include none, a directional wind, or a storm (cyclone or hurricane).

- Wind Parameters

  These parameters are used to define the parameters for a directional wind (the same direction magnitude for the entire model domain).

- Storm Parameters

  These parameters are used to define a hurricane or cyclone moving across the model domain. The storm parameters such as direction and central pressure are constant throughout the simulation. The effect of a storm surge combined with the wind fields can be defined using the surge parameters toggle. See the FESWMS manual for a description of the input parameters.

- Wave Parameters

  FESWMS has the ability to consider wave induced stresses. The wave height ratio is the breaking water wave height to depth ratio. Waves will break when the significant wave height equals the water depth times this ratio.

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Hydraulic Structures
- FESWMS Spindown

FESWMS Point Attributes Dialog

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

Attribute Type Frame

- *None*
- *Boundary Conditions* – Options button opens the *FESWMS Nodal Boundary Conditions* dialog.
    - *FESWMS BC Nodes* – FESWMS allows specifying boundary conditions on a nodal basis although it is generally preferable to use boundary conditions on nodestrings . Please refer to the FESWMS manual for information on the nodal boundary condition options.

Options Frame

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

Related Topics

- Feature Objects Menu
- Mesh Generation

FESWMS Sediment Control

The FESWMS engine was modified in 2005 to include a capacity for sediment transport. In fact at this time the official name of the system changed from Flo2DH to FST2DH. The "ST" stands for "Sediment Transport". That functionality is supported in the SMS interface for the model. Shortly after the capability was added to the numerical engine, a study was performed at Brigham Young University to verify the capabilities and provide sample applications. It was discovered that the FESWMS engine does indeed provide reasonable estimates of sediment transport, including scour and deposition patterns for some applications of non-cohesive sediments. However, the study also revealed that several of the advertised capabilities do not function, and the model is prone to numerical instablities. We advise caution and patience for modelers attempting to use this capability of the FESWMS engine.

Before simulating sediment transport, set up a non-transport project. Then add the sediment tranpsport parameters and attributes. This is done in three main steps:

1) Set the model parameters
2) Define the bed conditions and sediment grain sizes
3) Define the sediment boundary conditions

Model Parameters

A sediment transport simulation requires several settings in the *General* tab of the *FESWMS Model Control* be set. These include the *Run Type* , the *FST2DH Output* and the *Solution Type* . Set the *Run Type* to *Sediment* or *Semi-coupled* . The " *Sediment* option uses an existing hydrodynamic solution. The *Semi-coupled* option instructs the engine to switch back and forth between hydrodynamic calculations and sediment calculations at each time step. Tests have shown that for stability, the *Semi-coupled* option should be used. The "Solution Type" should be set to "Dynamic". The *Sediment Transport Solutions* toggle should also be turned on.

The *Sediment Transport* tab of the *FESWMS Model Control* provides access to the other parameters related to sediment transport calculations in FESWMS. The *Report Options* section allows specification of what information will be saved to the report (*.prt) file by FESWMS. The *Control Options* section of the sediment control allows for specification of the numeric properties such as convergence criteria an maximum iterations the engine should use while solving the sediment transport quantities. This section also includes two buttons to acces the sediment parameters and the bed definition options.

The parameters include a choice of eight transport formulae. Each formula requires associated variables to be specified.

Bed Conditions

FST2DH performs sediment calculations using a three layer system. The first layer is the "Active bed-layer". This is the layer where deposition and scour can occur. The scour at a single time step is limited to the thickness of the active layer. The second layer consists of material deposited on top of the native bed. If core samples have been taken and show a variation in particle distributions, a depositional-layer may be defined. The third layer defines the native bed. Total scour is limited to the depth or thickness of this layer.

The **Bed Control** button in the sediment control options allows accessing the interface to define the global bed. This include the thickness of the three computation layers and the distribution of particle sizes in each of those layers. FST2DH supports up to 8 particle sizes. Define the characteristic particle size for each gradation and the percent that gradation makes of each of the three layers. If a single size gradation is specified, the three percentages would all be 100%.

The global bed definitions apply to the entire domain unless a local specification is made. This is done by selecting the nodes in the area the local definition changes and define the "Local Paramters" in the FESWMS menu.

Limitation

Tests have shown that numerical sensitivity increases as the number of particle sizes increases. Use as few as possible to represent the situation. Sensitivity also increases with long duration runs (over 1 week).

Sediment Boundary Conditions

Sediment boundary conditions must be applied at all open boundaries just as hydraulic conditions must be defined at those locations. In the case of sediment, specify the amount of sediment entering the domain over that open boundary using one of the following options:

- Don't specify inflow sediment. This means clear water is entering the domain.
- Specify volume of material per time as an essential or natural boundary condition for each particle size.
- Specify concentration (ppm) as an essential or natural condition for each particle size.
- Allow FST2DH to compute the inflow sediments assuming equilibrium conditions.

Limitation

Tests have shown that only the clear water and equilibrium condition boundary conditions produce reasonable results.

Related Topics

- FESWMS

FESWMS Spindown

For cold start simulations, the initial velocities are zero and the water surface elevation is constant. This is often referred to as the "bathtub condition." Often FESWMS will not directly converge using these initial conditions.

Incremental Loading Strategy

The flow equations are nonlinear and thus require an iterative solution, starting from some initial guessed value. Convergence of the iterative solution is not guaranteed. Since the desired boundary conditions may be vastly different from a cold start condition, it may be impossible to get convergence starting from this bathtub type condition. However, a solution can be obtained using a series of “Runs” that generate solutions progressively closer to the desired answer. Intermediate boundary conditions that are closer to the final desired boundary conditions are specified to generate a set of flow conditions. These conditions do not represent the final desired flow conditions, but are closer to the final desired flow conditions than the original cold start, and can be used as initial conditions for a subsequent run. Starting the model from a previous solution is called “Hot Starting”. In the incremental loading strategy, “loads” consisting of applied flow rates and water surface elevations along the boundary increment from a cold start condition to the final condition. By choosing a suitably small increment in the boundary conditions, convergence can be attained.

FESWMS Spindown refers to the process of using the Steering Module to automate the process of incremental loading. The Steering Module can vary the following:

- Boundary Conditions
  - Water surface elevation
  - Flow rate
- Model Parameters
  - Eddy viscosity
- Finite Element Network
  - Geometry (nodal elevations)

This replaces the need to perform incremental loading by hand in FESWMS .

FESWMS Spindown Dialog

The *FESWMS Spindown* dialog updates with the progress of spinning down the model.

The top window explains the spindown convergence of the current run. Each iteration shows as a green point, allowing determination of if the run is converging or diverging (moving toward or away from 0 head change). The iteration being performed is shown just above this plot.

The bottom window shows the overall spindown of the model. The green points represent successful runs, and the red Xs represent failed runs. When this plot reaches 100% spun down, the model is finished. This percent is shown just above this plot.

This process can take several minutes to complete. When the spindown has finished, a window appears advising that the "Steering process has terminated – See status file for details." This status file is named "SteeringStatus.txt" and gives a summary of the steering process. A final solution file will also be created.

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- Steering

6.3 HEC-RAS – Hydrologic Engineering Center's River Analysis System

HEC-RAS

The Hydrologic Engineering Center's (CEIWR-HEC) River Analysis System (HEC-RAS) is a one- and two-dimensional model for computing water surface profiles for steady state or gradually varied flow. HEC-RAS supports networks of channels and is capable of modeling subcritical, supercritical, and mixed flow regime profiles. HEC-RAS is able to model obstructions in the flow path.

The SMS interface to HEC-RAS includes tools for setting up the river networks and cross-sections as well as post-processing capabilities. SMS reads and writes the HEC-GeoRAS GIS import format.

HEC-RAS Simulation

Starting in SMS 12.2, HEC-RAS uses a simulation process. SMS can run multiple HEC-RAS simulations using the same components. To create a new simulation, right-click in the Project Explorer and select *New Simulation* | **HEC-RAS** . Components can then be linked to simulations by dragging the components under the simulation or by right-clicking on the component item and using the *Link To* submenu.

Components for the HEC-RAS simulation include the following:

- 2D Mesh

HEC-RAS Simulation Menu

Right-clicking on a HEC-RAS simulation provides a menu with the standard simulation menu commands and the following commands specific to HEC-RAS:

- **Model Check** – Launches the model checker to look for problems with the simulation setup.
- **Export HEC-RAS** – Exports the *.rasgeo file containing the mesh information that will be converted during the simulation run.
- **Launch HEC-RAS** – Brings up the HEC-RAS model wrapper and starts the simulation run. At the end of the simulation run, HEC-RAS will have created a number of output files for use in the HEC-RAS program.
- **Save, Export, and Launch HEC-RAS –** Completes the processes of saving the SMS project, exporting the HEC-RAS files, and launching the HEC-RAS simulation run.

HEC-RAS Files

A HEC-RAS simulation in SMS can generate a number of files for use in the HEC-RAS program. The following files are generated:

- *.rasgeo – Contains the mesh data generated in SMS. This file is used to generate the other files and must be created before running the HEC-RAC simulation.
- *.g01 – HEC-RAS geometry file.
- *.g01.hdf
- *.prj – The HEC-RAS projection file. For post-processsing, open this file in the HEC-RAS software.

HEC-RAS RASGEO File Example

The RASGEO file contains unit, element, and node information from the mesh component in the HEC-RAS simulation.

```
RASGEOM_FROM_SMS  10
GridUnit  "METER"
Elem  1  2  4  1
Elem  2  2  5  4
Elem  3  4  6  3
Elem  4  4  7  6
Elem  5  7  4  5
Elem  6  6  7  19
Elem  7  6  19  9
Elem  8  7  8  19
Elem  9  20  8  10
Elem  10  8  20  19
Elem  11  22  9  21
...
Node  1  412266.749188  2048559.109411  0.000000
Node  2  412293.104804  2048577.696988  0.000000
```

```
Node  3  412206.837470  2048630.299699  0.000000
Node  4  412235.340370  2048594.711214  0.000000
Node  5  412292.972567  2048627.616721  0.000000
Node  6  412234.872220  2048666.466041  0.000000
Node  7  412264.180764  2048663.798224  0.000000
Node  8  412281.643700  2048705.121722  0.000000
Node  9  412214.903435  2048704.060367  0.000000...
```

HEC-RAS Troubleshooting

Starting in SMS 12.2, HEC-RAS uses the dynamic model interface. If HEC-RAS was purchased through Aquaveo, the DMI file and HEC-RAS library was included in the installation and should be available for use when starting SMS.

If installing HEC-RAS to an existing version of SMS 12.2 or later, the XML file must be placed in the *DynamicXml* directory inside the SMS program directory. HEC-RAS also requires a translator. The RAS2D_translator folder should be placed in the *Models* directory inside the SMS program directory.

SMS needs to know the location of the HEC-RAS executable. If an error appears stating the executable cannot be found, go to *Edit* | **Preferences** to open the *Preferences* dialog. In the *File Locations* tab, set the *HECRAS Translate* item to point to the "_runRunGeneric.bat" file in the RAS2D_tranlator.

Related Topics

- SMS Models
- HEC-RAS in WMS

External Links

- HEC-RAS website

6.4. HYDRO AS-2D

HYDRO AS-2D

HYDRO AS-2D	
Model Info	
Model type	2D current, pollutant, and sediment transport simulation.
Developer	Hydrotec, Dr.- Ing. Marinko Nujic
Web site	HYDRO AS-2D web site
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

HYDRO AS-2D performs 2D modeling of bodies of water. The procedure integrated in HYDRO AS-2D is based on the numerical solution of the 2D current equations with finite-volume-discretization. In addition to 2D current simulation, HYDRO AS-2D can also simulate pollutant and sediment transport.

SMS supports HYDRO_AS-2D through the Generic Model interface offering a simple way to set model parameters, run the model, and visualize the results.

The HYDRO AS-2D model can be added to a paid edition of SMS.

Functionality

Features

HYDRO AS-2D is characterized by:

- Solves hundreds of thousands of elements very quickly
- Calculation accuracy guaranteed by extensive lab tests and real-world applications
- High stability, robustness and exactness for a wide spectrum of hydrodynamic conditions
- Volumetric accuracy of the tidal wave propagation on complicated topography
- Solutions showing flow speed, direction, and flood depth
- The bed changes as a result of sediment erosion and sediment deposition are modeled
- Fully coupled transport and hydrodynamic solver ensures current changes due to sediment scour/deposition are accurately simulated
- Simulate transport of up to five grain dimensions at the same time
- Short calculating time for millions of elements

Applications

- River flooding analysis
- Sediment erosion/depositions studies in river channels and floodplains
- Tidal wave propagation
- Pollutant dispersion in a waterway

Extensions

HYDRO AS-2D is highly user-optimized and offers the following extensions:

- LASER_AS-2D: Net-generator based on laser-scan-data (intelligent thinning out and deriving Break lines) and Stream Network Generator
- JabPlot: Creating professional profile and longitudinal plots
- CHECK2DM: Testing the net based on geometrical and numerical aspects
- JabMap: Pre- and postprocessor /interface to GIS

Using the Model / Practical Notes

It is recommended that HYDRO AS-2D is used with an adequate graphics and co-processor. A list of graphics and co-processors can be found on Hydrotec's website .

Graphical Interface

HYDRO AS-2D uses the Generic Model Graphical Interface .

External Links

- Hydrotec HYDRO AS-2D website (in German)
- Hydrotec HYDRO AS-2D software website (in German)
- HYDRO AS-2D model developer website (in German)

Related Topics

- Generic Model Interface

6.5. SRH-2D: Sedimentation and River Hydraulics – Two-Dimensional

SRH-2D

SRH-2D	
Model Info	
Model type	Two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems
Developer	Yong Lai Bureau of Reclamation
Web site	Bureau of Reclamation
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • SRH-2D

SRH-2D , or the *Sedimentation and River Hydraulics – Two-Dimensional* model, is a two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems under development at the Bureau of Reclamation. It was known as *SRH-W* in version 1, and the name was changed to the current SRH-2D from version 2 onward.

Currently, it is recommended that the SRH-2D model only be used with the 64-bit version of SMS. Errors may occur if using the 32-bit version.

The SRH-2D model can be added to a paid edition of SMS.

Features

SRH-2D is a 2D model and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, with perched rivers, with multiple channel systems, and with complex floodplains. A 2D model may also be needed if one is interested in local flow velocities, eddy patterns and flow recirculation, lateral variations, flow spills over banks and levees, and flow diversion and bifurcation.

- SRH-2D solves the 2D depth-averaged form of the diffusive wave or the dynamic wave equations. The dynamic wave equations are the standard St. Venant depth-averaged shallow water equations
- Both the diffusive wave and dynamic wave solvers use the implicit scheme to achieve solution robustness and efficiency
- Both steady or unsteady flows may be simulated
- All flow regimes, i.e., subcritical, transcritical, and supercritical flows, may be simulated simultaneously without the need of a special treatment

- Solution domain may include a combination of main channels, side channels, floodplains, and overland
- Solved variables include water surface elevation, water depth, and depth averaged velocity. Output information includes above variables, plus flow inundation, Froude number, and bed shear stress

The Bureau of Reclamation does not provide technical support for SRH-2D.

Graphical Interface

SRH-2D uses a custom interface to specify boundary conditions, model paramters, model control and material parameters. The interface includes the following:

- SRH-2D Simulation
- SRH-2D Menu
- SRH-2D Model Control
- SRH-2D Material Properties – SRH-2D is able to work with a number of material zones. Materials may be created from a Materials Coverage, an SRH-2D coverage, or directly from the mesh. Materials are added by selecting *Edit* | **Materials Data** when the coverage or mesh is activated.
- SRH-2D Coverages
- SRH-2D input and output files – When wanting to execute an SRH-2D model, export the model native files using the **Export SRH-2D Files** or **Save, Export, and Launch SRH-2D** commands in the *SRH-2D* menu. The native files include:
  - SRHHYDRO – Contains key information about the simulation while acting as a directory to other files for SRH-2D to use.
  - SRHGEOM – Tells SRH-2D where each element is located and the characteristics of that element.
  - SRHMAT – Gives each element a material type.
  - SRHMONITORPTS – Tells SRH-2D that there are monitor points to watch and where those points are located.

In the past, this model has been utilized through the Generic Model Graphical Interface . The SRH-2D version 2.0 Distribution included SRH-2D template files for both SMS 8.0 and SMS 10.0. These are no longer needed with the custom interface.

Steps to Create an SRH-2D Model

To create an SRH-2D model, the following general steps should be followed:

1) Gather data pertinent to the project and location. This should include bathymetry data, roughness data (Manning's n value), coordinate system corresponding to the data, and flow data.
2) Specify a coordinate system. This is done in the *Disply Projections* dialog accessed through the *Display* menu.
3) Add bathymetry data. This may come as survey data, Lidar data, or Raster DEM data to name a few.
4) Check the triangulation or raw data display. It is important to make sure that SMS is reading the data the same way that it was measured. Turning on contours will allow the user to view what SMS sees and make adjustments as needed. Contours may be turned on using the **Display Options** command. Optionally, the user may use the tools available in SMS for refinement of the data.
5) Create coverages . A simple SRH-2D project would likely include a SRH-2D main coverage which would hold data about mesh type and bythymetry data, a materials coverage, which would map material types to region, and a monitor points coverage which will hold data relating to specific locations where site specific data will be gathered. Coverages may be created by right-clicking on the map data folder from the data tree and selecting **New Coverage** . The coverage is assigned a type upon creation. For an SRH-2D model, select the SRH-2D coverages as they relate to the data that will correspond to that type.

6) Outline the workspace with arcs. Here the user is defining regions of the model location that will have unique features. For example, locations of more water interaction will need more detail which equates to more nodes; locations with different roughness values will need to be separated for material type assignments. Create polygons for areas of similar characteristics. Keep in mind that SMS has a variety of tools available to adjust the arcs to meet the modeling needs of the project.
7) Build polygons.
8) Assign attributes to the polygons. Direct SMS to what materials and what mesh type is to be built over that polygon. Specify how SMS should assign elevation data by selecting the bathymetry source
9) Assign attributes to the arcs by giving the arcs boundary conditions . For interior arcs, the only option is a monitor line. For exterior arcs the user may choose from a variety of inlet conditions, whether subcritical or supercritical, as well as outlet or water surface elevation options. For no flow boundaries, the option of a wall or symmetry is available.
10) Prepare to build the mesh. Review the information given to SMS to ensure that the field data matches what is represented virtually for the region. After a review of the inputs for the model, the mesh is ready to be built. The mesh is built by converting the Map coverage to a 2D Mesh.
11) Build the mesh. If all of the attributes for the arcs and polygons were assigned correctly, the mesh will represent the same data over the elements and nodestrings. If the data is incomplete or changes need to be made, they will need to be done on the elements (for material properties) or the nodestrings (for boundary conditions). Note, that if necessary, it is possible to delete the mesh and adjust values over the polygons and arcs before regenerating the mesh.
12) Run the model. First, be sure that the project is saved. Next, from the *SRH-2D* menu, select **Export SRH-2D** . Finally, from the *SRH-2D* menu, select **Launch SRH-2D** .
13) Analysis of results and post processing.

Troubleshooting

Consult the SRH-2D Manual for troubleshooting difficulty in executing the model.

Some common errors in running SRH-2D include:

- Running SRH-2D requires both the pre-SRH-2D executable and the SRH-2D executable. If SMS cannot find these executables, the model run will encounter an error. The *File Locations* tab in the *Preferences* dialog can be used to specify the path to these executables if SMS cannot find them.

Releases

Version 1

SRH-W, or Sedimentation and River Hydraulics - Watershed, is a two-dimensional (2D) hydraulic model for river systems and watersheds developed at the Bureau of Reclamation. SRH-W was originally developed for Reclamation internal use for various projects, and version 1.1 was released for public use.

SRH-W v1.1 is used for hydraulic flow simulation in rivers and runoff from watersheds, but without the sediment capability. It solves the 2D dynamic wave equations (the standard depth-averaged St. Venant equations) that are mainly used for river simulation. In addition, the diffusive wave solver is used for watershed runoff simulation and river simulation.

Version 1.1 is comparable to many existing models such as RMA-2 (US Army Corps of Engineers, 1996) and MIKE 21 (DHI software, 1996) in its river simulation capability. For watershed applications, SRH-W v1.1 is a distributed model for event based runoff simulation and has capabilities similar to CASC2D (Julien, et al, 1995).

Version 2

In Version 2, SRH-W was renamed to SRH-2D. This is the version that is associated with the users document that is distributed by the United States Bureau of Reclamation.

Version 2 solves the 2D dynamic wave equations, i.e., the depth-averaged St. Venant equations. Its modeling capability is comparable to some existing 2D models but SRH-2D claims a few boasting features. First, SRH-2D uses a flexible mesh that may contain arbitrarily shaped cells. In practice, the hybrid mesh of quadrilateral and triangular cells is recommended though purely quadrilateral or triangular elements may be used. A hybrid mesh may achieve the best compromise between solution accuracy and computing demand. Second, SRH-2D adopts very robust and stable numerical schemes with a seamless wetting-drying algorithm. The resultant outcome is that few tuning parameters are needed to obtain the final solution. SRH-2D was evolved from SRH-W which had the additional capability of watershed runoff modeling. Many features are improved from SRH-W.

Major Features of Version 2

Major SRH-2D capabilities are listed below

- 2D depth-averaged dynamic wave equations (the standard St. Venant equations) are solved with the finite-volume numerical method
- Steady state (with constant discharge) or unsteady flows (with flow hydrograph) may be simulated
- An implicit scheme is used for time integration to achieve solution robustness and efficiency
- An unstructured arbitrarily-shaped mesh is used which includes the structured quadrilateral mesh, the purely triangular mesh, or a combination of the two. Cartesian or raster mesh may also be used. In most applications, a combination of quadrilateral and triangular meshes is the best in terms of efficiency and accuracy
- All flow regimes, i.e., subcritical, transcritical, and supercritical flows, may be simulated simultaneously without the need for special treatments
- Robust and seamless wetting-drying algorithm; an
- Solved variables include water surface elevation, water depth, and depth averaged velocity. Output variables include the above, plus Froude number, bed shear stress, critical sediment diameter, and sediment transport capacity.

SRH-2D is a 2D model, and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, with perched rivers, with side channel and agricultural returns, and with braided channel systems. A 2D model may also be needed if one is interested in local flow velocities, eddy patterns, flow recirculation, lateral velocity variation, and flow over banks and levees.

Version 3

SRH-2D version 3 is essentially Version 2 with Sediment Transport capability added. This version is currently distributed with the SMS package.

SRH-2D Files

See the article SRH-2D Files

External Links – SRH-2D Version 2.0

- SRH-2D version 2.0 Theory and User Manual
- SRH-2D version 2.0 Distribution – Includes software, manual, template file for integration into SMS interface, and tutorials
- SRH-2D version 2.2 Distribution – Includes software, manual, Ttmplate file for integration into SMS interface, and tutorials

Papers / Presentations

- SRH-2D Theory Paper
- SRH-2D Training Presentation
- 2005 US-China Workshop Paper [184]

- 2006 FISC Paper on Savage Rapids Dam Removal Project - "Comparison of Numerical Hydraulic Models Applied To The Removal of Savage Rapids Dam Near Grants Pass, Oregon"
- FISC 2006 Short Course Presentation
- Using Bathymetric LiDAR and a 2-D Hydraulic Model to Identify Aquatic River Habitat
- List of journal articles using SMS by Prof. Greg Pasternack, UC Davis

Project Reports

- Bountry J.A. and Lai, Y.G. (2006)."Numerical modeling of flow hydraulics in support of the Savage Rapids Dam removal."
- Lai, Y.G., Holburn, E.R., and Bauer, T.R. (2006)."Analysis of sediment transport following removal of the Sandy River Delta Dam."
- Lai, Y.G. and Bountry, J.A. (2006). "Numerical hydraulic modeling and assessment in support of Elwha Surface Diversion Project."
- Lai, Y.G. and Bountry, J.A. (2007). "Numerical modeling study of levee setback alternatives for lower Dungeness River, Washington"
- Lai, Y.G. and Greimann, B.P. (2011). "SRH Model Applications and Progress Report on Bank Erosion and Turbidity Current Models"

In the News

- article "Computer Modeling Smoothes a Dam Hard Job"
- Photos related to Wired.com article "Computer Modeling Smoothes a Dam Hard Job"

External Links – SRH-W

SRH-W Version 1.1

- SRH-W version 1.1 User Manual
- SRH-W version 1.1 Distribution Package – Includes Software, Manual, and Tutorials

Papers / Presentations

- 2006 FISC Watershed Modeling Paper - Watershed Simulation with an Enhanced Distributed Model

Related Topics

- SMS Models
- SRH-2D Project Workflow

SRH-2D Coverages

The SRH-2D model makes use of the simulation based modeling approach. This requires defining coverages in the Map module to build the components for use in the SRH-2D Simulation .

Boundary Conditions Coverage

SRH-2D support a variety of boundary conditions for hydraulic computation. The boundary conditions for the model are specified through the SRH-2D Boundary Condition coverage by selecting an arc. In SMS 11.2 and earlier versions a boundary could also be defined by selecting a nodestring in the mesh instead, though this is no longer supported in SMS 12.0 and later versions. Once the arc or nodestring has been selected, double-clicking, or right-clicking and using the **Assign Linear BC...** command, will bring up a dialog where the boundaries can be assigned.

All currently supported boundary types are exterior boundaries that must be placed on the mesh boundary with the exception of a monitor line. Other boundary types include the following: Inlet-Q, Inlet-SC, Exit-EX, Exit-H, Exit-Q, Wall, and Symmetry.

- Inlet-Q is a subcritical inlet boundary that may be given as a constant discharge or as a variable discharge hydrograph. The velocity distribution type may be selected from the boundary condition menu. Inlet flow volume values can only be positive.
- Inlet-SC is a supercritical inlet boundary that may be given as a constant discharge or as a variable discharge hydrograph. Inlet-SC also requires information about depth. Within the boundary condition menu, the velocity distribution type may be specified. Inlet flow volume values can only be positive.
- Exit-EX is a supercritical exit boundary condition.
- Exit-H is a stage type exit boundary where water surface elevation may be given as a constant number or as a stage-discharge or rating curve. Using a constant or rating curve allows accessing the *Populate* dialog.
- Exit-Q is an exit boundary with a discharge given as a constant number or as a hydrograph.
- Wall
- Symmetry is defined as a boundary where all dependent variables are extrapolated assuming the gradient of the variable in a direction normal to the boundary is zero except the velocity component normal to the boundary.
- Monitor-Line it is an internal polyline which may be used to monitor the total flow discharge through it.
- Weir – See the SRH-2D Structures article for more information.
- Culvert – See the SRH-2D Structures article for more information.
- Pressure zone – See the SRH-2D Structures article for more information.
- Gate – See the SRH-2D Structures article for more information.

See SRH-2D Boundary Conditions for more information on the options for each boundary type.

Boundary Conditions Coverage Right-Click Menu

The boundary conditions coverage has the standard coverage item right-click menu commands as well as specific right-click commands. The following commands are also part of the SRH-2D boundary conditions coverage:

- **Options...** – Opens the *Coverage Options* dialog. This dialog allows specifying an HY-8 file for use in creating culvert structures.

Obstructions Coverage

Used to create feature objects that represent obstructions, such as bank protrusions and boulder clusters. Obstructions can be assigned to feature arcs or points.

When this coverage is active, double-clicking on a feature arc or point, or right-clicking on an arc or point followed by selecting the **Assign Obstructions...** command, will bring up the *Obstructions* dialog.

The *Obstructions* dialog has the following options:

- *Obstruction Width/Diameter* – Each arc segment represents a rectangular area based on the assigned width. Each point represents a circular diameter based on the assigned width.
- *Obstruction Thickness (Z-dir)* – Represents the vertical thickness of the obstruction.
- *Drag Coefficient (Cd)* – A dimensionless coefficient used to describe the surface upon which the water will be flowing around.
- *Units* – May be assigned to either "ft" (feet) or "m" (meters).
- *Porisity* – Represents the ability for water to flow through the obstruction object. A porosity of 0 represents a solid surface with no pores allowing for water to pass through the object and 1 represents a surface comparable to a wire mesh with many holes allowing water to pass through it.

Monitor Points Coverage

Monitor Points are optional. If choosing to have monitor points, create a monitor points coverage and add points using the **Create Feature Point** tool. A monitor point is used to gather specific information for that location at all time steps. Information calculated by SRH-2D at a monitor point includes position in the X and Y direction, bed elevation, water elevation, water depth, X direction velocity component, Y direction velocity component, velocity magnitude, Froude number, and shear stress.

If a Monitor Points coverage has been created, then it must be linked to the SRH-2D simulation. This can be done by dragging the coverage under the simulation item in the Project Explorer or right-clicking on the coverage and using the *Link to* menu item. When the simulation is launched, monitoring point data will be collected and outputted.

In SMS 11.2 only, the Monitor Points coverage must be assigned and selected in the *Select the Monitor Point Coverage* dialog for SRH-2D to recognize them. The dialog is accessed through the **Assign Monitor Points Coverage** command in the *SRH-2D* menu. This process is obsolete in SMS 12.0 and higher.

Materials Coverage

The SRH-2D materials coverage allows creating material zones specific to the SRH-2D model. The materials zones must cover the domain extents of the project.

Materials Coverage Right-Click Menu

The materials coverage has the standard coverage item right-click menu commands as well as specific right-click commands. The following commands are also part of the SRH-2D materials coverage:

- **Material Properties** – Opens the *Material Properties* dialog.
- **Assign Material To** – Opens the *Assign Materials* dialog for managing multiple material coverages.

Assign Materials

SRH-2D allows the option to have multiple material coverages. When multiple material coverages exist, there are options to manages these coverages. By right-clicking on an SRH-2D materials coverage and selecting the **Assign Materials to...** command, the *Assign Materials* dialog can be accessed. This dialog allows adding the unique material definition of one material coverage to that of another material coverage. Select the target material coverage that will be modified by the material coverage used to access the dialog. There are then two option for how SMS will modify the target material coverage.

- *Append to target material list* – this will add any unique material definitions to the target material coverage.
- *Replace target material list* – deletes the existing material definitions in the target material coverage and replaces them with the selected material coverage.

After performing either of these actions, opening the *Material Properties* dialog for the target material coverage will show the material list modified with the new definitions.

Assigning Materials to Polygons

Materials can be assigned to individual polygons using the **Select Polygon** tool and double-clicking on a polygon in the materials coverage. This will bring up the *Assign Material Properties* dialog.

The *Assign Material Properties* dialog can also be reached by right-clicking on a polygon in the materials coverage and selecting the **Assign Material Properties** command.

See SRH-2D Material Properties for more information about the *Assign Material Properties* dialog.

Sediment Materials Coverage

This coverage is similar to the Materials coverage, however, the material zones in this coverage are meant for assigning sediment properties. The coverage has the same right-click menu options as the Materials coverage. Sediment materials are assigned to polygons. Thickness and density values can be assigned to sediment material polygons.

Related Topics

- SRH-2D

SRH-2D Boundary Conditions

SRH-2D supports a variety of boundary conditions for hydraulic computation. The boundary conditions for the model are specified through the SRH-2D Boundary Conditions coverage by selecting an arc or by selecting a nodestring in the mesh. Once the arc or nodestring has been selected, double-clicking, or right-clicking and using the **Assign Linear BC...** command, will bring up a dialog where the boundaries can be assigned.

All currently supported boundary types are exterior boundaries with the exception of a monitor line. Other boundary types include the following: Inlet-Q, Inlet-SC, Exit-EX, Exit-H, Exit-Q, Wall, and Symmetry.

SRH-2D Linear BC Dialog

The *SRH-2D Linear BC* dialog contains multiple boundary condition types. Selecting a type for an arc will determine what boundary conditions can be assigned to the arc.

Inflow and Outflow Boundary Conditions Options

SMS provides options for the following boundary conditions:

Inlet-Q

A subcritical inlet boundary that may be given as a constant discharge or as a variable discharge hydrograph. The velocity distribution type may be selected from the boundary condition menu.

- *Discharge (Q)* – Specify as either a "Constant" or a "Time Series".
  - "Constant" – Use for a steady state simulation.
    - *Constant Q* – Enter a real positive value along with the units as either "cfs" or "cms".
  - "Time Series" – Use for unsteady flow.
    - *Time Series Files* – Clicking the button will bring up the *XY Series Editor* . Units must be specified as either "cfs" or "cms".
- *Distribution at Inlet* – The lateral distribution of the velocity. Options include:
  - "Conveyance" – Calculates the conveyance parameter first then the velocity. The flow direction is assumed to be normal.
  - "Profile" – Uses a depth average velocity profile.
  - "Q" – A constant unit discharge, q=vh, is assumed with flow direction normal to the inlet boundary.
  - "Velocity" – A constant velocity magnitude is imposed at the inlet with flow direction normal to the inlet boundary.

Inlet-SC

A supercritical inlet boundary that may be given as a constant discharge or as a variable discharge hydrograph. Inlet-SC also requires information about depth. Within the boundary condition menu, the velocity distribution type may be specified.

- *Supercritical Inflow Options* – Specify the discharge type and water surface elevation units.
  - *Discharge (Q)* – Set as one of the following:
    - "Constant" – Use for a steady state simulation.
    - "Time Series" – Use for unsteady flow.
  - *Units* – Can be set as "U.S. Customary" or "Metric".
- *Supercritical Constant Inflow Options* – Enter values for a steady state simulation.
  - *Discharge (Q)* – Enter a real positive value for the flow.
  - *Water Elevation (WSE)* – Enter a real positive value for the water surface elevation.
- *Supercritical Varied Inflow Options* – Use for unsteady flow.
  - *Time Series Discharge File* – Clicking the button will bring up the *XY Series Editor* for entering the varied discharge values.
  - *Water Elevation (WSE)* – Can be entered as either a "Time Series" or a "Rating Curve"
  - *Time Series vs. WSE File* – Clicking the button will bring up the *XY Series Editor* for entering the time versus elevation values.
  - *Q vs. WSE Files* – Clicking the button will bring up the *XY Series Editor* for entering the flow versus elevation rating curve values.
- *Distribution at Inlet* – The lateral distribution of the velocity. Options include:

    - "Conveyance" – Calculates the conveyance parameter first then the velocity. The flow direction is assumed to be normal.
    - "Profile" – Uses a depth average velocity profile.
    - "Q" – A constant unit discharge, q=vh, is assumed with flow direction normal to the inlet boundary.
    - "Velocity" – A constant velocity magnitude is imposed at the inlet with flow direction normal to the inlet boundary.

Exit-H

A stage type exit boundary where water surface elevation may be given as a constant number or as a stage-discharge or rating curve.

- *Water Elevation(WSE)* – Set as either a constant, a time series, or a rating curve.
    - "Constant" – Use for a steady state simulation.
        - *Constant WSE* – Enter a positive value for the water surface elevation.
        - **Populate** – Launches the *Populate* dialog to automatically generate an estimated constant.
    - "Time Series" – Use for unsteady flow.
        - *Time Series vs. WSE file* – Clicking the button will bring up the *XY Series Editor* for entering the time versus elevation values.
    - "Rating Curve" – Use for unsteady flow.
        - *Q vs. WSE file* – Clicking the button will bring up the *XY Series Editor* for entering the discharge versus elevation rating curve values. The *Populate* dialog can be accessed from the *XY Series Editor* to automatically generate an estimated rating curve.

Exit-Q

An exit boundary with a discharge given as a constant number or as a hydrograph.

- *Discharge (Q)* – Specify as either a "Constant" or a "Time Series".
    - "Constant" – Use for a steady state simulation.
        - *Constant Q* – Enter a real positive value along with the units as either "cfs" or "cms".
    - "Time Series" – Use for unsteady flow.
        - *Time Series Files* – Clicking the button will bring up the *XY Series Editor* . Units must be specified as either "hrs-vs-cfs" or "hrs-vs-cms".

Wall

Solid wall boundaries may represent banks and islands. No-slip condition is assumed at solid walls for the dynamic wave solver.

- *Add Extra Roughness at Wall Boundary* – Can be turn "On" or "Off".
    - *Roughness Height in mm* – Only available with the "On" option.

Boundary Conditions Without Additional Options

The following boundary conditions do not have additional options.

- "Exit-EX" – A supercritical exit boundary condition.
- "Symmetry" – Defined as a boundary where all dependent variables are extrapolated assuming the gradient of the variable in a direction normal to the boundary is zero except the velocity component normal to the boundary.
- "Monitor-Line" – Not a real boundary at all; it is an internal polyline which may be used to monitor the total flow discharge through it.

Structure Boundary Conditions

In general, two arcs must be selected before assigning arcs a structure boundary condition.

- Weir – See the SRH-2D Structures article for more information.
- Culvert – See the SRH-2D Structures article for more information.
- Pressure zone – See the SRH-2D Structures article for more information.
- Gate – See the SRH-2D Structures article for more information.

Related Topics

- SRH-2D Boundary Conditions Coverage
- SRH-2D

SRH-2D Files

The available input and output files for SRH-2D are listed below.

SMS Input Files

Name	Description
DB3	dBASE III SRH-2D Information
H5	2D Scatter XMDF Information
MAP	Mesh Arcs Information
MATERIALS	Materials Types Information
PRJ	Projection

SRH-2D Input Files

Name	Description
DAT	Pre-SRH File
SOF.DAT	Pre-SRH Script Output File

Pre-SRH Input Files

Name	Description
SRHGEOM	Mesh Geometry
SRHHYDRO	Model Control
SRHMAT	Mesh Material
SRHMPOINT	Monitor Points
XYS	BC Flow Value Curve

SRH-2D Output Files

Name	Description
DIA.DAT	Diagnostic Grid Depth Values XMDF

DIP.DAT	Dynamic Input Telescoping Grid
INF.DAT	Courant–Friedrichs–Lewy Residuals
EXIT1.DAT	Q and WSE Time step Averages
LN#.DAT	Monitor Line Results
OUT.DAT	Model Run Summary Output
PT#.DAT	Monitor Point Results
RES.DAT	Time step Residuals
RST#.DAT	Restart (Hotstart) Result Output
TSO.DAT	Time step Series Output Index
XMDF.H5	Output WSE, Depth, Velocity

An explanation of files used by and generated by SRH-2D are as follows:

Output Files

A description of each file generated during an SRH-2D simulation run is as follows. In the file descriptions, * is a placeholder representing the specific case name as specified in the model control:

***.DAT**

File created when SRHpre is run, for use by SRH-2D. It contains model input information as well as geometry information about the mesh.

***_DIA.dat**

Diagnostic file with potential errors and warnings about the execution. It helps to identify causes of execution error or failure. For the tutorial case, the file is almost empty indicating a successful run of the model.

***_DIP.dat**

Dynamic Input file allows setting up or modify frequently used parameters during an SRH-2D execution. Parameters that can be set up or modified include the total simulation time, number of iterations within each time step, specification of restart files, time interval used for writing out intermediate results, time step interval, damping, relaxation for continuity and momentum equations, and the turbulence model type. See SRH-2D documentation for more information about the implementation of this file.

***_LNn.dat**

Monitor line file where flow discharge is recorded corresponding to time.

***_OUT.dat**

Output file providing general model information such as input parameters, mesh size, list of restart file numbers and their corresponding time, cpu time of the simulation, ect.

***_PTn.dat**

Monitor point file that provides time history of output variables at the user-specified monitor points. The file is in column format and may be imported into Excel for plotting. Output from the file may be used to decide if a steady state solution has been obtained or to examine unsteady change of a variable. If additional monitor points are used, files would have a similar naming convention with the only change being PT2, PT3, etc.

***_RES.dat**

Residual file that contains residuals of continuity and two velocity equations during the solution. Note that residuals are normalized. For example, the ResH is normalized by the maximum of the first three iterations. Therefore, residual of 1.0 is obtained for ResH if NITER is less than 4 in the c1_DIP.dat file.

***_RSTn.dat**

Restart file used as a model input in successive runs. These are written out at an interval specified within the model control. If there is a restart file, there is an option to start a model run using it as the initial conditions of the model.

***_SOF.dat**

Script Output File generated when SRHpre is run. In the script output file all inputs are saved. Can be used to rerun SRHpre by changing the name to *_SIF.dat

***_XMDF.h5**

Output Extensible Model Data Format (XMDF) file used by SMS for post-processing and visualization of results. Results include water surface elevation, water depth, depth averaged velocity, froude number, and bed shear stress. If a model includes sediment transport, output results could also include critical sediment diameter and sediment transport capacity.

Native Files

SRH-2D makes use of native files. The four native files are *.SRHHYDRO, *.SRMAT, *.SRHMONITORPTS, and *.SRHGEOM as described below:

SRHHYDRO File

SRHHYDRO is written out by SMS to guide SRH-2D through the hydraulic simulation. The SRHHYDRO file contains key information about the simulation while acting as a directory to other files for SRH-2D to use. The SRHHYDRO file stores the case name, simulation description, model type, turbulence model information, Manning's n values, boundary conditions, boundary types, unsteady flow designation, simulation time, resultant output information, and initial conditions. Details of each card in the file are given as follows:

Case	This is an identifier for SRH-2D to use when running to help recognize the files that correspond to a specific project. The case should be given a name that is unique for a simulation.
Description	The description is to show in review of what was done for a specific simulation
RunType	This card tells SRH-2D what to compute. Flow means a hydraulic model. Mobile refers to a sediment transport model.
ModelTemp	This card communicates to SRH-2D whether the model will be used to simulate temperature. Currently, temperature is not supported by SRH-2D v. 2.2
UnsteadyOutput	Unsteady output is labeled for unsteady, where intermediate calculations are performed, or as steady, where only final calculations are computed for accuracy.
SimTime	Three numbers are given to specify start time (hours), time step (seconds), and total simulation time (hours).
TurbulenceModel	This option is either parabolic or ke for the current version of SRH-2D.
ParabolicTurbulence	This card is dependent on TurbulenceModel being labeled parabolic. The value is a constant used in the parabolic turbulence equation.
InitCondOption	This card communicates to SRH-2D the condition of each element prior the model run. Options include dry, auto, and rst, where rst represents a start-up file from a previous run.
Grid	This card tells SRH-2D the name of the grid file.
HydroMat	This card tells SRH-2D the name of the material file.
MonitorPtFile	This card tells SRH-2D the name of the monitor point file if one has been created.

OutputFormat	This option represents how SMS will write out the final files to be read back for post processing. Two inputs are required, the file type and the resultant units.
OutputInterval	This card tells SRH-2D how often to write out results during the simulation. The value is given in hours.
ManningsN	In this location two values are given representing the material number and the value of Manning's n corresponding to that material value. SMS will always write a zero material type as a default.
BC	This card refers to the boundary type. Two values are given representing the boundary number and the type of boundary for each boundary number
IQParams	This card will be written for boundary types that ask for a subcritical inlet boundary. The values given represent the boundary id, the constant flow value or variable flow file name, the units of flow, and the distribution type
ISupCrParams	This card requires the same information as IQParams with the addition of constant water surface elevation or varable water surface elevation file name.
EWSParams	This card represents the stage exit boundary. Values include the boundary id, the constant watersurface elevation or variable watersurface elevation file, and units type.
EQParams	This card gives the constant discharge value or variable discharge file name and unit type.
NDParams	This card refers to a normal depth outlet boundary. Values include the nodestring number at which flow will be computed as well as the average bed slope at the exit location.

The file acts as a map guiding SRH-2D to other important files such as the SRHMAT file, the SRHMONITORPTS file, and the SRHGEOM file.

SRHHYDRO Example

```
SRHHYDRO 30
Case "Case"
Description "Description"
RunType FLOW
ModelTemp OFF
UnsteadyOutput UNSTEADY
SimTime 0 1 3
TurbulenceModel PARABOLIC
ParabolicTurbulence 0.7
InitCondOption DRY
Grid "HohRiv.srhgeom"
HydroMat "HohRiv.srhmat"
MonitorPtFile "HohRiv.srhmpoint"
OutputFormat XMDF ENGLISH
OutputInterval 1
ManningsN 0 0.02
ManningsN 1 0.025
ManningsN 2 0.07
BC 6 WALL
BC 5 WALL
BC 4 MONITORING
```

```
BC 3 MONITORING
BC 2 EXIT-H
BC 1 INLET-Q
IQParams 1 "HohRiv.srhcurve1.xys" EN  CONVEYANCE
EWSParams 2 "HohRiv.srhcurve2.xys" EN
```

SRHMAT File

The SRHMAT file gives each element a material type. This file will categorize each element to a Manning's n value.

SRHMAT Example

```
SRHMAT 30
NMaterials 3
MatName 1 "Channel"
MatName 2 "Forest"
Material 1 1 2 12 14 15 23 24 26 27 28
 29 36 37 38 39 40 41 42 49 50
 51 52 53 54 55 56 63 64 65 66
 67 68 69 70 71 82 83 84 85 86
 87 88 89 90 91 103 104 106 107 108
 109 110 111 112 113 114 115 116 117 118
 119 120 121 132 133 134 135 136 137 138
 139 140 141 142 143 144 145 146 147 148
 149 150 151 152 153 154 155 156 157 158
 159 170 171 172 173 174 175 176 177 178
 179 180 181 182 183 184 185 186 187 188
 189 190 191 192 193 194 195 196 207 208
 209 210 211 212 213 214 215 216 217 218

Material 2 3 4 5 6 7 8 9 10 11 13
 16 17 18 19 20 21 22 25 30 31
 32 33 34 35 43 44 45 46 47 48
 57 58 59 60 61 62 72 73 74 75
 76 77 78 79 80 81 92 93 94 95
 96 97 98 99 100 101 102 105 122 123
 124 125 126 127 128 129 130 131 160 161
 162 163 164 165 166 167 168 169 197 198
 199 200 201 202 203 204 205 206 228 229
 237 238 239 240 241 242 243 244 245 246
```

SRHMONITORPTS File

The SRHMONITORPTS file or SRHMPOINT file is tells SRH-2D that there are monitor points to watch and where those points are located. SRH-2D will take the coordinates from SMS to locate the areas to be monitored.

SRHMONITORPTS Example

```
SRHMON 30
NUMMONITORPTS 2
```

```
monitorpt 1 798814 309513
monitorpt 2 799387 305853
```

SRHGEOM File

The SRHGEOM file tells SRH-2D where each element is located and the characteristics of that element. The SRHGEOM file holds information about the units of the grid.

SRHGEOM Example

```
SRHGEOM 30
Name "HohRiverDomain"

GridUnit "FOOT"

Elem 1 5 1 6 15
Elem 2 1 2 7 6
Elem 3 3 1 5
Elem 4 2 1 3
Elem 5 5 8 3
Elem 6 3 8 10
Elem 7 9 8 4
Elem 8 13 4 14
Elem 9 14 4 8
Elem 10 11 4 13
Elem 11 4 11 9
Elem 12 14 5 15 24
Elem 13 8 5 14
Elem 14 6 7 17 16
Node 1 798908 309671 169.545
Node 2 798857 309733 170.299
Node 3 798975 309744 171.463
Node 4 799084 309550 170.097
Node 5 798959 309609 169.67
Node 6 798877 309645 169.34
Node 7 798828 309705 170.831
Node 8 799047 309635 171.189
NodeString 6 2 3 10 19 29 40 52 69 90 118
 149 183 217 254 292 330 368 405 441 476
 513 548 585 621 656 687 716 744 771 797
NodeString 5 171 205 240 278 316 354 391 426 462 500
 536 574 610 646 679 710 740 767 793 819
 843 867 891 915 939 963 986 1008 1031 1032
```

Related Topics

- For more information on these files see the manual .
- SRH-2D

SRH-2D Menu

The *SRH-2D* menu becomes available when the SRH-2D model has been created and is active. In SMS 12.0 and above, this menu is found by right-clicking on the SRH-2D simulation. From the SRH-2D menu, the specifics of the overall model are entered and the model is initiated. The SRH-2D menu includes the following commands:

Model Control

This command will bring up the SHR-2D *Model Control* dialog.

Model Check

This command will cause SMS to perform a check for invalid settings that could prevent SRH-2D from running properly. If errors are found, they will be reported in the *Model Check* dialog with information on how to correct them.

Export SRH-2D

Tells SMS that it is time to write out the files that SRH-2D needs to run. Note that the project must be saved before the files may be exported. The user will not see anything happen, but SMS uses the input information from the model control, the boundary conditions, and the materials data to create the files in preparation for SRH-2D to work.

Launch SRH-2D

When the model is ready and has been exported, it may be launched. Launching SRH-2D initiates the powerful, robust hydraulic model. Upon successful completion of the launch, the analysis is complete and results are ready to be read and displayed by SMS.

Save, Export, and Launch SRH-2D

This is a combinination of the previous steps put together into one.

The SRH-2D simulation right-click menu also has general simulation commands .

Obsolete Commands

The following commands are no longer available in current versions of SMS.

Assign Monitor Points Coverage

Brings up the *Select the Monitor Point Coverage* dialog.

Related Topics

- SRH-2D

SRH-2D Material Properties

The SRH-2D model has its own materials options. These options become available when SRH-2D is active. Individual SRH-2D items may have their own material properties. This allows having more than one set of material definitions in SRH-2D.

The SRH-2D Materials Coverage should be used for assigning materials to an SRH-2D simulation.

Dialog Description

The SRH-2D *Material Properties* dialog is accessible from the menu command *Edit* | **Materials Data** when a SRH-2D item is active. When the active item is not specified as part of SRH-2D, using the **Materials Data** command will bring up the global *Materials Data* dialog.

The *Material Properties* dialog can also be reached by right-clicking on an SRH-2D materials coverage item in the Project Explorer and selecting the **Material Properties** command. Right-clicking on a polygon with the **Polygon Selection** tool and choosing the **Assign Material Properties** command will also bring up the *Material Properties* dialog.

The *Material Properties* dialog is sometimes labeled as the *Assign Material Properties* dialog. The *Material Properties* dialog windows has the following elements:

Materials List

The *Materials* list contains two columns (*Name*, and *Color*) for the defined materials. The default "unassigned" 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 **Color** button in the *Color* column. To select a color and pattern, click on the button to open the *Choose* window.

Choose Window

Clicking on the button under the *Color* column will open a new *Choose* dialog. In the *Choose* 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. For more color options, click on the color button to bring up the *Select Color* dialog. Select a pattern by making a selection in the *Texture* section of the *Choose* window.

Insert and Remove Buttons

- Inserts a material into the spreadsheet with the lowest unique ID available and a default name and pattern.
- Removes the currently selected material from the spreadsheet.

Manning's Roughness Coefficient

This section of the Material Properties dialog edits the Manning's *n* or depth varied n for the select material. By default, a single default Manning's *n* of "0.02" is assigned by SMS if no Manning's *n* is indicated. It is possible to select multiple *n* values as a function of depth and then enter depth/ *n* pairs.

The Manning's *n* value can be set to a "Constant" or "Depth Varied" value. The "Depth Varied" option will display an option to enter values in an *XY Series Editor* .

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 Option* s dialog. These options are the same as in the *Materials Data* dialog.

Related Topics

- SRH-2D
- SRH-2D Coverages
- Materials Data

SRH-2D Model Control

The SRH-2D *Model Control* dialog can be reached by using the **Model Control** command in the *SRH-2D* menu. From the model control, enter data that will be written the files used by SRH-2D for computation.

General Tab

In the *General* tab, specify the name of the case used for the SRH-2D model run. The following options are available:

- *Simulation Description* – For noting information that may be useful later when reviewing the model. Information entered as the *Simulation Description* are not used the SRH-2D model for computation.
- *Case Name* – A keyword that SRH-2D will use to link files together of the same case. A simple word without spaces is needed by the model.
- *Temperature Modeling*

  - *Use temperature modeling*
- *Start Time (hours)* – The default is left at zero, but it may need to be adjusted to meet the time specifications of input hydrographs or other data.
- *Time Step (seconds)* Used by SRH-2D to make calculations. Depending on the model, the element size, and the input parameters, the time step will need to be adjusted.
- *End Time (hours)* – Determines the total period of time the simulation run will analyze.
- *Initial Condition* – Communicates to SRH-2D how each cell is to be treated upon launch. The following options are available:
  - "Dry" – The defualt setting in SMS. This suggests that each element does not have water or is not wet.
  - "Initial Water Surface Elevation" – Allows changing the *Initial Water Surface Elevation* for the simulation run. This sets the water surface elevation for all elements to one value. If the elevation at an element is greater than the water surface elevation specified, then the element is dry.
  - "Automatic" – Assumes backwater from the downstream boundaries with areas above this value are dry.
  - "Restart File" – Activates a **Select** button which brings up a browser to select an RST file from a previous run on the same mesh as the initial condition for each element. When a simulation is run, SRH-2D will write out a restart file for each output interval as specified in the *Output* tab.

Obsolete Options

- *Run Type* – Communicates to SRH-2D the type of analysis that is to be preformed. Options include the "Flow" option for hydraulic simulations and the "Morphological Analysis" and "Mobile" options for sediment transport simulations.
- *Unsteady Output* – Specifies whether a run will use steady or unsteady output. Selecting "On" will specify an unsteady output and selecting "Off" will specify a steady output. Using steady output will allow for quicker computation time; whereas, unsteady output requires more time for calculation. The difference lies between the intermediate results. The final results of a steady or unsteady analysis will be the same; however, unsteady allows for accurate intermediate results.

Flow Tab

Under the *Flow* tab are the options for the turbulence model used in the simulation. The current release of SRH-2D support the use of the the following turbulence models:

- Laminar
- Constant
- Parabolic
- KE

Using "Parabolic" requires the input of a constant for the turbulence equation. The default value is left as "0.7". Consult the SRH-2D User's Manual or other reference materials to find the value that is most appropriate for their run.

Output Tab

The *Output* tab has options that will tell SRH-2D what is to be written out as results and how often.

- *Result Output Format* – Specifies the type of file that will be written with results. SRH-2D supports and writes out to the following file formats:
  - XMDF
  - SRHN
  - SRHC
  - TECPlot

- *Result Output Unit* – Option dictates to SRH-2D the units to be included in the output file whether they be English or International (SI) units.
- *Result Output Method* – Allows the following options:
  - Specified Frequency
  - Specified Time
  - Simulation End
- *Result Output Fequency (hours)*

Related Topics

- SRH-2D

SRH-2D Populate Dialog

The *Populate* dialog is used to generated an estimate flow discharge on an "Exit-H" boundary condition.

Populate for Constant

This dialog is reached through the **Populate** button in the *SRH-2D Linear BC* dialog when the "Exit-H (subcritical outflow)" option is selected and the Water Elevation (WSE) is set to "Constant". The dialog is used to generate a best estimate water surface elevation constant value.

- *Type* – Select either a normal or critical depth.
  - "Normal depth" – Depth of water under conditions of steady, uniform flow.
  - "Critical depth" – Depth of flow where energy is at a minimum discharge.
- *Ground Elevation Dataset* – Brings up a *Select Tree Item* dialog were an elevation dataset can be selected as wel as any time steps in the dataset can be chosen.
- *Units* – Can select either "U.S. Units" or "SI Units (Metric)".
- *Composite Mannings N* – Enter a roughness value.
- *Slope* – Water surface slope. Available only for "Normal depth".
- *Flow* – Enter a constant flow value.
- **Plot** – Generates a plot of the WSE flow.

Populate for Rating Curve

This dialog is reached through the **Populate** button in the *XY Series Editor* . The **Populate** button only appears in the *XY Series Editor* when the editor is reached through *SRH-2D Linear BC* dialog when the "Exit-H (subcritical outflow)" option is selected and the *Water Elevation (WSE)* is set to "Rating Curve". The dialog will generate a best estimate of the rating curve variables. The variables will be automatically entered for the rating curve in the *XY Series Editor* when exiting this dialog.

- *Type* – Select either a normal or critical depth rating curve.
  - "Normal depth rating curve" – Depth of water under conditions of steady, uniform flow.
  - "Critical depth rating curve" – Depth of flow where energy is at a minimum discharge.
- *Ground Elevation Dataset* – Brings up a *Select Tree Item* dialog were an elevation dataset can be selected as wel as any time steps in the dataset can be chosen.
- *Units* – Can select either "U.S. Units" or "SI Units (Metric)".
- *Composite Mannings N* – Enter a roughness value.
- *Slope* – Water surface slope. Available only for "Normal depth rating curve".
- *Populate Flows* – Generate the flow values by entering the following values and clicking the **Add** button.
  - *Min* – Minimum flow value.
  - *Max* – Maximum flow value.
  - *Delta* – Percentage difference.
  - **Add** – Generates the flow rating curve based on the *Populate Flow* inputs. The values are shown in the *Flow (Q)* table.
- *Flow (Q)* – Table that shows the populated flow values that will be used in the rating curve. Values in the table can be manually edited. New values can be added using the button and deleted using the button.
- **Plot** – Generates a plot of the WSE flow.

Related Topics

- SRH-2D Boundary Conditions
- SRH-2D Boundary Conditions Coverage

SRH-2D Simulation

SRH-2D makes use of simulations starting in SMS 12.0 and later. Simulations are useful as multiple simulations can be used in the same project.

Simulations for SRH-2D are created by right-clicking in the Project Explorer and selecting *New Simulation* | **SRH-2D** . The simulation will appear in the Project Explorer under the "Simulation Data" item.

Right-clicking on the simulation gives access to the SRH-2D menus , including access to the SRH-2D *Model Control* dialog. Each SRH-2D simulation can have its own parameters based on what is entered into the model control.

SRH-2D Components

Components for SRH-2D usually include:

- A 2D mesh
- A boundary condition coverage
- A material coverage
- A monitor point coverage (optional)
- An obstructions coverage (optional)

A simulation cannot include multiple coverages of the same type. So if building multiple boundary condition coverages, only one of the boundary condition coverages can be included in the simulation. Coverages can be merged if needed.

Linking Components

After a simulation has been created, components may be added to the simulation. Components are usually added by clicking on the component item in the Project Explorer and dragging the item under the SRH-2D simulation. A link is then created between the component and the simulation. If the component is updated, it is updated automatically in all simulations that are linked to it.

Components can also be added to a simulation by right-clicking on the component in the project explorer and selecting the simulation name in the *Link To* submenu. Simularly, components can be unlinked from a simualtion by right-clicking on the component and selecting the simulation name in the *Unlink From* submenu. The *Link To* and *Unlink From* submenus become available once a simulation has been created.

Running an SRH-2D Simulation

After all components have been added to the simulation and the model parameters have been established, the simulation can be run. This is done by right-clicking on the simulation and choosing the **Launch SRH-2D** command or the **Save, Export, and Launch SRH-2D** command.

The SRH-2D Model Wrapper should appear. If there are any errors in the model run, the model wrapper will exit early. The model run can also be exited early by clicking the **Abort** button. When the model run is completed, select **Exit** to close the model wrapper.

The *Load solution* option in the model wrapper is selected by default. This option will load the solution from the SRH-2D model run in to SMS as soon as the model run is complete.

Running SRH-2D requires both the pre-SRH-2D executable and the SRH-2D executable. If SMS cannot find these executables, the model run will encounter an error. The *File Locations* tab in the *Preferences* dialog can be used to specify the path to these executables if SMS cannot find them.

Related Topics

- SRH-2D
- Simulations

SRH-2D Structures

SRH-2D has the capability to model hydraulic structures including: bridges, culverts, gates, and weirs.

The structure boundary conditions were added to the SRH-2D model at the request of FHWA in 2015 and included in the interface for SMS 12.0. General structure requirements include:

- Each structure is specified by selecting two arcs which represent the upstream and downstream faces of the structure.
- The two arcs *must be ordered in the same direction* (i.e. left to right as looking downstream). If this is not the case, model results are will not represent the structure.
- In the region of structures that represent one dimensional flow (Weir, Culvert, Gate) the two dimensional flow is disabled. SRH-2D will identify cells/elements enclosed by the upstream and downstream structure definitions (connecting the end points of the arcs/nodestrings) and disable the elements. In order to make the display in SMS match what the model is doing, the user will generally create this polygon in the material coverage and assign the area to the disabled material type.

Culvert

A structure that allows water to flow under a road, railroad, trail, or similar obstruction. SRH-2D can make use of culvert structures by coupling with the Federal Highway Administrations HY-8 culvert analysis application.

Specific guidelines include:

- It is necessary to specify a file (path and name) where all HY-8 crossing definitions used in this boundary condition coverage are stored. This is accessed by right-clicking on the coverage and selecting HY-8 **Options** .
- It is necessary to define the crossing before running SRH-2D. This is done by creating two arcs representing the upstream and downstream portion of the culvert then assigning the both arcs the "Culvert" *Type* in the *Linear BC* dialog. The crossing definition can be accessed in the boundary condition dialog after assigning the arcs to be a culvert.
- HY-8 computes flow both through the culvert and over the road/weir above the culvert. Attributes are specified in the crossing definition.

The boundary conditions options for culvert arcs include:

- Object *Role* – Allows assigning each arc to be either "culvert upstream" or "culvert downstream".
- *Units for HY-8 Display*
- **Launch HY-8** – Starts the HY-8 software and opens the culvert file that was specified in the coverage *Options* dialog.

Gate

Gate structures can simulate both overflow and underflow gates. Adding a gate structure to the boundary conditions coverage requires creating two arcs and assigning both the arcs the "Gate" *Type* in the *Linear BC* dialog.

The boundary conditions options for gate arcs include:

- Object *Role* – Allows assigning each arc to be either "gate upstream" or "gate downstream".
- *Units* – Can select feet or meters.
- *Crest Elevation*
- *Height of Gate Opening*
- *Width of Gate Opening*
- *Contract Coefficient with Underflow Orifice*
- *Type* – Allows assigning "paved", "gravel", "single", "double", "sharp", "broad", or "user" defined.

Pressure Flow Bridge

A bridge can be represented as a pressure flow boundary condition in SMS.

- The pressure zone definition defines a ceiling elevation that can vary linearly from the upstream arc to the downstream arc. This could also be side to side if the arcs are defined as the sides of the bridge instead of upstream/downstream. More complex ceiling elevation variation can be simulated using multiple pairs of pressure flow arcs.
- The pressure zone cannot touch any mesh/grid boundaries nor can voids in the mesh be defined in the pressure zone.
- Pressure zones do not allow any overtopping.

The boundary conditions options for pressure flow arcs include:

- Object *Role* – Allows assigning each arc to be either "gate upstream" or "gate downstream".
- *Units* – Can select feet or meters.
- *Ceiling elevation along upstream*
- *Ceiling elevation along downstream*
- *Manning roughness coefficient between water and ceiling*

Weir

A barrier across a river designed to alter its flow characteristics. In SMS, a weir can be represented as two arcs. One arc represents the upstream face of the weir structure and the other represents the downstream face of the weir.

Typically a weir would be represented in the geometry of the mesh/grid. One application of the weir structure would be to quickly investigate the impact of inserting a weir with a variety of crest elevations (designing the structure). In this case, the same grid/mesh could be used for all simulations.

The boundary conditions options for weir arcs include:

- Object *Role* – Allows assigning each arc to be either "gate upstream" or "gate downstream".
- *Units* – Can select feet or meters.
- *Crest elevation*
- *Length of weir*
- *Type* – Allows assigning the following:
  - *Paved roadway* – Flow over a roadway embankment with a paved surface
  - *Gravel roadway* – Flow over a roadway embankment with a gravel surface
  - *Single railroad track* – Flow over a railroad track embankment with one set of rails
  - *Double railroad track* – Flow over a railroad track embankment with two sets of rails
  - *Sharp crested weir* – Flow over a sharp crested weir
  - *Broad crested weir* – Flow over a broad crested weir

- *User defined* – Flow over a user defined weir

Related Topics

- Boundary Conditions Coverage

6.6. Steering

Steering

The steering tool has been added to facilitate the process of launching models multiple times. To launch the steering tool, choose the menu command *Data* | **Steering Module** .

The steering tool can be used for single model (spin down) simulations of RMA2 and FESWMS .

In SMS version 12.0 and earlier the steering tool can also be used to facilitate the transfer of data from wave models to circulation models and back. Currently, CMS-Flow / CMS-Wave links is supported. In versions 12.1 and later, the CMS-Flow Model control is used for steering.

Related Topics

- SMS Models
- RMA2 Spindown
- FESWMS Spindown
- CMS-Flow / CMS-Wave Steering

NOTE: CMS-Flow (formerly known as M2D) and CMS-Wave (formerly known as WABED) are components of the Coastal Modeling System (CMS).

External Links

- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [185]
    - Please see this forum post for an explanation of ADCIRC and STWAVE steering
- Jun 2002 ERDC/CHL CHETN-IV-42 Coupling of Regional and Local Circulation Models ADCIRC and M2D (now know as CMS-Flow) [186]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D (now know as CMS-Flow) and STWAVE [187]

6.7. TABS-MD (Multi-Dimensional) Numerical Modeling System – RMA2/RMA4

TABS

The TABS-MD (Multi-Dimensional) Numerical Modeling System was one of the first widely used collection of programs designed for studying multi-dimensional hydrodynamics in rivers, reservoirs, bays and estuaries. The hydrodynamic engine for the system is the RMA2 engine. RMA2 and RMA4 were written by Resource Management Associates, Lafayette, California, and modified by WES. SED2D was written jointly by Resource Management Associates and WES.

Models

GFGEN

The pre-processor for the TABS software programs. This utility converts ASCII geometry into binary format and does data checking along the way. SMS will launch this utility as needed before running other components of the system. See GFGEN for more information.

RMA2

A one-dimensional/two-dimensional numerical model for depth-averaged flow and water levels. RMA2 is a two dimensional depth averaged finite element hydrodynamic numerical model. It computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields. RMA2 computes a finite element solution of the Reynolds form of the Navier-Stokes equations for turbulent flows. Friction is calculated with the Manning's or Chezy equation, and eddy viscosity coefficients are used to define turbulence characteristics. Both steady and unsteady state (dynamic) problems can be analyzed.

It should be noted that the commercially available version of the model does not include all functionality included in the ERDC documentation.

See RMA2 for more information.

RMA4

A one-dimensional/two-dimensional numerical model for depth-averaged transport. This program uses a provided hydrodynamic solution (either node by node or in RMA2 format) to compute transport of a constituent in solution. It is assumed that the depth concentration distribution is uniform. While model documentation claims up to six constituents can be considered simultaneously, practical application has shown the only application of multiple constituents to be DO/BOD. Either conservative or non-conservative diffusion is computed. See RMA4 for more information.

SED2D

Formerly STUDH, a two-dimensional numerical model for depth-averaged transport of cohesive or a representative grain size of noncohesive sediments and their deposition, erosion, and formation of bed deposits. The interface for this model has been removed from version SMS 10.0. Boundary condition files can still be constructed/edited in text editors and SMS can still read solution data.

RMA10

A one-dimensional/two-dimensional/three-dimensional hydrodynamic numerical model. Not available for public use at this time.

Using the Model / Practical Notes

- The TABS models are built to expire after a set date. The latest version can be downloaded from the Downloads section of the Aquaveo website .

Related Links

- SMS Models page

External Links

- CHL TABS Numerical Modeling website [188]
- CHL RMA2 Frequently Asked Questions [189]

TABS Attribute Dialog

TABS models have attribute dialogs for both points and arcs.

TABS Point Attributes Dialog

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

- Refine point (checked = on)
  - Element Size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.
- Assign 1-D Geometry – Opens the *GWN Card Definition* dialog

TABS Arc Attributes Dialog

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

- Arc Type
  - **None**
  - **Boundary Conditions – Options** button opens the RMA2 Nodestring Boundary Conditions dialog
  - **Continuity Check/Flux**

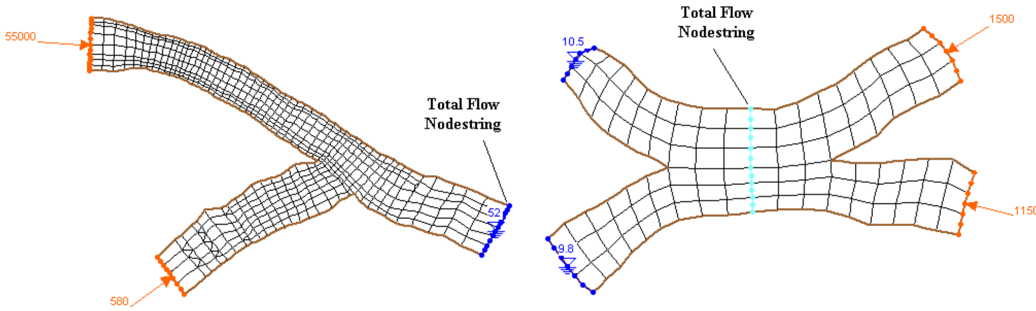
Related Topics

- Feature Objects Menu
- Mesh Generation

Total Flow Nodestring

Two hydrodynamic models supported by SMS, RMA2 and FST2DH , support a concept called the Total Flow Nodestring. These models print out a table of continuity or flux checks across specified nodestrings. Such checks are used to determine if there is mass loss through the model, to determine the amount of flow in various sections of a braided stream, or other similar purpose. In order to participate in the continuity table, the nodestring must either be assigned as a boundary condition or assigned as a flux (FST2DH) or geometric continuity (GC) (RMA2) string.

The nodestring assigned as total flow will always be reported first in the table, and will always have 100% of the flowrate across it. The flux across all other cross sections will be reported in the continuity table as a percentage of this flowrate, whether it be more, less, or equal to 100%. Note that nodestrings should all be created from right to left, while looking downstream. When a nodestring is selected, arrows appear on its ends which show the assumed downstream direction. The figures below show advisable locations of the total flow nodestring.



Related Topics

- Mesh Module

6.7.a. GFGEN

GFGEN

GFGEN (**G** eometry **F** ile **GEN** erator) converts ASCII geometry into binary format and does data checking along the way. GFGEN is capable of doing the following:

- Reading node and element data and constructing a finite element computational mesh for use by other programs in the TABS-MD modeling system.
- Identifying errors and potential errors in the constructed mesh.
- Renumbering the mesh, omitting unused nodes and/or element numbers.
- Fitting curved element sides to land boundaries and interior element sides may be specified.
- Developing an elemental solution order that permits the most efficient operation of models using the mesh.
- Providing a nodal cross reference and statistics summary.
- Creating a binary data file that contains the mesh information and geometry in a format suitable for use by other TABS-MD programs.
- Creating a device independent DISSPLA meta file of one to five mesh characteristics (mesh, node and element numbers, material types, and bathymetric assignments).

Primarily, GFGEN is used in conjunction with RMA2. It will run the first time a RMA2 project model run is launched.

Using the Model / Practical Notes

- The TABS models are built to expire after a set date. The latest version can be downloaded from the Software Updates section of the Aquaveo website .
- GFGEN Executable Known Issues

External Links

- Users Guide to GFGEN - Version 4.35 [190]

Related Topics

- SMS Models Page
- TABS Models
- RMA2
- RMA4

GFGEN Executable Known Issues

Unfortunately there are some known issues and likely some unknown issues as well related to the GFGEN executable. This page is intended to help brings awareness of known issues and how to work around them.

GFGEN (TABS) is developed by Resource Management Associates, Lafayette, California, and modified by WES . Aquaveo does not have access to the GFGEN source code and cannot make changes to fix the issues found in the GFGEN numerical model. Aquaveo can fix problems in the SMS interface, used for pre and post processing of GFGEN files.

This list may not contain all know issues, so please feel free to add to it.

Node Location Truncation

GFGEN does not correctly calculate the midside node location if the x or y coordinate of a node are large. Errors may be similar to the following:

```
* Node     50 Violates MIDDLE THIRD RULE *****
The node lies       0.0831 of the way from node      1 to node     49
```

```
=== Problem inside COEFS routine. Do 180.
Element= 535 Gauss pt= 1 out of NGP= 16
Divide by DETJ= -0.0229
J11-J22-J12-J21= 1.1526 0.38006 0.74030 0.62257
stop in coefs
```

The current work around is to translate the mesh so the x and y coordinates are smaller values.

See the following forum posts:

- RMA2 – same mesh, same BC, different results?, Results differ depending on nodal coordinates
- Problem inside COEFS routine

Element Limit

GFGEN can only handle up to 100,000 elements.

Related Topics

- GFGEN

6.7.b. RMA2 – Resource Management Associates

RMA2

RMA2	
Model Info	
Model type	Two-dimensional depth averaged finite element hydrodynamic numerical model. Computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields.
Developer	Resource Management Associates United States Army Corps of Engineers (USACE)
Web site	RMA2 web site
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • RMA2 • RMA2 Steering

RMA2 is a 1D/2D hydrodynamic model using the finite element method. RMA2 was written by Ian King and is maintained by the Army Corp of Engineers Engineering Resource Development Center (ERDC) . RMA2 has been applied to multi-dimensional problems since the mid 1970s. As such, it was one of the first widely used multi-dimensional hydrodynamics engine applied to riverine and estuarine applications. The current interface to RMA2 in the Surface-water Modeling System originated as a program named FastTABS.

The RMA2 model can be added to a paid edition of SMS.

Functionality

RMA2 has these capabilities:

- Compute water levels and velocities for a 2D mesh structure

- 1D elements (trapezoidal shape)
- 1D flow control structures
- Allow elements to wet and dry
- 2D structures are permitted in version 4.50 or higher (US Army Corps of Engineers employees only)
- Account for effects of the earth's rotation (Coriolis effect)
- Accepts a wide variety of boundary conditions including:
    - Discharge by node/element/ or line
    - Tidal radiation boundary conditions by line
    - Discharge as a function of elevation by line
    - Water surface elevation along a line
    - Apply wind stress either
        - Uniformly over the model domain; constant or time-varying
        - As a storm; front or tropical cyclonic event (time-varying)
- Read an STWAVE radiation stress file to incorporate wave induced currents
- Identify errors in the computational mesh specification
- Accept either English or standard SI units
- Restart (Hotstart) the simulation from a prior RMA2 run and continue
- Account for Marsh Porosity wetting and drying (wetlands)
- Employ either direct or automatic dynamic assignment of Manning's n-value by water depth
- Employ user selectable manual or automatically assigned turbulent exchange coefficients
- Compute flow across continuity check lines.
- Revisions within a time step (both coefficients and/or boundary conditions)

Using the Model / Practical Notes

- RMA2 Spindown Steering (Incremental Loading)
- RMA2 2D Control structures are only available to employees of the US Army Corps of Engineers.
- The TABS models are built to expire after a set date. The latest version can be downloaded from the Software Updates section of the Aquaveo website .
- According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, including storms in simulations remains experimental."
- Boundary condition time series curves are resampled by the model based on the time step. If changing the time step, re-enter the time series curve since SMS will automatically resample the curve and resampling the curve multiple times may result in a loss of important information.

Graphical Interface

SMS provides a graphical interface that is designed to visualize the projects being created, easily modify project parameters, and view the solutions produced by the RMA2 model. See RMA2 Graphical Interface for more information.

The RMA2 Graphical Interface contains tools to create and edit an RMA2 simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to RMA2. If a mesh has already been created for a RMA2 simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to RMA2. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the RMA2 Menu . See RMA2 Graphical Interface for more information.

Saving RMA2

When completing the *File* | **Save As...** command, the following files get saved in the *.sms

- *.mat referenced to new save location
- *.sim referenced to new save location

Related Topics

- SMS Models Page
- TABS Models
- RMA4
- Nodal Transition Dialog
- Roughness Options Dialog
- Rainfall Values Dialog

External Links

- CHL RMA2 Website [191]
- CHL RMA2 Frequently Asked Questions [192]
- Users Guide To RMA2 WES Version 4.5 [193]
- Users Guide to RMA2 Version 4.35 [194]
- MERGAVE – Utility for Merging RMA2 Solution Files [195]

Nodal Transition (Marsh Porosity) Dialog

The *RMA2 Nodal Transition (Marsh Porosity)* dialog is used to define the RD Card: Automatic Roughness Coefficient Assignment by Depth. See the section "Wetting and Drying", "Elemental Elimination", "Marsh Porosity", and DA and DM Card descriptions in the RMA2 model documentation for more information. This dialog is reached through the *Global Methods* tab of the *RMA2 Model Control* dialog.

Options in the dialog apply to the entire mesh. The following parameters can be set in the *Nodal Transition (Marsh Porosity)* dialog:

- *Minimum Land Elevation* – The options in this section adjust the land elevation to prevent the mesh from going dry. It has the following options:
  - *Distance below each node's bathymetry value* – Will override the land elevation if the distance between the elevation and the nodal bathymetry is less than the set value. Default is 3.0.
  - *Constant elevation, independent of nodal bathymetry* – Set slightly lower than the minimum expected water elevation for the simulation.
- *Other Parameters* – The follow values can be set:
  - *Transition range of distribution* – Default is 2.0.
  - *Minimum wetted surface area factor* – Default is 0.001.

Related Topics

- Marsh Porosity
- RMA2 Model Control dialog

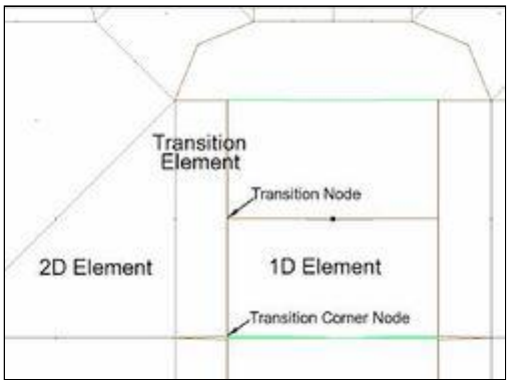
Rainfall Values Dialog

The *RMA2 Rainfall Values* dialog is used to assign rainfall or evaporation. Rainfall or evaporation is applied in inches/hour or cm/hour if using metric units. Positive values represent rainfall, while negative values represent evaporation. Rainfall and evaporation can occur at any time during the simulation. See the "Adding Rainfall And Evaporation" section and RA Card description in the RMA2 model documentation for more information.

Related Topics

- RMA2 Model Control dialog

RMA2 1D Control Structures



In RMA2 , a one-dimensional Control Structure element is a single point which contains two nodes and has an IMAT value ≥ 904. The order of the node numbering at a Control Structure element should be that the side with higher elevation comes first, then the side with the lower elevation. This is generally the “upstream” side of the structure followed by the “downstream” side. This order sets the “orientation” of the flow through the structure.

Since flow through and over some structures does not fit the 2D assumptions, some structures can be represented as 1D components in the 2D mesh. These include weirs, culverts, drop inlets and some gates.

There are strict rules that must be followed in order to get convergence with 1D elements in RMA2 . These rules include:

- Transition elements must be connected to the free end of a 2D element. The element should not be connected to other 2D elements on the sides.
- The transition element must be perpendicular to the free end of the 2D element.
- The bottom elevation of the free end should be flat.
- Control structure elements cannot be constructed on transition elements.

Culvert

A culvert is a conduit used to enclose a flowing body of water. It may be used to allow water to pass underneath a road, railway, or embankment, for example. Culverts can be made of many different materials; steel, polyvinyl chloride (PVC) and concrete are the most common.

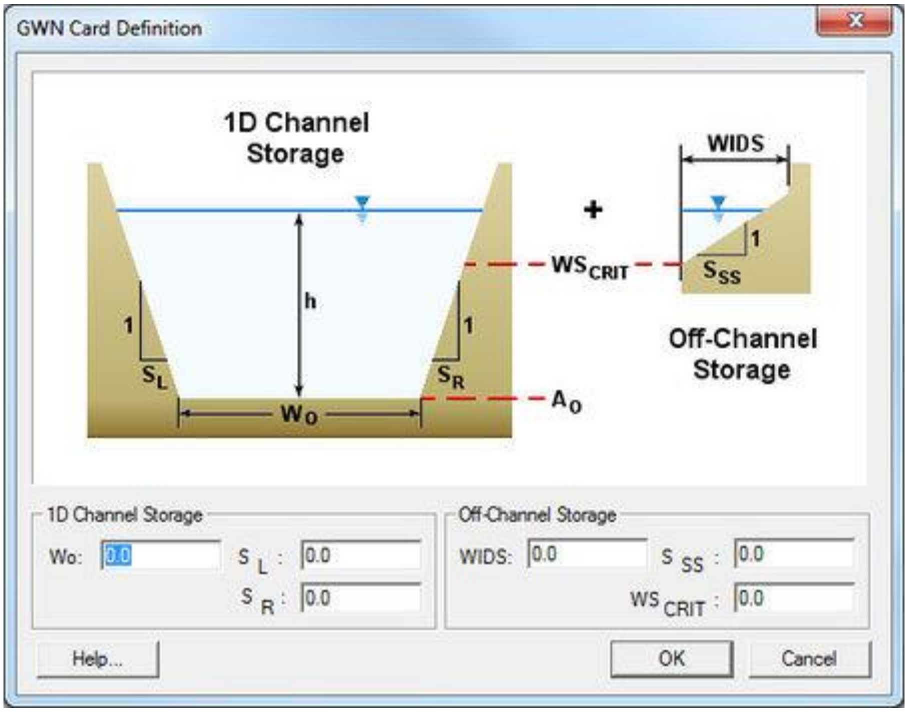
Culvert flow rate depends primarily on headwater depth. Other factors that affect flow rate are tailwater elevation and properties of the culvert such as shape, cross sectional area, inlet geometry, length, slope, and roughness.

Weir

A weir is a small overflow-type dam commonly used to raise the level of a river or stream. Weirs have traditionally been used to create mill ponds in such places. Water flows over the top of a weir, although some weirs have sluice gates which release water at a level below the top of the weir. The crest of an overflow spillway on a large dam is often called a weir.

Weirs also give hydrologists and engineers a simple method of measuring the rate of fluid flow in small to medium sized streams. Since the geometry of the top of the weir is known, and all water flows over the weir, the depth of water flowing over the weir will be an indication of the flow. There are different types of weir. It may be a simple metal plate with a V notch cut into it or it may be a concrete and steel structure across the bed of a river. A v-notch weir will give a more accurate indication of low flow rates.

1D Node Geometry Dialog



Related Topics

- RMA2

RMA2 Boundary Conditions

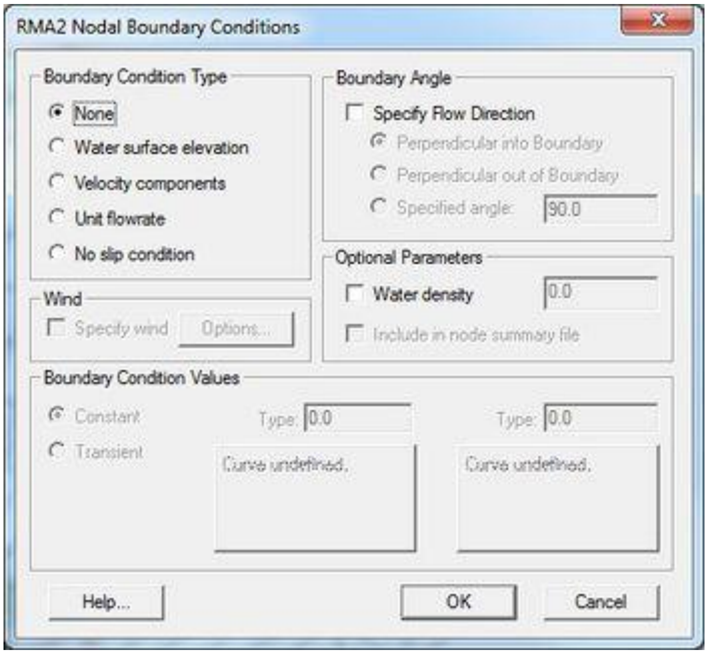
Assigning Boundary Conditions

Boundary conditions are generally defined on nodestrings but may also be defined on nodes. The default boundary condition is a closed boundary (no flow).

Deleting Boundary Conditions

Boundary conditions are removed using the menu command *RMA2* | **Delete BC** .

Nodal Boundary Conditions



Boundary Condition Type

- *None*

  No Boundary Condition is assigned to this Nodestring

- *Water Surface Elevation*

  A Water surface elevation BC is assigned to this Nodestring. This boundary condition is normally located at the tailwater location. However. RMA2 does sometime edit the water depth at the exit boundary to satisfy the finite element equations.

- *Velocity Components*
- *Unit Flowrate*
- *No Slip Condition*

Boundary Angle

- Specify Flow Direction
  - Perpendicular into Boundary
  - Perpendicular out of Boundary
  - Specified angle

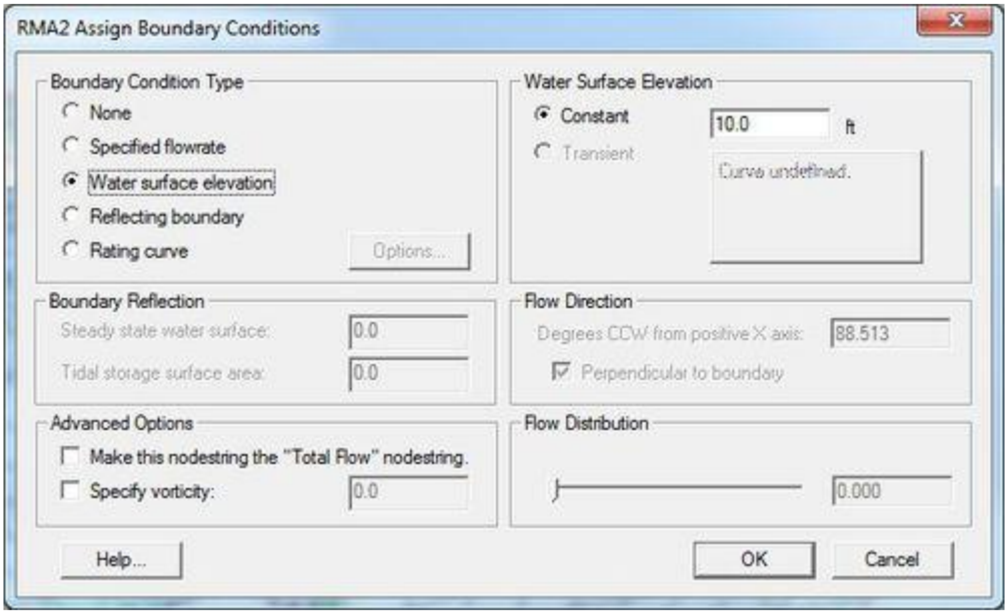
Optional Parameters

- Water Density
- Include in node summary file

Boundary Condition Value

- Constant
- Transient

Nodestring Boundary Conditions



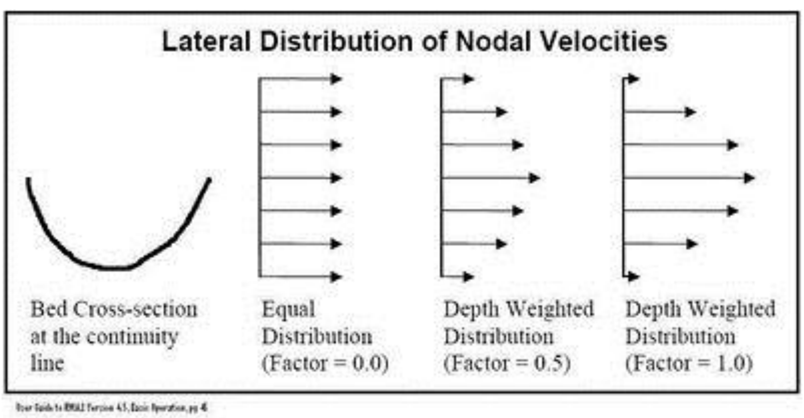
Boundary Condition Type

- *None*

  No Boundary Condition is assigned to this Nodestring

- *Specified Flow Rate*

  A flow rate BC is assigned to this nodestring. It is the most popular way to provide a total flow across a particular continuity line. The total flow (cubic units per second), flow angle (degrees from positive x-axis) and distribution factor can be specified.



- *Water Surface Elevation*

  A Water surface elevation BC is assigned to this Nodestring. This boundary condition is normally located at the tailwater location. (However RMA2 does sometime edit the water depth at the exit boundary to satisfy the finite element equations)

- *Reflecting Boundary*

  Considered an advanced feature – for more information see the "Boundary Permeability (Reflection/Absorption)" section on page 99 of the Users Guide to RMA2 version 4.5

- *Rating Curve*

  A Rating curve can also be defined by selecting the **Options...** button

Flowrate

- Constant
- Transient

Boundary Reflection

- Steady state water surface
- Tidal storage surface area

Flow Direction

- Degree CCW from positive X axis
- Perpendicular to boundary

Advanced Options

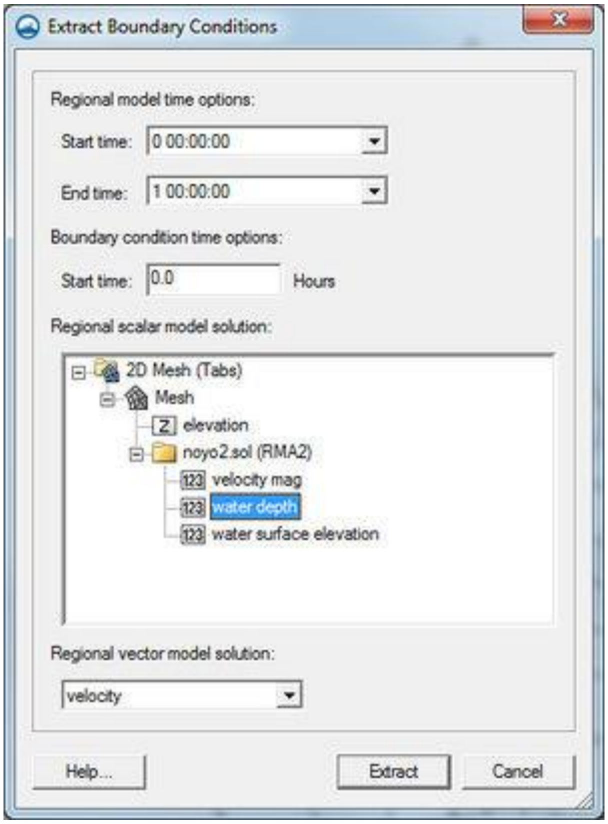
- Make this nodestring the "Total Flow" nodestring
- Specify vorticity

Flow Distribution

A real number between 0.0 and 1.0, that is used to weight the nodal x and y velocity components according to the nodal depths and friction across the continuity line.

Extract Nodal Boundary Conditions

- Boundary Condition Type:
  - None
  - Water surface elevation
  - Velocity
  - Unit-Flow
- Specify Flow Direction
  - Perpendicular info boundary
  - Perpendicular out of boundary
  - Specified Angle
- Extract



- View Extracted

Related Topics

RMA2

RMA2 Files

Input and Output files for RMA2 .

Input Files

- ASCII Geometry (*.geo) → GFGEN → Binary Geometry (*.bin) and printed output file (*.ot1) – Consists of the nodes and elements that define the size, shape, and bathymetry of the study area.
- Run Control Input File (*.bc) – Used to tell RMA2 how to run the simulation.
- Hotstart File (*.hot) – (optional) Used to load critical information for the new simulation.
- Alternate Boundary Condition File (*.abc) – Used in conjunction with the primary run control file.
- Run File (*.run) – Indicates to launch _gr.run (GFGEN) and _rm.run (RMA2).

Output Files

- Binary Solution (*.sol) – Contains the results of computations for all time steps defined in the run control file.
- Printed Output File (*.ot2) – Contains the a full list of results from the simulation run.
- Node summary file (*.sum) – Contains: a table of the steady state solutions of *x* and *y* velocity components, depth, etc., for each node listed on the TRN cards; and a dynamic hydrograph of the previous information for each node listed on TRN cards.
- Hotstart Output (*.hot) – (optional) Used to preserve the critical information at the end of a simulation in order for the run to be restarted and continued at a later time.

Related Topics

- RMA2
- RMA2 Hydraulic Structures
- RMA2 Model Control Dialog
- RMA2 Spindown

RMA2 Graphical Interface

The RMA2 Graphical Interface includes tools to assist with creating, editing, and debugging a RMA2 model. The TABS (RMA2) interface exists in the Mesh Module . SMS can visualize and create RMA2 projects, easily modify project parameters, and view the solutions produced by the RMA2 model. Below are links describing the SMS interface that interacts with the RMA2 model.

Model Control

The *RMA2 Model Control* Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. Boundary conditions in RMA2 include flows in/out of the model domain or known water surface elevations. In RMA2 boundary conditions are generally defined on nodestrings but may also be defined on nodes . The default boundary condition is a closed boundary (no flow). See Nodestring Boundary Conditions and Nodal Boundary Conditions for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

RMA2 Hydraulic Structures

RMA2 includes support for several different types of structures including weirs, culverts, drop inlets, gates and piers. See RMA2 Hydraulic Structures for more information.

Running the Model

The RMA2 Files are written automatically with the SMS project file or can be saved separately using the *File* | **Save RMA2** or *File* | **Save As** menu commands. See RMA2 Files for more information on the files used for the RMA2 run.

RMA2 can be launched from SMS using the *RMA2* | **Run FSTD2H** menu command. A check of some of the common problems called the *Model Checker* is done each time the model is launched, or by selecting the *RMA2* | **Model Check** menu command.

RMA2 Menu

See RMA2 Menu for more information.

Related Topics

- SMS Models Page
- TABS Models
- RMA2

- RMA4
- Total Flow Nodestring

RMA2 Material Properties

For an in depth discussion of the RMA2 material properties, consult the RMA2 Users Guide. [196] [197]

Turbulence

The global turbulence values (specified in the RMA2 model control) can be overridden for each material.

- Standard eddy viscosity method
  - E_{xx} – The x-momentum of turbulent exchange in the x-direction (lb-sec/ft ^{2} or Pascal-sec for SI-units)
  - E_{xy} – The x-momentum of turbulent exchange in the y-direction (lb-sec/ft ^{2} or Pascal-sec for SI-units)
  - E_{yx} – The y-momentum of turbulent exchange in the x-direction (lb-sec/ft ^{2} or Pascal-sec for SI-units)
  - E_{yy} – The y-momentum of turbulent exchange in the y-direction (lb-sec/ft ^{2} or Pascal-sec for SI-units)
  - Note: 1 lb-sec/ft ^{2} = 47.879 Pascal-sec
- Smagorinski method
  - TBFACT – Smagorinsky coefficient for turbulent exchange. A negative value applies the default coefficient (0.05).
  - TBFACTS – Not applied in RMA2. Smagorinsky coefficient for diffusion (Vorticity). A negative value applies the default coefficient (0.05).
  - TBMINF – Smagorinsky minimum turbulent exchange factor. A negative value applies the default coefficient (1.0).
  - TBMINFS – Not applied in RMA2. Smagorinsky minimum diffusion (Vorticity) factor. A negative value applies the default coefficient (1.0).

Roughness

The global roughness value (specified in the *RMA2 model control*) can be overridden for each material. The material roughness can be specified as a single value, or depth dependent.

Marsh porosity

The global marsh porosity values (specified in the *RMA2 model control*) can be overridden for each material. Since wetting and drying cause abrupt changes to the shape of the finite element mesh, numerical stability problems can result when wetting and drying occurs. In an attempt to address the fact that wetting and drying is a continuous process, while the mesh is a discreet data set, marsh porosity was implemented for RMA2. When marsh porosity is used, elements are considered "dry" only if ALL nodes on the element are dry.

Abrupt changes in the marsh porosity parameters between adjacent node can lead to divergence. The results from a simulation using marsh porosity can be difficult to interpret due to the fact that partly dry marsh elements appear completely wet even thought the water surface elevation is lower than part of the element.

Marsh porosity is only available when running a dynamic RMA2 simulation.

Wind

According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, this feature remains experimental."

Wind shear stress at the water surface is caused by friction between the moving air and water. The shear stress coefficient is a function of wind speed. The global wind values (specified in the RMA2 model control) can be overridden for each material.

Wind parameters are only available when running a dynamic RMA2 simulation.

Rainfall

The global rainfall and evaporation values (specified in the *RMA2 model control*) can be overridden for each material.

Coriolis

The global coriolis values (specified in the *RMA2 model control*) can be overridden for each material.

Related Topics

RMA2

RMA2 Menu

The following menu commands are available in the *RMA2* Menu:

Boundary Conditions

- **Assign BC** – Specify boundary conditions for the selected node(s) or nodestring(s)
- **Delete BC** – Deletes the boundary condition assigned to the selected node(s) or nodestring(s)
- **Extract Nodal BC** – Extracts nodal boundary conditions (dynamic simulations only)

Hydraulic Structures

- **1D Node Geometry** – Specify 1D node geometry (1D Control Structures).

Model Parameters

- **Add GC Strings** – Designates the selected nodestrings as Geometry Continuity Check Line nodestrings
- **Material Properties** – Specify turbulence, roughness, marsh porosity, wind, rainfall, and coriolis properties for each material
- **Model Check** – Checks the RMA2 simulation for common input errors
- **Model Control** – Organize inpute files, specify model parameters, choose output options, etc.
- **Revisions** – Create and organize revisions. See below.
- **Run RMA2** – Launches the RMA2 model

RMA2 Revisions

This functionality has been replaced by the RMA2 Spindown option, though the function is still available for use if desired.

RMA2 allows revising both model coefficients and/or boundary conditions for each time step. This is done through the *RMA2* menu selecting the **Revisions** command which will bring up the *RMA2 Revisions* dialog.

Right-click in the *Existing Revisions* field to select the **New Revision** command. This will create a new revision and allow use of the *Selected Revisions* field.

Clicking the **Add** button will create a "Material Properties", "Node BC", or "Nodestring BC" revision. If "Material Properties" is selected, the *RMA2 Material Properties* dialog will appear to set the revision properties. If "Node BC" is selected, the *RMA2 Nodal Boundary Conditions* dialog will appear. If "Nodestring BC" is selected, the *RMA2 Assign Boundary Conditions* dialog will appear.

The "Node BC" option requires that a node with an assigned boundry condition be selected before opening the *RMA2 revisions* dialog. The "Nodestring BC" option requires that a nodestring be selected before opening the *RMA2 revisions* dialog.

Related Topics

- RMA2

RMA2 Model Control Dialog

This article highlights frequently used RMA2 model control parameters. Please see the RMA2 model documentation for a description of items not discussed or for more information on specific items.

RMA2 model parameters are divided into the following groups in the RMA2 Model Control:

General

Simulation Titles / $T1 Card

The RMA2 .geo file contains three comment lines, which can be set using the Title 1, Title 2, and Title 3 edit fields.

Machine Type / $M Card

Set the machine type the RMA2 simulation will be run on. Options include:

- *Microprocessor (PC)* – Default option. Direct access record length is unlimited and is define in terms of bytes.
- *Prime mini-computer* – Direct access record length is unlimited and is defined in terms of small words (i.e. 2 bytes).
- *DEC VAX* – Direct access record length, limited to 32K bytes and defined in terms of long words (4 bytes).
- *Cray or Cyber-205* – Direct access defined for systems using 64 bit or 8 byte words and where record lengths are defined in bytes.
- *HP or Alpha workstations* – Direct access defined using multiple sequential access files that are opened as required.

Water Properties

Set the properties of the water being simulated.

- *Temperature / FT Card* – Average initial water temperature.
- *Density / FD Card* – Used to supply a fluid density for all nodes or a specified node.
- *Coldstart initial water surface / IC Card* – Used to provide initial conditions to start a cold run.
- *1D node initial conditions*
    - *Minimum depth / IC Card* – Minimum depth used for one-dimensional nodes at startup.
    - *Initial velocity / IC Card* – Nominal velocity for one-dimensional nodes.

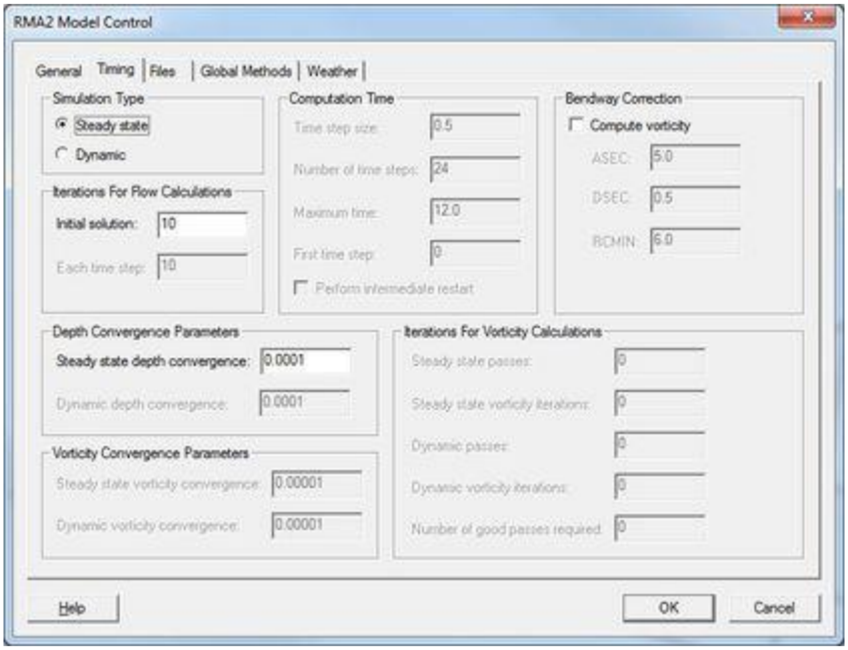
Scale Factors

- *X scale / GS Card* – Scale factor for X-coordinates.
- *Y scale / GS Card* – Scale factor for Y-coordinates.
- *Z scale / GS Card* – Scale factor for Z-coordinates (bottom elevation).

Model Stability

- Special calculations

Timing



Simulation Type

- *Steady state* – Boundary conditions do not change with time. A "snapshot" in time.
- *Dynamic* – Boundary conditions change with time. Computation Time parameters will need to be specified.

Iterations For Flow Calculations

See the TI Card description in the RMA2 model documentation.

- *Initial solution* – Maximum number of iterations to perform for the initial solution.
- *Each time step* – Used for dynamic simulation - maximum number of iterations to perform after the first time step.

Computation Time

See the TZ Card description in the RMA2 model documentation.

- *Time step size* – DELT Variable
- *Number of time steps* – NCYC Variable
- *Maximum time* – TMAX Variable
- *First time step* – NSTART Variable
- *Perform intermediate restart* – MBAND Variable

Bendway Correction

Used to supply values associated with the calculation of vorticity. See the VO Card description in the RMA2 model documentation.

- *Compute vorticity* – IVOR Variable
- *ASEC* – ASEC coefficient for the vorticity equation. Recommend: 5.0
- *DSEC* – DSEC coefficient for the vorticity equation. Recommend: 0.5
- *RCMIN* – Minimum radius of curvature that will be allowed. Recommend: 6 feet (or 2 meters).

See also BV card, TV card.

Depth Convergence Parameters

- Steady state depth convergence
- Dynamic depth convergence

Vorticity Convergence Parameters

See the VO Card description in the RMA2 model documentation.

- Steady state vorticity convergence
- Dynamic vorticity convergence

Iterations for Vorticity Calculations

The maximum number of iterations allowed for the vorticity calculation is set by tvariables NVITI or NVITN on the TV card. If the number of passes between the phases reaches NPASS1 or NPASS2, then the calculation has failed to converge. If the number of “good passes” between the two phases exceeds NGOODMAX, the calculation is finished and the code can proceed to the next time step. See the "Bendway Correction (Vorticity)" section and TV Card description in the RMA2 model documentation.

The following TV Card variables are used to control the number of iterations for vorticity calculations:

- *Steady state passes* – NPASS1 Variable
- *Steady state vorticity iterations* – NVITI Variable
- *Dynamic passes* – NPASS2 Variable
- *Dynamic vorticity iterations* – NVITN Variable
- *Number of good passes required* – NGOODMAX Variable

Files

RMA2 Input Files

- *Specify geometry file* – Select an existing RMA2 ASCII Geometry (*.geo) file to use with the simulation
- *Hotstart input file* – Specify an existing RMA2 Hot Start File (*.hot) to use with the simulation
    - *Use specific time for hotstart*
- *Alternate dynamic BC file* – See the "RMA2 Alternate Dynamic BC File Format" section of the RMA2 model documentation
- *Write dynamic memory file* – See the "Dynamic Memory Allocation" section of the RMA2 model documentation

RMA2 Solution Files

- *Write RMA2 solution file*
    - *Write frequency in time steps*
- *Write hot start file* – Write an RMA2 Hot Start Output (*.hot) file. See RMA2 Spindown for a discussion of hot starting.
    - *Save only the last time step* – Write only the last successful RMA2 time step solution to the hot start file.
    - *Write frequency in time steps* – Write multiple time steps to the hot start file.

Informational Files

- Echo card input to screen
- Write ASCII results file

    - Do not echo any node/element input data
    - Echo all input data except initial conditions
    - Echo all input data
    - Write frequency of nodal results in time steps
    - Trace subroutines level
- Write summary by node file
- Write automatically computed parameters file
- Write vorticity file
    - Write frequency in time steps

Global Methods

Global Wet/Dry Technique

See the Material Properties article for a description of the *Marsh Porosity* parameters.

- Elemental wet/dry check
    - Check frequency in timesteps
    - Dry depth
    - Active depth
- Nodal Transition (Marsh Porosity)
    - Options – Opens the *Nodal Transition (Marsh Porosity)* dialog

Global Roughness Assignment

- Default roughness value
- Roughness by depth
    - Options – Opens the *Roughness Options* dialog

Coriolis Forces

- Latitude

Global Eddy Viscosity Assignment

See the Material Properties article for a description of the *Turbulence* parameters.

- Traditional eddy viscosity approach (default)
- Peclet Number
- Smagorinski Method
    - Exx ratio
    - Eyx ratio
    - TBFACTS
    - TBMINFS
    - Peclet Number
    - Minimum velocity

Weather

Wind

According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, this feature remains experimental."

See the "Simulating With Storms" section and BWC Card description in the RMA2 model documentation for more information.

- Method for wind
  - Do not use wind
  - Original RMA2 Formula
  - Van Dorn Formula
  - Wu Formula
  - Safaie Formula
  - Ekma Formula
  - Generic Formula
- A
- exp
- C
- rho
- Specify global wind
  - Define – Opens the *Wind Values* dialog.

Rainfall/Evaporation / RA Card

See the "Adding Rainfall And Evaporation" section and RA Card description in the RMA2 model documentation [198] for more information.

- Specify rainfall or evaporation
  - Define – Opens the *Rainfall Values* dialog

Related Topics

- RMA2

RMA2 Spindown

For cold start simulations, the initial velocities are zero and the water surface elevation is constant. This is often referred to as the "bathtub condition." Often RMA2 will not directly converge using these initial conditions.

Incremental Loading Strategy

The flow equations are nonlinear and thus require an iterative solution, starting from some initial guessed value. Convergence of the iterative solution is not guaranteed. Since the desired boundary conditions may be vastly different from a cold start condition, it may be impossible to get convergence starting from this bathtub type condition. However, a solution can be obtained using a series of “Runs” that generate solutions progressively closer to the desired answer. Intermediate boundary conditions that are closer to the final desired boundary conditions are specified to generate a set of flow conditions. These conditions do not represent the final desired flow conditions, but are closer to the final desired flow conditions than the original cold start, and can be used as initial conditions for a subsequent run. Starting the model from a previous solution is called “Hot Starting”. In the incremental loading strategy, “loads” consisting of applied flow rates and water surface elevations along the boundary increment from a cold start condition to the final condition. By choosing a suitably small increment in the boundary conditions, convergence can be attained.

RMA2 Spindown refers to the process of using the Steering Module to automate the process of incremental loading. The Steering Module can vary the following:

- Boundary Conditions
    - Water surface elevation
    - Flow rate
- Model Parameters
    - Eddy viscosity
- Finite Element Network
    - Geometry (nodal elevations)

This replaces the use of "REV" cards in RMA2 .

RMA2 Spindown Dialog

The *RMA2 Spindown* dialog updates with the progress of spinning down the model.

The top window explains the spindown convergence of the current run. Each iteration shows as a green point, allowing determination of if the run is converging or diverging (moving toward or away from 0 head change). The iteration being performed is shown just above this plot.

The bottom window shows the overall spindown of the model. The green points represent successful runs, and the red Xs represent failed runs. When this plot reaches 100% spun down, the model is finished. This percent is shown just above this plot.

This process can take several minutes to complete. When the spindown has finished, a window appears advising that the "Steering process has terminated – See status file for details." This status file is named "SteeringStatus.txt" and gives a summary of the steering process. A final solution file will also be created.

Related Topics

- RMA2
- Steering

Roughness Options Dialog

The *RMA2 Roughness Options* dialog is used to define the RD Card: Automatic Roughness Coefficient Assignment by Depth. See the RD Card description in the RMA2 model documentation for more information. This dialog is reached through the *Global Methods* tab of the *RMA2 Model Control* dialog.

The following parameters can be set in the *Roughness Options* dialog:

- *Manning's* n *, no vegetation* – Maximum roughness value for non-vegetated water. A positive value will override the default.
- *Depth for no vegetation* – Depth at which vegetation effects roughness. A positive value will override the default.

- *Manning's* n *, with vegetation* – Roughness value for vegetated water. A positive value will override the default.
- *Roughness coefficient* – Roughness by depth coefficient. A positive value will override the default.
- *Minimum Manning's n* – Minimum allowed roughness. Default is 0.01.
- *Maximum Manning's n* – Maximum allowed roughness. Default is 0.20.
- **Set Defaults** – Opens the *Roughness By Depth Defaults* dialog. Defaults from several Army Corps projects can be automatically entered into the *Roughness Options* dialog.

The Roughness By Depth Defaults dialog has the following options:

- "Mississippi River Delta Defaults"
- "S-shaped River Example Defaults"
- "San Francisco Bay Estuary Defaults"

A graph of the Depth vs. Roughness is calculated using the given parameters and shown in the *Roughness Options* dialog.

Related Topics

- [RMA2 Model Control dialog]
- [RMA2]

RMA2 GCL Card

The GCL card is used to specifiy a line within the grid where the flow rate is of interest. GCL lines may be used to specify the location of boundary conditions.

Example GCL Card

The GCL Card permits the line # to be assigned, and the end of the line is marked with a negative number.

```
CO Line#   Corner Node numbers... Negative marks end of line.
GCL    1   10   11   12   13   14   15
GCL        16   17   18   19   20   -1
GCL    2   100  101   102  109  107  99  -1
```

Related Topics

- [RMA2 Menu]
- [RMA2]

6.7.c. RMA4 – Resource Management Associates

RMA4

RMA4	
Model Info	
Model type	Finite element water quality transport numerical model in which the depth concentration distribution is assumed uniform. It computes concentrations for up to 6 constituents, either conservative or non-conservative, within the computational mesh domain.
Developer	Resource Management Associates United States Army Corps of Engineers (USACE)
Web site	RMA4 web site
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • RMA4

RMA4 is a finite element water quality transport numerical model. RMA4 was designed to simulate the depth-average advection / diffusion process in an aquatic environment. The model can be used for the evaluation of any conservative substance that is either dissolved in the water or may be assumed to be neutrally buoyant within the water column. The model is also used for investigating the physical processes of migration and mixing of a soluble substance in reservoirs, rivers, bays, estuaries and coastal zones. The model is useful for evaluation of the basic processes or for defining the effectiveness of remedial measures. For most applications, the model utilizes the depth-averaged hydrodynamics from RMA2 . The input hydrodynamics solution can be either a steady state or a dynamic simulation and is used to compute particle dispersions.

The RMA4 model can be added to a paid edition of SMS.

Functionality

RMA4 has been applied to:

- Define horizontal salinity distribution.
- Trace temperature effects from power plants
- Calculate residence times of harbors or basins
- Optimize the placement of outfalls
- Identify critical area for oil spills or other pollutants spread
- Monitor water quality criterion within game and fish habitats
- Determine the limits of salinity intrusion
- Perform flushing analysis

- Turbidity monitoring
- SMS does not support BOD/DO modeling with RMA4 because of issues with RMA4.

Using the Model / Practical Notes

- RMA4 includes support of the SI card which controls what units the engine will use. If SI is set to 0, the model runs with English units. If it is set to 1, the model runs in metric units. SMS saves the SI card based on the coordinate system (projection) specified. However, it is recommended that metric units be used for RMA4. This applies to the hydrodynamics that are fed into RMA4 as well.
- RMA4 is more sensitive to mass gain/loss than RMA2 across large closed boundary angles. The suggested minimum boundary angle between any two adjacent elements is 10°. This becomes a very great concern, as this applies simply to the wet/dry interface and not to the outer mesh boundary.
- RMA4 should not be used to evaluate highly volatile materials such as gasoline, nor materials that do not mix with water, such as oil. Although RMA4 has been used to measure temperature effects from such locations as power plant discharges, this is not a recommended application. Although the documentation claims to be able to model up to 6 constituents, the SMS developers have never had success with more than a single constituent at a time. Multiple constituents require multiple boundary condition files, each with values for one of the constituents. Multiple constituents can be named and joined through the Data Calculator and exported to a single file for future use.
- Card order matters. The RMA4 run control and full print files are very helpful for determining what options are being used.
- The TABS models are built to expire after a set date. The latest version can be downloaded from the Software Updates section of the Aquaveo website .

Graphical Interface

SMS provides a graphical interface that is designed to visualize the projects being created, easily modify project parameters, and view the solutions produced by the RMA4 model. See RMA4 Graphical Interface for more information.

Saving RMA4

When completing the *File* | **Save As...** command, the following files get saved in the *.sms:

- *.mat referenced to new save location
- *.mat referenced to new save location
- *.qsl referenced to new save location (RMA4 solution file)
- *.sol referenced to new save location and original location (RMA2 solution file)

RMA4 Files

Input and Output files for RMA4 .

Input Files

- Hydrodynamic Solution (usually an RMA2 *.sol file with a path name less than 67 characters long)
- RMA4 Run Control (Boundary Condition) input (*.trn)
- GFGEN Output Geometry (*.bin)

Output Files

- Transport Solution (*.qsl)
- Full Print Listing File (*.ot3)

Related Topics

- SMS Models Page
- TABS Models
- RMA2

RMA4 Boundary Conditions

RMA4 has a number of tools for working with boundary conditions.

Assigning Boundary Conditions

Boundary conditions are generally defined on nodestrings but may also be defined on elements. The default RMA4 boundary condition is a closed boundary (no constituent). To assign boundary conditions, select the item (nodestring, element, etc.) to which the boundary condition should be applied and use the menu command *RMA4* | **Assign Boundary Conditon** which will open a dialog allowing the specification of boundary condition options.

Element Loading

Starting in SMS 10.1, it is now possible to assign a mass loading to an element.

To do this, select the element then go to *RMA4* | **Assign BC** . This will bring up the *RMA4 Assign Element Mass Loading* dialog.

Deleting Boundary Conditions

Boundary conditions are removed using the menu command *RMA4* | **Delete Boundary Conditon** .

Nodestring Boundary Conditions

Nodestrings can be used as boundary conditions to define constituent concentrations being modeled. Constituent concentrations can be defined as constant or transient using the *XY Series Editor* .

RMA4 does not care about the units of the concentration because the output is relative to the initial number specified. For example, if we specify a concentration of 1,000, the values in the solution will range from 0 to 1,000 as the plume spreads downstream. We can say that the concentration is ppm, ppt, or kg/kg; RMA4 treats all concentrations as relative values.

Since a concentration in water is rarely rigidly maintained, a shock factor may be applied to allow fluctuation of the concentration when the flow direction changes. If no shock factor is applied, no matter how much the flow pushes the concentration out of the model, the concentration at the boundary will not change. However, applying a shock factor is like creating a buffer zone outside the model where the constituent can go until the flow begins to carry it back into the model. This provides for a more realistic solution in some cases. Depending on the situation, a different shock factor may be applied from zero for no shock to 1.0 for a gradual change due to a change in flow direction.

By right-clicking on a selected nodestring and selecting *Assign BC* | **RMA4...** will bring up the *RMA4 Assign Boundary Conditions* dialog. The dialog has the following options:

- *Constituent*
- *Constant*
- *Transient*
  - **Curve** button – Brings up the *XY Series Editor* .
- *No factor applied when flow direction changes*
- *Factor applied when flow direction changes*
- *Rigby maintained boundary*

Element Boundary Conditions

A boundary mass loading can be added to an element in the mesh by selecting first the desired element and then the *RMA4* | **Assign BC** menu command. This opens the *RMA4 Assign Element Mass Loading* dialog where a constant or transient entry can be made in grams/second for each constituent defined in the *RMA4 model control* dialog . The time variant option uses a XY Series File, XYS Format editor to enter a series of time step-mass loading pairs. The time steps should be consistent with those of the RMA4 simulation and for each constituent. The mass loadings may be deleted using the *RMA4* | **Delete BC** menu command.

Card Type	**RMA4 BLE Card: Boundary Loading (Mass)**		
Description	Used to assign mass loading on the element boundary.		
Required	NO		
Format	BLE		
Sample	BLE 237 2 3		
Field	**Variable**	**Value**	**Description**
0	IC1	BLE	Card type identifier.
1	ISTART	+	Element id
2	PBCX(J,1)	+	Mass loading for constituent 1 at ISTART=J (g/s)
3	PBCX(J,2)	+	Mass loading for constituent 2 at ISTART=J (g/s)
4	PBCX(J,3)	+	Mass loading for constituent 3 at ISTART=J (g/s)
5	PBCX(J,4)	+	Mass loading for constituent 4 at ISTART=J (g/s)
6	PBCX(J,5)	+	Mass loading for constituent 5 at ISTART=J (g/s)
7	PBCX(J,6)	+	Mass loading for constituent 6 at ISTART=J (g/s)
Card Type	**END**		
Description	Identifies the end of the time step		
Required	NO		

Related Topics

- RMA4
- RMA2 Boundary Conditions

RMA4 Graphical Interface

The RMA4 Graphical Interface includes tools to assist with creating, editing, and debugging a RMA4 model. The TABS (RMA4) interface exists in the Mesh Module . SMS can visualize and create RMA4 projects, easily modify project parameters, and view the solutions produced by the RMA4 model.

The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to RMA4. If a mesh has already been created for a RMA4 simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to RMA4. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the *RMA4* menu.

RMA4 Menu

The following menu commands are available in the *RMA4* Menu:

Boundary Conditions

Two commands are available for selected boundary nodes and nodestrings.

Assign BC

Specify boundary conditions for the selected nodestring(s) or element(s).

Delete BC

Removes the boundary condition assigned to the selected nodestring(s) or element(s).

Model Parameters

These commands are available for setting parameters and running the simulation.

Delete Simulation

Deletes the current RMA4 simulation. Sets all values in the *Model Control* to default values.

Material Properties

Brings up the *RMA4 Material Properties* dialog which specifies turbulence, roughness, marsh porosity, wind, rainfall, and coriolis properties for each material.

Model Control

Brings ups the *RMA4 Model Control* dialog which can organize inpute files, specify model parameters, choose output options, etc.

Run RMA4

Launches the *RMA4* model.

Related Topics

- TABS Models
- GFGEN
- RMA2
- RMA4

RMA4 Material Properties

The *RMA4 Material Properties* dialog is reached through the *RMA4* | **Material Properties...** menu command in the Mesh module.

The dialog has the following options:

- Material list – a list of available materials is populated on the left side of the dialog. Selecting a material allows the diffusion to be changed for the material.
- *Overide global specification* – When turned on the global diffusion will not be used for the material.
- *Diffusion coefficient method* – Specifies using diffusion coefficients to approximate turbulence.
- *Peclet number method* – Specifies the use of a Péclet number to determine diffusion.
- *Dx* – Specifies diffusion in the x direction.
- *Dy* – Specifies diffusion in the y direction.
- *Peclet Number* – Enter the Péclet number value.
- *Minimum Velocity* – The lowest velocity allowed for diffusion.

- **General Material Properties** – Brings up the *Materials Data* dialog.

Diffusion

Because RMA4 does not have the ability to model turbulence, diffusion coefficients may be used to approximate turbulence. By assigning a diffusion coefficient in the x and y directions for each material, the flow over that material will be altered somewhat to provide an approximation of turbulent flow over that region. A value of -1.0 may be applied to allow normal flow over the material. Positive values provide turbulence. The higher the value, the greater the effect is.

Related Topics

- RMA4

RMA4 Model Control

The *Model Control Dialog* is used to define general, file, and constituents characteristics. The General tab includes options to set up the period of time the model will run, and hydrodynamic and geometry scale factors. The *File* tab includes the time step used from the RMA2 velocity file and the time subtracted from the RMA2 velocity file, RMA4 *Output files* options, and *Informational Files* to be produced when running the model. The *Constituents* tab includes diffusion coefficient for the constituent, constituent control, and mass check.

General

Simulation Titles. The RMA4 *.geo file contains three comment lines, which can be set using the Title 1, Title 2, and Title 3 edit fields.

Time control

- Start time
- Time Step
- Total Steps
- Max. Time

Model Stability

- Special Calculations

Hydrodynamic Scale Factors

- X-Velocity scale
- Y-Velocity scale
- Depth scale

Geometry Scale Factors

- X scale
- Y scale

Files

RMA4 Input Files – Time steps from the RMA2 solution file that will be used for the RMA4 simulation.

- Last time step used from the RMA2 velocity file
- Time subtracted from the RMA2 velocity file
- Hotstart Input file

- Alternate Dynamic BC File

RMA4 Output files – Output files when running RMA4

- Write Hotstart File
- Write RMA4 Solution File
    - Write specific time range

Information Files – Files to create when running RMA4

- Echo card input to screen
- Write ASCII results file
    - Suppress initial conditions and detailed geometry
    - All input data
    - Suppress detailed geometry
    - Save nodal results every __ time step
    - Subroutine trace
    - Debug trace at level
- Write summary by node file
- Write automatically computed parameters file
- Activate full report

Constituents

- *Diffusion Coefficient* – Diffusion Coefficient Method and Peclet Number Method.
- *Constituent Control* – Number of constituents, initial concentration for each of the constituents, and decay coefficient for each coefficient.
- *Mass Check* – Mass conservation check.

Related Topics

- RMA4

6.8. TUFLOW – Two-dimensional Unsteady FLOW

TUFLOW

TUFLOW	
Model Info	
Model type	One-dimensional (1D) and two-dimensional (2D) flood and tide simulation software. It simulates the complex hydrodynamics of floods and tides using the full 1D St Venant equations and the full 2D free-surface shallow water equations.
Developer	Bill Syme BMT WBM (Australia)
Web site	TUFLOW web site

Tutorials	Models Section • TUFLOW 2D • TUFLOW 1D

TUFLOW (which stands for Two-dimensional Unsteady FLOW) is a one-dimensional (1D) and two-dimensional (2D) flood and tide simulation software. It simulates the complex hydrodynamics of floods and tides using the full 1D St Venant equations and the full 2D free-surface shallow water equations.

The TUFLOW model can be added to a paid edition of SMS.

TUFLOW Numeric Engine

TUFLOW is a computational engine that provides two-dimensional (2D) and one-dimensional (1D) solutions of the free-surface flow equations to simulate flood and tidal wave propagation. The engine is very stable making it an excellent choice for models with lots of wetting and drying. It is specifically beneficial where the hydrodynamic behaviour in coastal waters, estuaries, rivers, floodplains and urban drainage environments have complex 2D flow patterns that would be awkward to represent using traditional 1D network models.

A powerful feature of TUFLOW is its 2D/1D dynamic linking, first pioneered in1990, and subsequently enhanced to the point where it offers unparalleled flexibility and robustness.

TUFLOW continues to develop and evolve to meet the challenges of hydrodynamic modeling. Its strengths include:

- Rapid and stable wetting and drying;
- 1D and 2D linking;
- Multiple 2D domains (optional);
- Both 1D and 2D representation of hydraulic structures;
- Automatic upstream/downstream controlled flow regime switching;
- 1D and 2D supercritical flow;
- Highly flexible and efficient data handling;
- GIS based; and
- Extensive quality control outputs.

It is suited to modeling flooding in major rivers through to complex overland and piped urban flows, along with estuarine and coastal hydraulics. [199]

TUFLOW was written and is maintained by Bill Syme at WBM TUFLOW Webpage .

SMS Interface

The TUFLOW interface in SMS can be used to construct TUFLOW models and view and analyze the results.

Some of the features of the interface include:

- Create 2D domains from bathymetry data
- Extract 1D cross-section domains from TIN data
- Define 1D pipe domains including links to the surface
- Import data from ArcGIS or MapInfo GIS formats
- Define boundary conditions at 1D nodes, 2D lines, or 2D polygons
- Create 2D geometry modifications to model objects such as levees
- Set material properties such as Manning n values and hydrologic losses
- Define simulations from building blocks above

- Simulations share common data to prevent data duplication errors and make it easy to update project with new data

The interface consists of the Cartesian grid tools, the TUFLOW menu and the TUFLOW coverages .

Using the Model / Practical Notes

For the past several years, TUFLOW required the horizontal and vertical coordinates used for input files to be in meters. Starting with the release of TUFLOW 2012, the numeric engine now supports an option for customary units. It's recommended to set the units to be the desired units at the beginning of a project. SMS will allow switching units, but since this feature is so new, be certain to verify model parameters and boundary conditions are in the correct units. Geographic Coordinates cannot be used since it is a latitude/longitude system defined in decimal degrees.

How to use TUFLOW?

The SMS tutorials are a good place to start learning to use SMS and associated models. The following tutorials will help with getting started using TUFLOW.

- Models – TUFLOW 2D
- Models – TUFLOW 1D
- General – Data Visualization
- General – Observations

General Steps to Build a TUFLOW Model

1) Define the domain or domains
2) Setup boundary conditions
3) Create the simulation
4) Define material sets
5) Set Model Parameters
6) Run TUFLOW
7) Find problems and verify model setup using check files
8) View and analyze the results
9) Repeat above steps to generate new simulations

Optional Steps

- Define flow constrictions
- Define geometry modifications such as levees

Saving TUFLOW

When completing the *File* | **Save As...** menu command, the following files get saved in the *.sms:

- *.mat referenced to new save location
- _grds.h5 referenced to new save location
- *.mat_h5 referenced to new save location
- Geomcomps.h5 referenced to new save location
- scatter.h5 referenced to new save location
- tuflow files don't get saved to new location unless exported again

Running TUFLOW

Before running TUFLOW, the TUFLOW files must be generated from SMS. These files are separate from the files SMS uses to store its TUFLOW information.

The options to export the TUFLOW files and run TUFLOW are in the right-click menu of the simulation.

- **Export TUFLOW files** – This will export the TUFLOW model files into the run directory (by default a directory named TUFLOW under the project directory).
- **Launch TUFLOW** – This will launch TUFLOW under the assumption that the files have already been exported as above.
- **Save Project, Export TUFLOW files, and Launch TUFLOW** – This option will save the current SMS project file, export the TUFLOW files, and launch TUFLOW on these files.

Viewing Results

TUFLOW results are written in SMS mesh format in the \TUFLOW\results folder. The mesh module tools and menus are used to visualize the solution.

The solution files all start with the name of the simulation. The extension identifies the type of data in the result file.

- *.2dm – Two-dimensional output mesh in SMS format.
- *.dat – The dataset files associated with the mesh solution (d for depth, h for water surface, etc.)
- *.mat – The output materials from TUFLOW. Most of the element materials come from the *.tmf file. TUFLOW also uses the materials to identify boundary cells.
- *.ALL.sup – A file that opens the *.2dm file, all the *.dat files, and the *.mat file.
- *.hV.sup – A file that opens the *.2dm file, the water surface and velocity datasets, and the *.mat file.

External Links

- TUFLOW 2016 Manual
- TUFLOW Wiki
- Older TUFLOW 2010 Manual (.docx)
- Older TUFLOW Documentation Downloads Page

Related Topics

- TUFLOW Coverages

TUFLOW Simulation

A simulation includes the domain, boundary conditions, model parameters, event definition, and material set used for a single TUFLOW run. Multiple simulations can exist within the same project in order to compare alternatives such as before/after a levee or to look at multiple events such as 10 year, 20 year, and 50 year.

Working with Simulations

A simulation can be created by right-clicking on a blank portion of the project explorer and choosing *New Simulation* | **TUFLOW** . Additional TUFLOW simulations can be created by right-clicking on the Simulations folder and selecting the **New Simulation** command. This command will automatically create a new TUFLOW simulation.

Once a simulation exists future simulations are often similar to one already created. Often it is easier to make a copy of the simulation and make changes where appropriate. A simulation can be copied by right-clicking on the simulation and choosing **Duplicate** .

The right-click menu for a simulation includes commands to change model parameters or 1D model parameters as well as launching TUFLOW to run the simulation.

Multiple simulations can be run at once by right-clicking on the Simulation folder and selecting the **Batch Run** command.

Simulation Components

Much of the data for a simulation is contained in geometry components and TUFLOW coverages . These items are included in a TUFLOW simulation using Project Explorer links . The model parameters are stored with the simulation item and can be accessed from the simulation's right-click menu.

Simulation Links

A TUFLOW simulation is comprised of a grid, feature coverages, and model parameters. SMS allows for the creation of multiple simulations and each includes “links” to these items. Links are like shortcuts in windows and they call for the geometry component as well as each coverage used that is not part of the geometry component.

By using links the data is not duplicated; it just knows where to go to get the data that has to be used for each simulation. The use of links allows these items to be shared between multiple simulations. Multiple simulations can be created if some components need to be changed to create different scenarios and links would call for the same components used in previous simulations. Simulations also store the model parameters used by TUFLOW.

Links are located in the Project Explorer under TUFLOW Simulation once a simulation has been created.

Batch Runs

SMS allows multiple TUFLOW simulations (each with their own defined scenarios/events to run) to run simultaneously. A batch run is initiated by right-clicking on the TUFLOW simulations folder (the folder that holds the individual simulations) and choosing **Batch Run** .

The **Batch Run** command brings up the *Batch Run* dialog to choose the simulations to run. After checking the simulations to run, choose whether to **Export and Run** the simulations or **Run only** . The run only option can be used if the simulation files have been exported previously and don’t need to be changed.

After the simulations have been selected in the *Batch Run* dialog, the runs will be launched and can be monitored in the *Run TUFLOW* model wrapper dialog. The top part of this dialog has progress bars showing the individual progress of each run. The screen output for each run can be viewed by clicking on the associated progress bar. The screen output is shown in the bottom part of the dialog.

Related Topics

- TUFLOW
- TUFLOW Menu

Define TUFLOW domain

TUFLOW supports 2D domains, 1D cross-section based domains, and 1D pipe network domains. Multiple domains of each type can be used within the same TUFLOW simulation. To use Multiple 2D domains in TUFLOW, purchase the add-on module.

Types of Domains

2D Domains

Two-dimensional domains are Cartesian grid (rectilinear) and include elevation data.

TUFLOW 2D domains are cartesian (rectilinear) grids. 2D domains are created using a Cartesian Grid Frame inside a TUFLOW Grid Extents Coverage .

Grids store elevation values at each corner, midside, and center. There are several Grid Options that apply to a grid that can be specified by right-clicking on the grid and choosing *Properties* .

A 2D domain is represented as a 2D Geometry Component in the TUFLOW interface. A 2D Geometry component includes the grid and associated coverages.

Grid elevations can be modified using the select grid location tool by selecting and changing the z values at specific directions. A TUFLOW 2D Z Lines/Polygons (Simple) coverage can also be used to force elevations using arcs or polygons.

Multiple grids can be used inside the same simulation. The grid used in each area of the domain is specified using a TUFLOW 2D/2D Link Coverage .

1D Cross Section Domains

Cross Section based domains include channel and cross-section definitions.

1D cross aection domains use both the TUFLOW network coverage and the TUFLOW cross section coverage . The network coverage describes the channel attributes such as the length and the cross sections are defined in the cross section coverage.

1D Pipe Network Domains

Pipe network domains can be used to model such things as culverts and storm sewers.

1D Pipe networks are created using culvert channels inside a 1D Network Coverage . The shape of the pipe segments are specified using the attributes button in the channel attributes dialog.

Channel inlets, storage, and invert elevations can be defined on the arc nodes (endpoints).

Starting in version 10.1, it is possible to create custom inlet definitions and use them within the pipe network model. See TUFLOW_Inlet_Database

Combining Multiple Domains

1D to 2D Domains

A very powerful and useful feature in TUFLOW is the ability to combine a 1D cross-section domain and a 2D domain. This method can be used to embed a 2D domain between two 1D domains, embedding pipe networks, or to model the channel in 1D and floodplain in 2D. See Combining 1D and 2D Domains .

2D Domain to 2D Domain

One of the disadvantages to TUFLOW is that it uses a fixed grid rather than a mesh which can transition very well from large to small cells. This can be overcome somewhat by using 1D and 2D domains. TUFLOW also supports using multiple 2D domains of different resolutions. The SMS interface makes it easy to set up see Linking 2D Domains .

Related Topics

- TUFLOW

TUFLOW Coverages

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

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

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

Grid Extents Coverage

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

Boundary Conditions Coverage

Boundary conditions are defined in TUFLOW with points, arcs, or polygons in a 1D–2D BCs and Links coverage. BC coverages can also be used to specify cell code (active/inactive) areas of the 2D model domain. The kinds of boundary conditions available depend upon whether it is applied to a point, arc or polygon.

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

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

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

2D Materials Coverage

On this coverage polygons can be created to define material areas. This is the same as the Area Properties Coverage .

1D/2D Connections Coverage

1D–2D connections are used with the 2D boundary condition coverage to link 2D and 1D domains .

Two types of arcs can be created in a 1D–2D Connection coverage: 1D/2D connection arcs (CN) and Flow vs. Head Connection arcs (SC). The type of arc is assigned by selecting the desired arc and using the *Feature Objects* | **Attributes** command to bring up the *Arc Attributes* dialog for this coverage. Designate the arc *Connection type* in the dialog. There is also an option to apply *Weighting* to 1D–2D connection arcs. Weighting sets the proportion to be applied in distributing the water level from the 1D node to the 2D cell.

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

The coverage has a couple coverage specific commands that appear when right-clicking on the 1D–2D Connections coverage. The commands are:

Properties

Brings ups a *Select Boundary Condition Coverage* dialog where a boundary condition coverage can be assigned to the connection coverage.

Clean Connections

Opens a *Clean Options* dialog.

TUFLOW Clean Options

This dialog makes sure that connections arcs end at HX boundaries and the HX boundaries have nodes at the connection endpoints. The dialog has the following options:

- *Tolerance* – Determines the tolerance level SMS will use in determining if the connection arcs end at the HX boundaries.
- *2D BC coverage* – Sets the boundary condition coverage that will provide the HX boundary. The coverage is selected in the tree below.
- *1D Network coverage* – Sets the network coverage to be used. The coverage is selected in the tree below.

1D Cross Section Coverage

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

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

This coverage has a few unique commands available when right-clicking on the coverage in the Project Explorer. The commands are:

Extract from Scatter

Extracts the elevations for the arc from a scatter set (TIN) after cross section arcs are created. This will extract the cross section data from the active dataset on the active scatter set. Values are extracted at each triangle edge in the scatter set and each node or vertex on the cross section arc.

Map Materials from Area Property Coverage

Maps the materials from an area property coverage to the cross-sections after cross sections have been created.

Add Arcs to Mesh

If there is an existing mesh, adds the cross section arcs to the active mesh.

Properties

Opens the *CsDb Management* dialog. All the cross sections in the coverage are stored in a cross section database. (see Editing Cross Sections).

Trim to Code Polygon

Trims the cross sections to the code polygons in the boundary condition coverage. This is done automatically if there is only one boundary condition coverage. If working with a project that uses more than one such coverage, a *Select Coverage* dialog will appear to allow choosing a coverage.

Refresh Cross Section Database Materials List

If there have been changes to the material list in the cross section database, this command with update the coverage to match.

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

1D Network Coverage

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

There are a variety of channel types including open channels, weirs, and culverts (pipe networks). The channel also has a variety of attributes depending upon the channel type. The channel type and attributes are defined in the *Channel Attributes* dialog which can be reached by selecting an arc and using the *Feature Objects* | **Attributes** command.

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

The TUFLOW documentation sections 5.12.4: Connecting Pits and Nodes to 2D Domains and 5.4: 1d_nwk Attributes list some new TUFLOW features that SMS now supports. SMS's *Network Node Attributes* dialog has several new additions to the *Create Connection to 2D Domain (SX)* section. These new options allow controlling elevations at the connections, how many cells are connected, and the method for selection of additional cells (Grade or Sag). Each option correlates fairly directly to a TUFLOW field and some are labeled as such to make look-up easy.

The coverage also has a few unique commands accessed by right-clicking on the coverage in the Project Explorer. These are:

Add Arcs to Mesh

If there is an existing mesh, adds the cross section arcs to the active mesh.

Create Cross Section Arcs

Brings up the *Create Cross Sections Arcs* dialog where cross sections arcs can be automatically generated in the coverage.

Create Water Level Arcs

Brings up the *Create Water Level Arcs* dialog where water level arcs can be automatically generated in the coverage.

2D/2D Linkages Coverage

Any number of grids of varying sizes and/or orientations may be used in TUFLOW. A 2D/2D Linkages coverage is used to setup TUFLOW to use multiple 2D domains.

Polygons are created on this coverage that enclose each domain. Then each polygon can be assigned to a domain by using the *Feature Objects* | **Attributes** command to bring up a *Select TUFLOW Grid* dialog.

See TUFLOW Linking 2D Domains for more information.

2D Flow Constriction Shapes Coverage

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

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

For more information on using this coverage, see TUFLOW Flow Constriction Shapes .

2D Miscellaneous Coverage

Spatially varied attributes can be defined using the 2D Miscellaneous (FLC, WRF, IWL, SRF and AD) coverage. When creating this coverage, the coverage type needs to be set in the *Coverage Properties* dialog. This dialog can be reached by right-clicking on the coverage in the Project Explorer and selecting the **Properties** command.

The available spatial attributes are:

- Initial Water Level (IWL) – Initial water levels at the start of the model simulation. Assigning spatially varied initial water levels may be necessary to ensure that water bodies have water in them initially.
- Weir Factor (WRF) – Allows calibration or adjustment where the broad-crested weir equation is applied. A value of zero removes automatic weir calculations.
- Form Loss Coefficient (FLC) – Applies the form loss attribute values to all cells within each region. Note that FLC values will need to be changed if the 2D cell size changes.
- Storage Reduction Factor (SRF) – Reduces the area in the cell available for water.
- AD Initial Condition (AD) – Defines the spatial distribution of initial conditions for a given constituent. (See TUFLOW AD)
- AD Minimum Dispersion (AD) – Defines the spatial distribution of minimum dispersion coefficients for a given constituent. (See TUFLOW AD)

The values for the attributes are assigned to polygons created in the coverage. TUFLOW assigns the values from the polygons to the cells that exist within the polygon. Selecting a polygon and using the *Feature Objects* | **Attributes** command will being up a dialog with the value can be assigned. The name of the dialog will match the coverage property type.

2D Z Lines (Advanced) Coverage

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

- Make sure the coverage is in the Geometry Component(s) that is to be applied to the modification.

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

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

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

See TUFLOW ZShape for more information.

2D Z Lines/Polygons (Simple) Coverage

These coverages are used as geometry modifications and force grid elevation values using arcs or polygons. This can be useful to ensure ridge or gully lines are represented in the model, simulate a proposed levy, or to simulate a proposed fill/excavation scenario.

It is possible to only have the elevations assigned from a feature object if the feature elevation is higher or lower (user specified) than the existing elevations in the grid. This is controlled in the *2D Z Lines/Polygons (Simple) Coverage Properties* dialog (right-click on the coverage and choose **Properties**). The options specified in the coverage properties dialog apply to any geometry modification arcs and polygons within the coverage. If wanting to use multiple settings within a simulation, there must be multiple 2D z line/polygon (simple) coverages.

Z Values

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

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

4) Add – The z value becomes the existing elevation plus the z value from the arc/polygon (which may be negative).

Thick Arcs

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

Make sure the coverage is in the Geometry Component(s) that is to be applied to the modification.

Points

Points are ignored in the geometry modification coverage.

Arcs

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

Polygons

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

Water Level Lines Coverage

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

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

Right-clicking on the coverage in the Project Explorer has the following option:

Trim to Code Polygon

Trims the water lines to the code polygons in the boundary condition coverage. This is done automatically if there is only one boundary condition coverage. If working with a project that uses more than one such coverage, a *Select Coverage* dialog will appear to allow choosing a coverage.

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

Water Level Points Coverage

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

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

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

Creating the Water Level Points Coverage

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

Related Topics

- TUFLOW model

TUFLOW Menu

TUFLOW Simulation Menu

The *TUFLOW Simulation* menu is accessed by right-clicking on a TUFLOW simulation item (Sim) in the Project Explorer. The following menu commands are available:

Delete

Removes the TUFLOW simulation.

Duplicate

Creates another copy of the simulation.

Rename

Changes the name of the simulation.

Export TUFLOW files

SMS will save the TUFLOW input files in the TUFLOW directory for that simulation. If one does not already exist, a prompt will ask to select a location.

Launch TUFLOW

Launches TUFLOW using input files that have previously been exported.

Save Project, Export and Launch TUFLOW

Saves the project, writes out the TUFLOW input files, and launches TUFLOW .

2D Model Control

Brings up the *2D Model Control* dialog where various model parameters are set. See TUFLOW 2D Model Control page.

1D Control

Brings up the *Control 1D* dialog where various model parameters are set. See Model Parameters page.

Model Check

Causes SMS to perform a check for invalid settings that could prevent TUFLOW from running properly. If errors are found, they will be reported in the *Model Check* dialog with information on how to correct them.

Components Menu

The *TUFLOW Components* menu is accessed by right-clicking on the TUFLOW Compenents folder in the Project Explorer. The following menu command is available:

New 2D Geometry Component

Creates a geometry components. The TUFLOW simulation can contain components. 2D Grids and certain map coverages can be linked to this component.

Once a 2D geometry component is created the component item below the components folder has a right-click menu with the following commands:

Delete

Removes the component.

Duplicate

Creates a copy of the component with all linked grids and coverages.

Rename

Changes the name of the component.

Add External Files

Launches a browser to add external files to the component.

Grid Options

Brings up the TUFLOW *Grid Options* dialog.

Material Sets Menu

The *TUFLOW Material Set* menu is accessed by right-clicking on the TUFLOW Material Set folder in the Project Explorer. The following menu command is available: **New Material Set**

Creates a new material set. The TUFLOW simulation can contain multiple material sets.

A material set item below the Material Sets folder also has a right-click menu with the following commands:

Rename

Changes the name of the material set.

Properties

Brings up the *TUFLOW Material Properties* dialog.

Delete

Removes the material set.

Export

Saves the material set as a TUFLOW Materials File (*.tmf).

TUFLOW Menu Bar Commands

The following commands are available in the *TUFLOW* menu in the menu bar when the Cartesian Grid Module is enabled and a cell or cells are selected in a TUFLOW grid.

Cell Attributes

Enabled when one or more cells on a TUFLOW grid are selected. The attributes that can be set using this command are the cell code and the option to override the global initial water level (set in the 2D Model Control). The cell code options are:

- *active cell* : cell is part of calculations and the results grid domain; Cell Code = 1
- *inactive cell – in mesh* ; cell is not part of the calculations, but is a part of the results grid domain, Cell Code = -1; because comparing two simulations with the Data Calculator in SMS requires the SMS mesh results (*.2DM file) have the same domain, this option is primarily used to keep the model results compatible for comparison while taking the differences into account (e.g. areas of a floodplain that are raised to prevent flooding and are thus no longer included in the computation)
- *inactive cell – not in mesh* : cell is entirely removed from the computation and results, Cell code = 0

For SMS versions 11.1 and earlier the selected cells will only be used if *Specify cell by cell* is selected in *Cell codes* section of the *TUFLOW Grid Options* dialog.

Assign Material Type

This command is enabled when one or more cells on a TUFLOW grid are selected. The command will open the *Materials Data* dialog. The *Materials Data* dialog will allow selecting a material to assign to a cell. The material set for the cell will only be used if *Specify cell by cell* is selected in the *Materials* section of the *TUFLOW Grid Options* dialog.

Related Topics

- TUFLOW
- TUFLOW Simulation

Command Objects

Command Objects provide the ability to provide additional commands for TUFLOW simulations using the TUFLOW command file syntax. The SMS interface provides some guidance for using these commands and allows linking to objects created in SMS. Command objects can be used for:

- Create scenarios to do multiple runs using the same simulation.
- Share commands between multiple simulations or 2D Geometry components.
- Utilize commands not available in the SMS interface.
- Enter commands to read GIS data generated outside of SMS.

TUFLOW Command Object syntax is described in the TUFLOW Manual. Each line (except for blank/comment lines) has a command identifier. After the identifier most commands have are followed by a double equals "==" and a command value. In some cases, the command value may have multiple parts separated by blank spaces or "|." A few commands don't have any arguments and do not have a "==" at all.

Types of Commands

There are several kinds of commands that can be utilized through command objects. The types of commands that can be issued used include: 2D Model control commands (*.tcf), 1D Model control commands (ecf), 2D Geometry commands (tgc), Boundary conditions commands (*.tbc), and commands for specific events (*.tef). The event commands are defined with each event when using composite events. The rest are managed in the project explorer as command objects.

The Appendix of the TUFLOW manual contains a complete list of the commands that are available, the purpose of each command, and its syntax. The identifiers above in parentheses (*.tcf, *.ecf, *.tgc, *.tbc, and *.tef) are the abbreviations used by TUFLOW for file naming and also used to find the appropriate section of the Appendix for a set of commands.

Creating Command Objects

Command objects are created using right-click commands in the project explorer. After creating a TUFLOW simulation, a folder named "Command Objects" in the TUFLOW root folder should appear. Create command objects by right-clicking on the "Command Objects" folder, choosing *Create Command Object* , and choosing the appropriate type from the submenu. This will create a folder for command objects of this type along with a new command object. This approach can be repeated to create additional command objects of the same or different type.

Editing Command Object Contents

When command objects are first created, their contents are blank. Edit the contents of a command object by double clicking on the object or right-clicking and choosing "edit." This will bring up the command object editor dialog.

The command object editor dialog has two main parts. One part consists only of a text editor. The text in this editor is what is stored with the command object and will be used with the TUFLOW model. The commands for the object may be typed directly into the editor. Comments may be added and must start with the exclamation mark (!).

The other part of the dialog contains controls to assist in creating commands for the command object. There is a control showing the list of commands in a tree structure organized by the appendix where the command is found in the TUFLOW manual. This part of the dialog will also contain other controls based upon the command that is selected in tree control. For example, if the command requires a single floating point value a edit control will show where to specify the value for the command.

Once the command creation controls have been specified as desired, the command is inserted into the command text using the insert button. Once this has been done, editing of the command must be done in the editor. It is not possible to edit commands already created using the create command controls. Delete an existing command and insert another to use the controls to setup the command text.

Using Command Objects in Simulations

With the exception of event command objects (discussed later), command objects are used by creating links to the objects from simulations and geometry components. Links are created by dragging the command object under the simulation object. 2d model control command objects and 1d model control command objects are linked to simulations. 2D geometry commands and boundary condition commands are linked to 2D Geometry Components.

When a command object is linked to a simulation or geometry component, SMS writes a file with the exact text from the command object and links to the file from the TCF or TGC file as appropriate. Command objects are always used after any commands specified in the SMS interface and therefore will override duplicate specifications. Multiple command objects can be linked to the same object and will be used in order from top to bottom. Command objects can override settings defined in command objects used previously.

Event command objects are only used with composite events (** need to add link to this help topic). To set commands for a composite event, click on the button “Additional commands” in the TUFLOW events dialog (***link to dialog help). This text will be stored with the event and written to a file with the extension “tef” which will be used when running TUFLOW. The event command objects do not show up in the project explorer.

Scenarios in Command Objects

TUFLOW command objects can define scenarios in order to setup multiple models that have shared data without the need for separate simulation objects. Anything not specified for one or more scenarios is used in all scenarios for the simulation. Scenarios could be used to model different geometries by including z-lines or tins for specific scenarios but not others.

Scenarios can only be defined using command objects. Scenarios are defined by using the following commands:

- If Scenario
- Else If Scenario
- Else
- End If

A scenario block must start with “If Scenario” and end with “End If.” The other commands are optional.

Related Topics

- TUFLOW

Reading a TUFLOW Simulation

TUFLOW simulation files can be opened into SMS by opening the simulation's control file (*.tcf). This can be done by dragging and dropping the file into the graphics window or by using *File* | **Open.** Multiple simulations can be opened together, and SMS will only create one copy of the data that is shared between the projects.

TCF File

The following commands are supported when reading a TCF file into SMS:

- BC CONTROL FILE – Read the specified TUFLOW Boundary Condition file
- BC DATABASE – Set the active database
- BC EVENT NAME – Set the active event filename to be used
- BC EVENT TEXT – Set the text to be substituted by the BC EVENT NAME value
- CELL SIDE WET/DRY DEPTH – Set the depth for when cell sides wet and dry
- CELL WET/DRY DEPTH – Set the wet/dry depth for when a cell wets and dries
- CHECK INSIDE GRID – Set to ERROR, WARNING, or OFF
- DISPLAY WATER LEVEL – Display water level for cell at specified location
- END 1D DOMAIN – End block of 1D commands
- END 2D DOMAIN – End block of 2D domain definition
- END TIME – Set the simulation finish time in hours
- ESTRY CONTROL FILE (AUTO) – Set the ESTRY control filename
- GEOMETRY CONTROL FILE – Read the specified TUFLOW Geometry Control file
- HX ZC CHECK – Set whether or not to check the minimum ZC elevation is above the 1D bed level
- INSTABILITY WATER LEVEL – Set the water level used to detect instabilities
- MAP OUTPUT DATA TYPES – Define data types to be output in map format
- MAP OUTPUT INTERVAL – Set the output interval for map based output in seconds
- MASS BALANCE OUTPUT – Set whether or not to output mass balance data
- MI PROJECTION – Set the geographic projection
- NUMBER ITERATIONS – Specify the number of iterations per timestep
- READ FILE – Read the specified external file as an addition TUFLOW control file
- READ GIS FC – Read the specified flow constriction mif/mid files
- READ GIS IWL – Read the specified initial water level mif/mid files
- READ MATERIALS FILE – Read the specified TUFLOW materials file
- READ MI PROJECTION – Set the projection
- READ RESTART FILE – Set the model to use the specified restart file
- SCREEN/LOG DISPLAY INTERVAL – Set the frequency for display of output to the computer screen and log file
- SET IWL – Set a default initial water level for all cells
- START 1D DOMAIN – Start a block of 1D commands
- START 2D DOMAIN – Begin block of 2D domain definition
- START MAP OUTPUT – Set simulation time in hours when map output begins
- START TIME – Set the start time of the simulation in hours
- STORE MAXIMUMS AND MINIMUMS – Set whether or not to store the highest and lowest values
- SX ZC CHECK – Set whether or not to check if the minimum ZC elevation is below the connected or snapped 1D node bed level
- TIMESTEP – Set the computation time step in seconds
- VISCOSITY COEFFICIENT – Set the viscosity coefficient(s)

- VISCOSITY FORMULATIONS – Set the viscosity formulation
- WRITE RESTART FILE AT TIME – Set when to write the restart file in hours
- WRITE RESTART FILE INTERVAL – Set the interval in hours between writing the restart file
- ZERO NEGATIVE DEPTHS IN SMS – Set whether or not to zero depths if negative

The following commands are not supported and are written to an external file:

- ADJUST HEAD AT ESTRY INTERFACE
- APPLY WAVE RADIATION STRESSES
- APPLY WIND STRESSES
- BC WET/DRY METHOD
- BC ZERO FLOW
- BED RESISTANCE CELL SIDES
- BED RESISTANCE DEPTH INTERPOLATION
- BED RESISTANCE VALUES
- BOUNDARY CELL SELECTION
- CALIBRATION POINTS MI FILE
- CHANGE ZERO MATERIAL VALUES TO ONE
- CHECK MI SAVE EXT
- CSV TIME
- DEFAULTS
- DENSITY OF AIR
- DENSITY OF WATER
- DEPTH/RIPPLE HEIGHT FACTOR LIMIT
- DISTRIBUTE HX FLOWS
- DOUBLE PRECISION
- EXCEL START DATE
- FIRST SWEEP DIRECTION
- FREE OVERFALL
- FREE OVERFALL FACTOR
- FROUDE CHECK
- FROUDE DEPTH ADJUSTMENT
- GLOBAL FC CH FACTOR
- GLOBAL WEIR FACTOR
- INPUT DRIVE
- INSIDE REGION
- LATITUDE
- LINE CELL SELECTION
- MAP CUTOFF DEPTH
- MAXIMUM POINTS
- MAXIMUM VERTICES
- MESHPARTS

- MODEL PRECISION
- NULL CELL CHECKS
- NUMBER 2D2D LINK ITERATIONS
- OBLIQUE BOUNDARY ALIGNMENT
- OBLIQUE BOUNDARY METHOD
- OUTPUT DRIVE
- READ GIS CYCLONE
- READ GIS HURRICANE
- READ GIS ISIS NETWORK
- READ GIS ISIS NODES
- READ GIS ISIS WLL
- READ GIS ISIS WLL POINTS
- READ GIS GLO
- READ GISLP
- READ GISPO
- READ GIS XP NETWORK
- READ GIS XP NODES
- READ GISXP WLL
- READ GIS XP WLL POINTS
- READ ROWCOL IWL
- RECALCULATE CHEZY INTERVAL
- SHALLOW DEPTH STABILITY FACTOR
- SHALLOW DEPTH WEIR FACTOR CUT OFF DEPTH
- SHALLOW DEPTH WEIR FACTOR MULTIPLIER
- SNAP TOLERANCE
- START TIME SERIES OUTPUT
- START WIND OUTPUT AT TIME
- SUPERCRITICAL
- SX HEAD ADJUSTMENT
- TIME SERIES OUTPUT INTERVAL
- TIMESTEP DURING WARMUP
- UK HAZARD DEBRIS FACTOR
- UZ HAZARD FORMULA
- UZ HAZARD LAND USE
- UNUSED HX AND SX CONNECTIONS
- VERBOSE
- VG Z ADJUSTMENT
- WARMUP TIME
- WATER LEVEL CHECKS
- WAVE PERIOD
- WETTING AND DRYING

- WIND OUTPUT INTERVAL
- WIND/WAVE SHALLOW DEPTHS
- WRITE EMPTY MI FILES
- WRITE PO ONLINE
- WRITE Z1D CHECK FILES

The following commands are ignored when read in by SMS because they are hardwired:

- CHECK MI SAVE DATE – Hardwired to "OFF"
- LOG FOLDER – Hardwired to ".\\log"
- MI PROJECTION CHECK – Hardwired to "WARNING"
- OUTPUT FOLDER – Hardwired to "..\\results"
- WRITE CHECK FILES – Hardwired to "..\\check\\"

TGC File

The following commands are supported when reading a TGC file into SMS:

- ALLOW DANGLING Z LINES – Allows dangling z lines
- CELL SIZE – Sets the cell size
- GRID SIZE (N, M) – Sets the number of I and J in the grid
- GRID SIZE (X, Y) – Sets the grid size in x and y
- ORIENTATION – Sets the x,y location of a point along the x-axis
- ORIENTATION ANGLE – Sets the angle of the grid
- ORIGIN – Sets the location for the origin
- READ FILE – Reads an external file
- READ GIS CODE – Reads mif/mid files and stores the codes in polygons in a bc coverage
- READ GIS FC SHAPE – Reads mif/mid files to create a flow constriction coverage
- READ GIS FLC – Reads mif/mid files to create a 2D spatial coverage with form loss coefficient as the coverage property
- READ GIS IWL – Reads mif/mid files to create a 2D spatial coverage with initial water level as the coverage property
- READ GIS LAYERED FC SHAPE – Reads mif/mid files to create a flow constriction coverage
- READ GIS LOCATION – Set the grid location
- READ GIS MAT – Reads mif/mid files to create an area property coverage
- READ GIS VARIABLE Z SHAPE – Reads mif/mid files to create a z-shape coverage
- READ GIS WRF – Reads mif/mid files to create a 2D spatial coverage with weir factor as the coverage property
- READ GIS Z LINE – Reads mif/mid files to create a z-lines coverage
- READ GIS Z SHAPE – Reads mif/mid files to create a z-shape coverage
- READ GIS ZPTS – Reads mif/mid files to create points in a z-lines coverage
- READ ROWCOL CODE – Sets code values for grid cells
- READ ROWCOL MAT – Sets material values for grid cells
- READ ROWCOL ZPTS – Sets z values for grid cells
- READ TIN ZPTS – Creates a scatter set from the points

- SET CODE – Set the default code
- SET IWL – If opening with a simulation, sets the default initial water level, otherwise the card is written to an external file
- SET MAT – Set the default material id
- SET ZPT – Set the default z-value for each grid cell

The following commands are not supported and are written to an external file:

- READ TGA
- READ TGC
- EXTERNAL BNDY
- READ GIS CNM
- READ GIS FRIC
- READ ROWCOL GRID
- SET CNM
- SET FRIC
- SET CODE ZERO ABOVE ZC
- WRITE MI DOMAIN
- WRITE MI GRID
- CREATE TIN ZPTS
- DEFAULT LAND Z
- INTERPOLATE ZC
- INTERPOLATE ZHC
- INTERPOLATE ZUV
- INTERPOLATE ZUVC
- INTERPOLATE ZUVH
- PAUSE WHEN POLYLINE DOES NOT FIND ZPT
- TIN COINCIDENT POINT DISTANCE
- READ MI Z HX LINE
- READ MI Z HX LINE RIDGE or MAX or RAISE
- READ MI Z LINE RIDGE or MAX | GULLY or MIN | HX
- READ MI Z SHAPE RIDGE or MAX or RAISE | GULLY or MIN or LOWER

TBC File

The following commands are supported when reading a TBC file into SMS:

- BC DATABASE – Set the database to be used
- BC EVENT NAME – Set the name of the event file to be used
- BC EVENT TEXT – Set the text to be replaced by the event name
- READ GIS BC – Read the boundary condition MIF/MID files
- READ GIS RF – Read the rainfall MIF/MID files
- READ GIS SA – Read the source flow MIF/MID files

The following commands are not supported and are written to an external file:

- GLOBAL RAINFALL AREA FACTOR

- GLOBAL RAINFALL BC
- GLOBAL RAINFALL CONTINUIN LOSS
- GLOBAL RAINFALL INITIAL LOSS
- READ GIS SA ALL
- READ GIS SA PITS
- READ GIS RF
- READ ROWCOL RF
- UNUSED HX AND SX CONNECTIONS

ECF File

The following commands are supported when reading a ECF file into SMS:

- DEPTH LIMIT FACTOR – Set the depth limit for detecting instabilities
- MANHOLE DEFAULT C EXIT COEFFICIENT – Set the K coefficient for automatic circular manholes
- MANHOLE DEFAULT LOSS APPROACH – Set the loss approach for automatic manholes
- MANHOLE DEFAULT R EXIT COEFFICIENT – Set the K coefficient for automatic rectangular manholes
- MANHOLE DEFAULT SIDE CLEARANCE – Set the side clearance for circular and rectangular manholes (automatic)
- MANHOLE DEFAULT TYPE – Set the type (circular, rectangular, no chamber, automatic) for automatic manholes
- MANHOLE K MAXIMUM BEND/DROP – Set the K coefficient for energy loss when using Engelhund approach with automatic manholes
- MANHOLE MINIMUM DIMENSION – Set the minimum dimension for circular and rectangular automatic manholes
- MANHOLES AT ALL CULVERT JUNCTIONS – Set whether or not to automatically create manholes at culvert junctions
- MINIMUM NA – Set the minimum surface area in all NA tables
- OUTPUT INTERVAL – Set the output interval for ESTRY output
- PIT INLET DATABASE – Read the specified inlet database file
- READ GIS NETWORK – Read the network mif/mid files
- READ GIS TABLE LINKS – Read the table link mif/mid files
- READ GISI WLL – Read water level lines mif/mid files
- READ GIS WLL POINTS – Read elevations and points from water level lines from mif/mid files
- STORAGE ABOVE STRUCTURE OBVERT – Define how surface area is to be contributed to the NA table
- TIMESTEP – Set the computation time step in seconds
- READ FILE – Read the external file as an ECF file
- SET IWL – Set the initial water level

The following commands are not supported and are written to an external file:

- BG DATA
- BRIDGE FLOW
- CHECK MI SAVE DATE
- CHECK MI SAVE EXT
- CREATE NODES

- CS DATA
- CSV FORMAT
- CSV TIME
- CULVERT ADD DYNAMIC HEAD
- CULVERT CRITICAL H/D
- CULVERT FLOW
- DEFAULTS
- END TIME
- FIXED FIELD FLAGS
- FLOW CALCULATION
- FROUDE CHECK
- HEAD RATE CREEP FACTOR
- HEAD RATE LIMIT
- HEAD RATE LIMIT MINIMUM
- INTERPOLATE CROSS-SECTIONS
- INTERPOLATE CULVERT INVERTS
- LOG FOLDER
- M11 NETWORK
- MI PROJECTION
- MINIMUM CHANNEL STORAGE LENGTH
- MINIMUM NA PIT
- MOMENTUM EQUATION
- NA DATA
- ORDER OUTPUT
- OUTPUT FOLDER
- OUTPUT TIMES SAME AS 2D
- PIT CHANNEL OFFSET
- READ MATERIALS FILE
- READ GIS IWL
- RELATIVE RESISTANCE
- S CHANNEL APPROACH
- SNAP TOLERANCE
- START OUTPUT
- START TIME
- STRUCTURE LOSSES
- TAPER CLOSED NA TABLE
- TRIM XZ PROFILES
- VEL RATE CREEP FACTOR
- VEL RATE LIMIT
- VEL RATE LIMIT MINIMUM
- VG DATA

- WEIR FLOW
- WLL ADDITIONAL POINTS
- WLL ADJUST XS WIDTH
- WLL AUTOMATIC
- WLL NO WEIRS
- WLL VERTICAL OFFSET
- WLLP INTERPOLATE BED
- WRITE CHECK FILES
- WRITE CSV ONLINE
- WRITE EMPTY MI FILES
- XS DATABASE
- ZERO CULVERT COEFFICIENTS

The following commands are ignored when read in by SMS because they are hardwired:

- APPLY ALL INVERTS – Hardwired to “ON”
- CONVEYANCE CALCULATION – Hardwired to “ALL PARALLEL”
- FLOW AREA – Hardwired to “TOTAL”
- WLL APPROACH – Hardwired to “Method B”

Related Topics

- TUFLOW

TUFLOW 2D Geometry Components

A 2D geometry component groups grids and data that apply to the grid. The other kinds of data in a geometry component include coverages and TIN (scatterset). A TIN is used to modify the Z values. The purpose of each coverage is dependent upon the coverage type. Some coverages will modify z values and others will control other attributes. There can be multiple objects of the same type in a geometry component. When this happens, the data is applied in the order of the objects in the component. This means that the objects below in the project explorer are the ones that will control. This applies to Z as well as other attributes. Links can be rearranged by dragging them up or down within the 2D Geometry domain.

Coverages in a geometry component

TUFLOW 2D Geometry components associate grids with TUFLOW coverages that apply specifically to the domain. For example a grid component may contain an area property coverage that modifies the materials in the grid. Grids and coverages are represented as TUFLOW Project Explorer Links inside of a 2D Geometry component. Only one grid or grid frame coverage may be linked in each geometry component.

The following coverages can be put inside of a 2D Geometry domain

- Area Property Coverage
- Boundary Condition Coverage
- 2D Z Lines/Polygons (Simple)
- 2D Miscellaneous(Spatial Attributes)
- 2D Z Lines (Advanced)

TINs (scatterset) in a geometry component

- In addition, Scatter Data can also be put inside a 2D Geometry domain to modify Z values. Unlike the other components, it has additional TUFLOW options that can be set in its link's right-click menu. The TUFLOW options that can be set for how the scatter data modifies Z values are "All", "Add", "Min" and "Max".
- Clip Regions for Read Tin z pts. Using a coverage to specify a clip region, makes it so only elevations within coverage polygons are changed by a TIN dragged under a 2D geometry component. This is particularly useful for clipping out a TIN due to unwanted or irregular triangulation around the periphery, especially for secondary TINs of proposed developments lying within the primary TIN.

To use this, create a coverage with one or more polygons. Any coverage type can be used, but it must be a coverage that allows polygons. Only areas within polygons and within the TIN boundaries will have elevations assigned. There must be a scatter set in the 2D Geometry Components. Then drag and drop this coverage under the scatter tree item, under the 2D Geometry Components.

After exporting the simulation files, the clip region will be referenced in the line "Read TIN Zpts" in the TGC file.

Considerations for modifying Z data

As mentioned previously, the order links are listed matters for coverages and scatter data. This can be particularly useful for Z data but make certain the attributes are applied in the correct order.

Suppose there is a situation where needing to incorporate a 2.0 m high fenceline and also raise a portion of the domain to model a proposed fill for a development. The fenceline is defined a z-line and the raised domain is a TIN (use a 2D Z Lines/Polygons (simple) coverage if it is a simple fill scenario). If the fenceline is above the TIN in the project explorer, anywhere that they overlap the TIN would take precedence and the fenceline would get wiped out. If the fenceline is below the TIN, the TIN elevations will get applied and then the fenceline will raise the elevations 2.0 m above the new elevations. Depending upon the scenario to be represented, either option may be the preferred mechanism.

External files

An external file can be read in as part of a geometry component. As previously mentioned, the order of links is important, and the external file behaves the same way as the z-modifying data. An external file is added by right-clicking on the geometry component and selecting **Add External File...** .

After a file has been selected, a tree item with the file name will appear at the bottom of the geometry component tree item group. This external file item can be moved around to the desired location and the file name can be changed by right-clicking and selecting **Browse...** .

Related Topics

- TUFLOW

TUFLOW AD

SMS supports the TUFLOW Advection Dispersion (AD) module. This allows modeling constituents using TUFLOW.

The TUFLOW AD module can be added to a paid edition of SMS.

Creating the Constituents

The constituent data is entered in the *Model Control* of the TUFLOW simulation. The *Constituents* tab allows entering in the following parameters for each constituent:

- *Name* – Enter the name for the constituent here.
- *Decay rate (units of day^-1)* – The decay rate of the constituent in units of day-1. This value is used in a first order decay calculations at each time step, i.e. Enter zero or blank if no decay is required.

- *Settling rate (units of m/day)* – The settling rate of the constituent in units of m.day-1. This value is used in a simple mass balance calculation that removes the constituent from the water column based on this rate. Enter 0 or leave blank if no settling is required.
- *Initial Condition Type* – Specified for each constituent as either a constant value or spatially variant. Select either "Constant" to enter a constant value or "Coverage" to select a spatial attributes (Miscellaneous) coverage .
  - *Initial Condition* – Specify a constant value or **Select** option to open a *Select Coverage* dialog to specify a spacial attributes coverage polygon.
- *Minimum Dispersion Type* – Specified for each constituent as either a constant value or spatially variant. Select either "Constant" to enter a constant value or "Coverage" to select a spatial attributes (Miscellaneous) coverage .
  - *Minimum Dispersion* – Specify a constant value or **Select** option to open a *Select Coverage* dialog to specify a spacial attributes coverage polygon.
- *Dispersion Coefficient* – Allows variation between constituents to permit simultaneous simulation of multiple constituents with varying dispersion properties.
  - *Longitudinal* – Direction parallel to flow.
  - *Transverse* – Direction perpendicular to flow.

Boundary Conditions

For each boundary condition and constituent, provide a curve representing concentration at the boundary. This can be done by selecting the **Advection Dispersion BC...** button in the *Properties* dialog for the arc or polygon. This brings up the *Advection Dispersion* dialog. In the dialog, select a constituent from the list provide. The list is generated from constituents in the *Constituents* tab of the *TUFLOW 2D Model Control* dialog.

Clicking the curve button to the right will bring up an *XY Series Editor* dialog where the curve information can be entered.

External Links

- TUFLOW AD 2011 Manual

Related Topics

- TUFLOW

TUFLOW Boundary Conditions

TUFLOW Boundary conditions may be defined at nodes, arcs, or polygons in a boundary conditions coverage .

Boundary Condition Locations

Boundary condition attributes must be assigned to a feature object. The boundary condition type is determined by the feature object being used.

1D Boundary Conditions at Nodes

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

Available boundary condition types for nodes and points include:

- "No BC" – No boundary condition exists at the location.
- "Flow vs Time" – Sets the flowrate as a curve through time.
- "Flow vs Wse" – Sets the flowrate based upon a relationship with the water surface elevation.
- "Wse vs Time" – Sets the water surface elevation as a curve through time.
- "Wse vs Flow" – Defines a relationship to determine the water surface elevation from a flowrate. The relationship can be specified or TUFLOW can compute it using water surface slope.

All of the boundary condition types for nodes or points include the options described below.

2D Boundary Conditions at Arcs

Arcs can be used to define 2D Boundary conditions applied to the 2D domain. The boundary conditions attributes are specified in the *Boundary Conditions* dialog.

Arcs can be assigned any of the following boundary condition types:

- "No BC" – No boundary condition exists at the location.
- "Flow vs Time" – Sets the flowrate as a curve through time.
- "Flow vs Wse" – Sets the flowrate based upon a relationship with the water surface elevation.
- "Wse vs Time" – Sets the water surface elevation as a curve through time.
- "Wse vs Flow" – Defines a relationship to determine the water surface elevation from a flowrate. The relationship can be specified or TUFLOW can compute it using water surface slope.
- "1D Flow/2D Water Level Connection (HX)" – Defines the interface between a 1D and 2D domain. See Combining 1D and 2D Domains . Options unique to this type include:
    - *Set ZU and ZV elevations to ZC value if lower*
    - *Allocate water levels along dry sections*
    - *Set 1D and 2D water levels to same value*
    - *Adjust ZC to slightly above 1D bed elevation*
- "Non-directional Flow vs Time" – Defines a flowrate that enters along each cell of the boundary.
- "Non-directional Flow vs Wse" – Defines a flowrate that enters along each cell of the boundary based upon a relationship to the water surface elevation.
- "Flow Source from 1D model (SX)" – Defines an interface between a 1D and 2D domain which is often useful for culverts rather than using the 1D Flow/2D Water Level Connection. Options unique to this type include:
    - *Adjust z elevation in each cell center*
    - *Do not set 1D water level*

Most of the boundary condition types for arcs include the options described below.

2D Boundary Conditions for Polygons

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

Polygon boundary conditions can be assigned one of the following three types:

- "No BC" – No boundary condition exists at the location. Includes the option *Set cell node* which can be set to:
    - "Inactive—not in mesh"
    - "Inactive—in mesh"
    - "Active".
- "Flow/Rainfall over Area" – Defines either an amount of flow entering the polygon (applies to all wet cells or lowest cell if none) or rainfall amounts (applies to all cells in the polygon).
- "Cell Codes" – Specifies whether cells are active or inactive and whether or not they will be included in the mesh output files.

Convert GIS Rainfall Data to TUFLOW BC Polygons

GIS files containing TUFLOW rainfall data may be imported into SMS and a Boundary Condition coverage with the data assigned to polygons will be created. The data for each polygon should be contained in the following columns: 8 STARTYEAR – Year of first data point (integer)

- STARTMONTH – Month of first data point (integer)
- STARTDAY – Day of first data point (integer)
- STARTHOUR – Hour of first data point (integer)
- STARTMIN – Minutes of first data point (integer)
- STARTSEC – Seconds of first data point (integer)
- TIMESPAN – Time span between data point in minutes (integer)
- NUMRAINPTS – Number of data points (integer)
- PT0 – Data for the first point (double)
- PT1, PT2,... – Data for each point

When converting GIS rainfall data to TUFLOW rainfall polygons, a prompt will appear to set a reference time. Each rainfall data point will be stored as total mm for each interval of time (hrs) from the reference time.

Events

Every boundary condition should be setup for each event that will be used with the boundary condition. The events can be defined by clicking on the **Add** or **Remove** buttons. Event names can be changed by double-clicking on the event. The list of events is common to all the boundary conditions in the project.

Options

Depending upon the type of boundary condition, there are several additional boundary condition options. Options common to all boundary types include:

- *Override Default Name* – SMS creates a default unique boundary condition identifier based upon the coverage and id of the arc. Override this value to more easily find boundary condition information in the TUFLOW check, and log files.
- *Spline Curve* – If this option is on, TUFLOW will smooth the boundary condition curve using a spline algorithm. This should generally only be used for smooth, cyclic boundary conditions such as a tidal boundary.
- **Advection Dispersion BC** – Brings up the *Advection Dispersion* dialog for TUFLOW AD. Requires having constituents created in the *TUFLOW 2D Model Control* dialog.

Related Topics

- TUFLOW
- TUFLOW Coverages

TUFLOW Check Files

TUFLOW provides excellent feedback for finding and fixing model errors. This feedback exists in several files. All of these files are prefixed by the name of the simulation that is running and the filename ending identifies the type of data contained in the file.

Log Files

In the \runs\log directory (within the TUFLOW files for the simulation), are the log files for individual run.

The most useful file for finding model problems is the *_messages.mif file. This file contains spatially located messages including errors, warnings or checks (listed in descending order of severity). These can be read into the GIS module of SMS. This provides quick feedback of model errors and the exact locations that the errors occur. This is very useful for pinpointing model setup problems and should be the first step in fixing model issues.

The *.tlf file contains information (in ASCII format) about how the run proceeded. It includes information on the parameters and settings used, the files that were read and used, as well as a copy of the screen information that is written as TUFLOW runs. This file can be used to verify the model is setup and is running as expected.

Check Files

In addition to the log files mentioned previously, TUFLOW writes additional files that can be used to verify model setup. These files are in the directory labeled "\\check" and are only written if the *Write Check Files* toggle is checked in the *Misc.* tab in the simulation *Model Control* dialog (this is the default).

Check files can be generated as text files and as MapInfo MIF/MID files.

The MIF/MID files can be read into the SMS GIS module where the **Get Attributes** tool can be used to see the values at individual locations.

TUFLOW potentially can generate a lots of check files. A description of the check files and what they contain can be found in the TUFLOW manual .

Related Topics

- TUFLOW
- MapInfo MID/MIF

External Links

- TUFLOW Wiki Check Files Article

TUFLOW Combining 1D and 2D Domains

1D and 2D Domains can be linked in three different ways.

1) A 2D domain within a 1D domain (1a)
2) 1D elements representing culverts or pipe networks (1b or 1c)
3) 1D cross-section based domain for river channel and 2D domain for floodplain (1c)

Linking 1D Pipe network within a 2D network

The easiest way to link a 1D pipe network to a 2D network is to double-click on one of the nodes in the network coverage and define an inlet (connection to the 2D domain). The inlet can be circular or rectangular and the shape and losses can be specified.

Linking 1D domain and 2D domains

Except for pipe networks defined above, linking a 1D network to a 2D network requires several coverages.

- Network coverage – Contains the 1D channels
- BC coverage – 1D Flow/2D Water Level Connection (HX) or Flow source from a 1D model (SX) arcs define the interface between the 1D and 2D domains.
- 1D/2D Connections – Arcs connect 1D nodes to ends of 1D Flow/2D Water Level arcs or Flow source arcs. When used with Flow source arcs only 1 connection arc is needed where the 1D Flow/2D Water Level arcs require 2 arcs (one to each end of the arc).

Arcs in the connections coverage must snap to the 1D network nodes and the endpoints of the 1D Flow/2D Water Level Connection arcs. The snapping option setup in the *Map* tab of the *Preferences* dialog should be used to ensure that the nodes are placed correctly.

The following image shows how 1D/2D linkages are setup both for items 1 and 3 above.

Related Topics

- TUFLOW

TUFLOW Flow Constriction Shapes

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

Standard (non-layered) Flow Constrictions

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

Layered Flow Constrictions

Here is the description from the TUFLOW manual:

> Four layers (not GIS layers!) are represented, with the bottom three layers each having their own attributes. The top, fourth, layer assumes the flow is unimpeded (eg. flow over the top of a bridge). Within the same shape, the invert of the bed, and thickness of each layer can vary in 3D. Each layer is assigned its own percentage blockage and form loss coefficient. For example, the layers of a bridge structure would be along the following lines.
>
> - Layer 1: Beneath the bridge deck. Might be 5% blocked due to the bridge piers and have a small form loss for the energy losses associated with the piers.
> - Layer 2: The bridge deck. This would be 100% blocked and the form loss coefficient would increase due to the additional energy losses associated with flow surcharging the deck.
> - Layer 3: The bridge rails. These might be anything from 100% blocked (solid concrete rails) to 10% blocked (very open rails). Some form losses would be specified depending on the type of rails.
> - Layer 4: Flow over the top of the rails – flow assumed to be unimpeded.
>
> Layered FCs function by adjusting the flow width of the 2D cell so as to represent the combination of blockages of the four layers, and by accumulating the form losses. When the flow is only within Layer 1, only the attributes of Layer 1 are applied. As the water level rises into Layer 2, the influence of the Layer 2 attributes increase as the water continues to rise. Similarly for Layer 3 and Layer 4.
>
> The cell side flow width is calculated by summing the flow areas of each layer (including the effects of layer blockages), and dividing by the water depth. The form losses are applied as follows:
>
> - If the water is entirely within Layer 1, the Layer 1 FLC is applied.
> - If the water level has reached Layer 2, the value applied is the Layer 1 FLC plus a fraction of the Layer 2 FLC based on the depth of water within Layer 2. For example, if Layer 1 FLC is 0.1 and Layer 2 is 0.5, and the water is 40% of the way up Layer 2, the FLC applied is 0.1 + 0.4*0.5 = 0.3.
> - Similarly, if the water level is into Layer 3, the FLC is the Layer 1 FLC plus the Layer 2 FLC plus a fraction of the Layer 3 FLC.
> - Once the water level is above Layer 3, ie. is into Layer 4, the FLC is held constant at the sum of the FLCs for Layers 1 to 3.
>
> -- *TUFLOW Manual 2008-08*

Varying Elevations Within Flow Constriction

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

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

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

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

Creating Flow Constrictions

To create a flow constriction for an arc or polygon:

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

Flow Constriction Options

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

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

Breakline Elevations

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

Layered Flow Constriction Options

- **Invert Elevation Offset** – This will offset the invert elevations whether they are specified or are the original cell elevations.
- **Layer 1 Low chord** – The low chord (obvert) represents the top of layer 1. For a bridge this is the low chord of the bridge deck.
- **Layer 2 and 3 Depths** – This represent the depth of the layers after layer 1. The obvert of each layer is determined on a cell by cell level from the layer1 low chord/obvert elevation.
- **Blockage %** – This represents the percentage of the flow width lost due to large piers, railings, or other flow impediments. If half of the flow is blocked, the percentage lost is 50%.

- **Form Loss** – The form loss is the ratio of the dynamic head that is lost in the structure. This value can be larger than 1.0 since all of the energy does not have to come from the velocity component. This can be estimated from experience or by comparing with other numeric codes.

Non-Layered Flow Constriction Options

- **FC Type** – Flow Constriction Type. Options are:
  - **General** – Does not include allowances for any vertical walls or friction from underside of deck.
  - **BC** – Box Culvert
  - **BD** – Bridge Deck
  - **FD** – Floating Deck
- **Low chord (obvert)/BC Height/Bridge Deck Depth** – The usage of this field changes depending on FC Type (above). Enter a sufficiently high value (eg. 99999) if there is no obvert constriction.
  - **General** or **BD** – Low chord/Obvert (soffit) of constriction in m above datum.
  - **BC –** The height of the box culvert.
  - **FD –** Floating depth (m) of the deck (ie. depth below the water line).
- **BC Width** – The width of one BC culvert barrel in metres. For example, if there are 10 by 1.8m wide culverts, enter a value of 1.8.
- **Blockage %** – The percentage blockage of the cells. For example, if 40 is entered (ie. 40%), the cell sides are reduced in flow width by 40%, ie. is set to 0.6 times the full flow width.
- **Manning's –** According to TUFLOW's documentation:

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

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

Ignored for General FC Types.

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

Node Options

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

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

Example: Arched Bridge

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

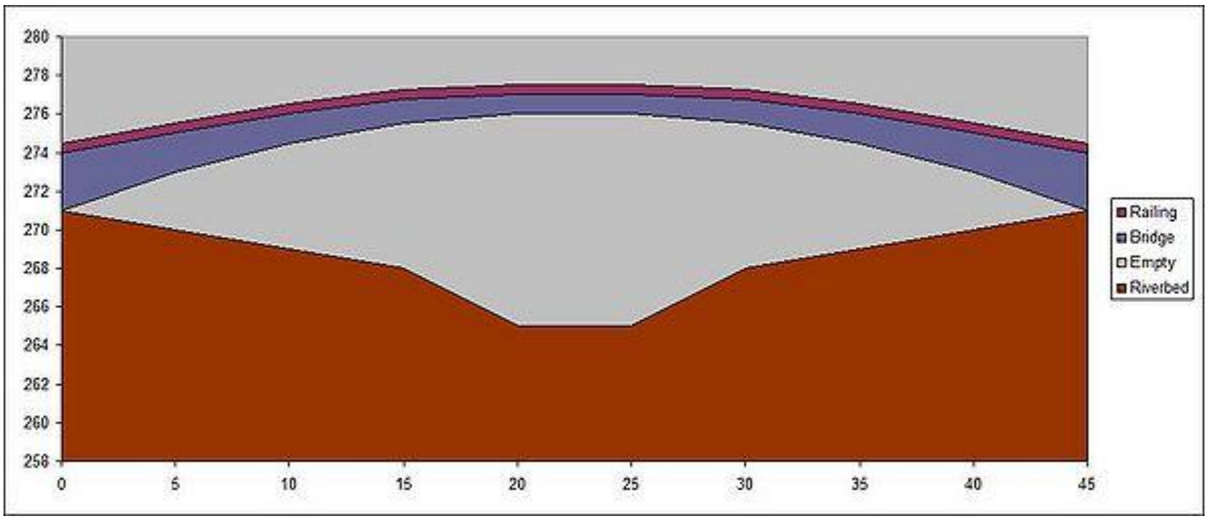
1) Start by creating the Flow Constriction Shapes coverage.
2) Next create 4 connected arcs in a rectangle for the polygon for the bridge. Go to *Feature Objects* | **Build polygons** to create the polygon.
3) Select the polygon and enter its attributes dialog. Fill the values as follows:
4) Click the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

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

Node7	275.5	1.25	0.5
Node8	274.5	1.5	0.5
Node9	273	2	0.5
Node10	271	3	0.5

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

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



Related Topics

- TUFLOW 2D Flow Constriction Shapes Coverage

TUFLOW Grid Options

The TUFLOW *Grid Options* dialog (accessed by right-clicking on TUFLOW 2D Geometry Component under the TUFLOW Components folder in the project explorer and selecting **Grid Options**) is used to specify how TUFLOW will handle the grid data. Some of the options deal with what types of data will be specified for the grid cells. This data can alternatively be specified using coverages and this has several advantages including size of the data, speed of execution, and more opportunities for reuse.

The general options include:

- *Materials*
  - *Specify cell by cell*
  - *Specify using area property coverage(s)*
  - *Default material* – Allows selection of the default material for grid cells that have not already been assigned.
  - **Materials** button – Brings up the general *Material Data* dialog.
- *Cell Codes*
  - *Specify cell by cell*

- *Specify using BC coverage(s)*
- *Default code*
- *Miscellaneous*
  - *Allowing dangling z lines* – Turns on/off the TUFLOW command to check for dialing z lines (see the TUFLOW manual for description).
  - *Override Global Time Step* – Since the required time step depends heavily on the size of grid cells, it is often convenient for each grid to have a unique time step when using multiple grids in a simulation. These controls allow overriding the global time step on a grid by grid basis.
  - *Default SRF*
- *External Files (Advanced)*
  - *Read External File in TGC file* – Can be used to read an external file from the TUFLOW TGC file.
  - *Read External File in 2D Domain block of TCF file.*

The rest of the dialog is used to tell SMS what kind of data to write from the grid in SMS. The optional data types are materials and cell codes . Materials may be specified using an area property coverage and cell codes can be specified from polygons in a TUFLOW BC Coverage. If a grid (instead of a TUFLOW grid extents coverage) is being used in the 2D geometry command component, these properties can also be specified cell by cell. If using an area property coverage, a default material is specified for any areas not inside of a polygon.

Related Topics

- TUFLOW
- TUFLOW Menu

TUFLOW Inlet Database

TUFLOW supports several types of inlets that connect 1D pipe network domains to 2D overland domains. These include circular, rectangular, and weir. TUFLOW now also supports the ability to define depth vs flow curves to allow for more flexibility. The user defined inlet information is stored in an inlet database.

The inlet database stores information about the user defined inlet types. This information includes the flow area, width, and the depth vs flow curve. The flow area is used to compute the velocities in the inlet channel and the width is used to determine how many 2D cells to connect the inlet to. For more information about how TUFLOW uses the inlet database, see the TUFLOW documentation.

To define the Inlet Database

1) Create a 1D network coverage and an arc.
2) Double-click on the end node of an arc (which is where inlets are defined)

3) Change the inlet type to "Inlet — From Database"
4) Click on the button labeled **Inlet Database...**
5) In the *Inlet Database* dialog that comes up, define the type (name), flow area, flow width, and curve for each of the inlets to be defined.

To use an inlet from an Inlet Database

1) Create a 1D network coverage and an arc.
2) Double-click on the end node of an arc (which is where inlets are defined)
3) Change the inlet type to "Inlet – From Database"
4) Select the inlet type by name in the combo-box labeled "Inlet Database Name."

Importing/Exporting the Inlet Database

From the *Inlet Database* dialog, it is possible to export or import the database in to the format used by TUFLOW (two or more csv files). This format is very easy to create or use for graphs using a spreadsheet program. For more information on this format, see the TUFLOW documentation .

Related Topics

- TUFLOW

TUFLOW Irregular Culverts

SMS supports irregular culverts for TUFLOW.

Creating the Irregular Culverts

In the network coverage, change the type of an arc to "Irregular Culvert" in the *Channel Attributes* dialog. By clicking on **Attributes** at the bottom of the *Channel Attributes* dialog, specify what shape the culvert is. The drop-down specifies what polygon will be used for this arc. New polygons can be created and named. The table specifies the points of the polygon in clock-wise order. Please note that negative values cannot be entered for y-values. The number of barrels, contraction coefficients, and losses can be entered in this dialog, similar to other culverts. The perimeter length of the polygon, and its area, are given for convenience.

When the TUFLOW simulation is exported, an HW table will be generated for each unique irregular culvert shape in use.

TUFLOW Linking 2D Domains

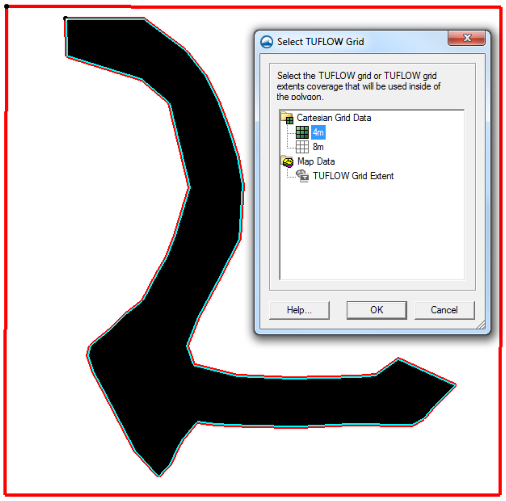
Any number of grids of varying sizes and/or orientations may be used in TUFLOW. Using multiple 2D domains requires that multiple 2D domains are licensed in TUFLOW.

To setup multiple 2D domains in SMS:

1) Create multiple TUFLOW grids and associated grid components.
2) Define a 2D/2D Linkage coverage and attributes (see below).
3) Add all the 2D domains being used and the 2D/2D Links Coverage to the simulation.

There are a few important items to keep in mind when setting up a 2D/2D Linkage coverage.

1) It's necessary that a polygon is enclosing each domain to be used. In the case below, a polygon is needed that covers the area outside of the main channel polygon.
2) Each polygon needs to be linked to a specific 2D domain. This is done by double-clicking on each polygon and specifying the appropriate 2D grid.
3) Set the vertex spacing on the arcs that join multiple 2D domains. The spacing along these arcs determines how flows move between the domains. The rule of thumb to start with is to have the vertex spacing equal to 1.5 times the larger cell size.



Related Topics

- [TUFLOW 2D/2D Links Coverage]

TUFLOW Manholes

TUFLOW in SMS allows for both automatic and manual manhole settings.

Automatic Manholes

Global manhole attributes can be set up for a TUFLOW simulation in the TUFLOW 1D model control. TUFLOW will then automatically create manholes when the following criteria are met:

- There is at least one incoming and one outgoing culvert – a culvert is a C, I or R channel.
- There are no open channels connected (ie. no bridges, weirs or any other channel that is not a culvert).
- Pit channels can be connected, but are not included in any of the calculations for determining manhole energy losses.

The following global attributes are set:

- *Automatic Manholes at All Culvert Junctions* – Whether or not to have TUFLOW automatically create manholes.
- *Manhole Type* – Circular, rectangular, junction, or automatically determined.
- *Side Clearance* – For circular manholes, the default side clearance from the side of the largest culvert to the side of the manhole chamber. For rectangular manholes, the side clearance from the side of the culvert to the side of the manhole chamber.
- *Minimum Dimension* – Minimum diameter for circular manholes and minimum width and length for rectangular manholes.
- *C Exit Coefficient* – K coefficient for flow out of circular manholes.
- *R Coefficient* – K coefficient for flow out of rectangular manholes.
- *Loss Approach* – Loss approach to be used (none, Engelhund, Fixed).
- *K Maximum Bend/Drop* – Maximum K energy loss coefficient that can occur for the sum of the loss coefficients for bends and drops at a manhole when using the Engelhund approach.

Manual Manholes

The global manhole parameters can be overridden by creating manholes in a TUFLOW 1D Network coverage. Toggling on *Manhole* and selecting the **Manhole Attributes...** button in the 1D Network *Node Attributes* dialog allows setting the following parameters for manually created manholes:

- *User specified ID* – Sets a name for the manhole
- *Override global manhole type* – Includes options for: "Circular (C)", , "Rectangular (R)", or "No chamber (J)".
- *Flow Width* – The flow width in meters. This is the diameter for circular manholes.
- *Flow Length* – The flow length in meters. Not used for circular manholes.
- *Override global loss approach* – Loss approach to be used. Options include: "None", "Engelhund", or "Fixed".
- *K fixed* – Set the fixed component of the calculated manhole loss coefficient.
- *Override global Km* – Exit coefficient used by Engelhund approach.
- *Override global K bend/max* – Upper limit of the combination of K theta and K drop used by Engelhund approach.
- *Additional storage surface area* – Usually set to zero.
- *Invert* – Bottom or bed elevation of manhole.

If a TUFLOW 1D Network coverage has manholes defined, the manholes will be written out as their own layer when exporting TUFLOW files.

(TUFLOW Manual 2016)

Related Topics

- TUFLOW
- TUFLOW 1D Network Coverage

TUFLOW Material Properties

TUFLOW makes use of both the SMS general materials and its own material properties.

Material properties include roughness and hydrologic losses (if used). TUFLOW uses a manning n value for roughness which can be specified as a single parameter or it can vary with depth. Hydrologic losses include an initial loss (in mm) and a continuing loss rate (in mm/hr). Hydrologic losses are generally only necessary when using direct rainfall on the model domain.

Material Sets

Multiple definitions of material properties can exist in a project. Each definition includes the material properties for each material in SMS and is referred to as a material set or material coverage. A material set can represent roughness for different conditions such as winter/summer conditions or can be used in a sensitivity analysis.

To build a material set, the TUFLOW simulation must already be built. Right-click on the Project Explorer "Material Set" item under the TUFLOW simulation and choose the **New Material Set** command. A "Material Set" item will appear in the Project Explorer.

Editing Material Properties

Material properties can be edited by right-clicking on the "Material Set" item in the TUFLOW simulation and choosing **Properties** . This will bring up the *TUFLOW Material Properties* dialog.

Material Properties Dialog Description

The *TUFLOW Materials Properties* dialog lists all materials created in the general *Materials Data* dialog. Each material can be selected. After selecting a material from the list, definitions can be added. The following definitions are available:

- Manning's n – This has two options: "Single Value" and "Roughness by Depth"
- *Provide hydrologic losses* – Allows setting *Initial* and *Continuing* hydrologic losses.
    - *Initial* – Sets the initial loss in millimeters if using direct rainfall boundary.
    - *Continuing* – Sets the continuing loss rate in mm/hr if using direct rainfall boundary.
- *Storage reduction factor* – Reduces or increases the storage capacity of the area. The default is 0 (no change). Positive values will reduce the storage capacity. For example, entering 0.1 will reduce the storage capacity by 10%. Negative values will increase the capacity.
- **General Material Properties** – Will open the *Materials Data* dialog.

Related Topics

- TUFLOW Model
- Material Data

TUFLOW Model Parameters

Run control parameters are divided into two sections for TUFLOW simulations. The main model control dialog and the model control parameters that apply specifically to the 1D portions of the TUFLOW run. Right-click on the TUFLOW simulation and choose the appropriate menu command to edit either set of model parameters.

This document will try to highlight frequently used model control parameters. Please see the TUFLOW documentation for a description of those items left out or for more information on specific items.

TUFLOW 2D Model Control

The main *TUFLOW 2D Model Control* dialog sets parameters for the run in general and some 2D specific controls.

Time

This page is used to set the run times for the simulation. The optimum time step size is determined by the courant number. The rule of thumb is that the time step should be about half as many seconds as the grid cell size in meters. So a grid with 10 meter cells, should run with a 5 second time step. The time step may be reduced to increase stability, but generally problems indicate another type of problem.

Output Control

This *Output Control* tab specifies what two dimensional (spatially varied) data should be saved in the TUFLOW run. The tab is broken down into three sections. These include the *Map Output* , *Output Datasets* and *Screen/Log Output* sections.

"Map Output"

This section specifies the format and frequency of output created by TUFLOW. Specify the following:

- *Format* – Choose between XMDF and binary data (*.dat) file format or both. (Map Output Format line in the *.tcf file)
- *Format Type* – Choose between normal grid resolution (SMS 2dm), double normal grid resolution (SMS high resolution), or reaveraged grid resolution (SMS low resolution fancy). (Map Output Format line in the *.tcf file)
- *Start Time* – Choose the simulation time in hours at which spatially variable dataset output starts. (Start Map Output line in the *.tcf file).
- *Interval* – Choose the time interval in seconds between times steps in spatial datasets saved by TUFLOW (Map Output Interval line in the *.tcf file).
- *Minimums/Maximums* – TUFLOW can compute minimum and maximum values or only the maximum values for the simulation. This can be very handy in visualizing flood extents. (Store Maximums and Minimums line in the *.tcf file).
- *Mass Balance Output* toggle – Turn this toggle on to have TUFLOW save the mass error (%) or cumulative mass error (%). The specific option must also be turned on in the *Output Datasets* section. (Mass Balance Output line in the *.tcf file).
- *Zero negative depths* – By default, TUFLOW zeros depths that are computed as negative values at cell locations. If this option is turned off and the display options are set in SMS to only show positive depths, partially wet cells can be seen. (Zero Negative Depths in SMS line in the *.tcf file).

"Output Datasets"

This section controls which datasets should be created by TUFLOW. The list is long and getting longer with each revision of TUFLOW. Select the toggle next to the desired items. (Map Output Data Types line in the *.tcf file).

"Screen/Log Output"

By default TUFLOW outputs to the screen information for each time step. This information is also written to a log file (*.tlf). TUFLOW supports various options for this type of run time feedback that are controlled in this section of the output control tab.

- *Show water level for a point* toggle and location – Instructs TUFLOW to display the water level on the screen for the cell located at the specified location. (Display Water Level line in the *.tcf file).
- *Display interval* – Decreases the frequency of the output information during a TUFLOW run. A value of 0 or 1 outputs at each time step. A larger positive value decreases the frequency to one in the specified number of time steps. A value of -2 suppresses all output except negative depth warnings and a value of -3 suppresses all time step display (Screen/Log Display Interval line in the *.tcf file).
- The user or developer can now specify additional text in the TUFLOW model control output tab which will be treated as custom output types. This will be used to specify output types that are not yet supported directly in the interface.

Wetting/Drying

See the FESWMS manual for information on these options. Generally the default options are fine.

- Wet/Dry Depth
- Cell Side Wet/Dry Depth
- 2D Cell Elevation Checks
    - Ensure HX Cells Are Above 1D Node Bed

- Ensure That SX Cells Are Below 1D Node Bed

Restart Files

Restart files can be used to start a simulation part way through a run. They are generally used to generate an initial condition for several simulations that start with the same boundary conditions but are later changed for different events. For example to evaluate 10, 20, and 100 year events in a tidal area it might be useful to run a couple tidal cycles of a normal event and save these to a restart file. The restart file could be used with the 10, 20, and 100 year events without rerunning the first couple tidal cycles.

Water Level

- *Initial Water Level* – All models need to have an initial water level. Any cells with a lower elevation than the initial water level will start out wet and the rest will start dry. A TUFLOW model does not require a fully wet domain when starting like FESWMS or RMA2. It is generally a good idea for the initial water level to match the downstream boundary condition at the start of a run. The global initial water level can be overridden on a local level using a 2D Spatial Attributes Coverage .
- *Override default instability level* – By default TUFLOW will assume that if the water rises 10 m above the highest elevation in the grid then the run is unstable. This value may be overridden to allow the water level to get higher than this without causing TUFLOW to abort.

Eddy Viscosity

Eddy viscosity is used to compute energy losses due to turbulence not modeled at the scale of the simulation. The following options are available:

- *Formulation* – Supports specification of a "Constant" eddy viscosity or using "Smagorinsky" specification or "Both".
- *Smagorinsky coefficient* – When using this specification, the coefficient is generally between 0.06 and 1.0.
- *Constant coefficient (m^2/s)* – When using this specification, it is recommended to use the default eddy viscosity of 1.0.

BC

Boundary conditions in the TUFLOW model run can be modified with the following options:

- *SA Minimum Depth* – Sets the minimum depth a wet cell must have to apply an SA inflow.
- *Use SA Proportion to Depth* – Sets whether or not to proportion SA inflows according to the depth of water.
- *BC Event Name* – The boundary condition event to use is specified.

Materials

This section is used to specify which material set the simulation will use. Available material sets are listed based on sets that have been created in the TUFLOW simulation in the project explorer. Only one material set may be selected for use in the simulation run.

Misc.

- *Write Check Files* – TUFLOW creates a number of check files that can be used to verify model inputs. TUFLOW will run faster if not writing check files.
- *GIS check/output format* – Include options for "Mif file" and "Shapefile" outputs.
- *Check inside grid error setting* – By default TUFLOW will generate an error of entities exist outside the grid domain. If wanting to use data that extends beyond the grid, change this to warning or off.
- *Read External File* – The SMS interface does not support every option available in TUFLOW. Unsupported options can be used by creating a TUFLOW command file (see the TUFLOW documentation) and specifying the file name here.

- *Write Z pts as Binary Files (*.xf)*
- *TUFLOW Executable* – This option determines which executable to use when running the simulation. The options are double and single precision for both 32 and 64bit. By default, it is the option chosen in the startup preferences.
- *Use Mass Balance Corrector* – This option sets whether or not TUFLOW should use the mass balance corrector, which carries out an additional iteration of the mass balance equation every half time step. This can result in significant reductions in mass error for problematic models, particularly those with steep and/or very shallow flow.

Constituents

This is tab is used for setting parameters related to TUFLOW advection dispersion module. See TUFLOW AD for more information.

1D Control

General

- *Time Step* – The optimum 1D time step is based upon the courant number. The 1D time step can generally be made quite small without affecting run times.
- *Output Interval* – How often to write 1D solution data to the csv solution files.
- *Initial Water Level* – The initial water level should generally be the same as the 2D water level.
- *Water level lines vertical offset* – This value sets the vertical adjustment of the water level line elevations in the SMS *.2dm file. This command only affects the *.2dm file read in by SMS for visualization and does affect the hydraulic calculations.
- *Write Check Files* – If this is off TUFLOW will not write check files for the 1D data.
- *Read external estry control file* – The SMS interface does not support all of the options available for 1D controls. It is possible to build an estry control file (see the TUFLOW documentation for instructions) and have it read at the end of the SMS generated files.

Network

- *Depth Limit Factor* – This determines the depth that TUFLOW will consider unstable and abort the run. The default value is 10, meaning that if the channel depth is 10 times larger than the depth of the channel the simulation will be considered unstable.
- *Conveyance calculation* – For 1D cross-section based channels, the conveyance calculations are divided into subsections. The subsections can be divided anywhere there is a change in resistance or a channel can be formed around each channel point(All parallel option). If the all parallel option is not used it is possible for the conveyance in a section to decrease when the depth increases which gives TUFLOW and error.
- *Minimum nodal area* – Nodes with a small storage area can have stability problems. Adding a minimum nodal area for all nodes has the potential to make the run more stable but may attenuate the model solution.

Automatic Manholes

- *Automatic manholes at all culvert juntions* – whether or not to have TUFLOW automatically create manholes.

See the article TUFLOW Manholes for more information.

- *Type*
  - *Manhole type* – Circular, rectangular, junction, or automatically determined.
  - *Side clearance (m)* – For circular manholes, the default side clearance from the side of the largest culvert to the side of the manhole chamber. For rectangular manholes, the side clearance from the side of the culvert to the side of the manhole chamber.

  - *Min. dimension (m)* – Minimum diameter for circular manholes and minimum width and length for rectangular manholes.
  - *K coefficient (circular)* – K coefficient for flow out of circular manholes.
  - *K coefficient (rectangular)* – K coefficient for flow out of rectagular manholes.
- *Loses*
  - *Method* – Loss approach to be used (none, Engelhund, Fixed).
  - *K max bend/drop* – Maximum K energy loss coefficient that can occur for the sum of the loss coefficients for bends and drops at a manhole when using the Engelhund approach.

Related Topics

- TULFOW

TUFLOW Network Node Attributes

SMS's Network *Node Attributes* dialog has several new additions to the *Create Connection to 2D Domain (SX)* section. These new options allow controlling elevations at the connections, how many cells are connected, and the method for selection of additional cells (Grade or Sag). Each option correlates fairly directly to a TUFLOW field and some are labeled as such to make look-up easy.

The TUFLOW Network *Node Attributes* dialog has the following options:

- *Type* – Each node can be assigned as one of the following types: "Generic", "Inlet – Circular", "Inlet – Rectangular", "Inlet – Weir", or "Inlet – From Database".
- *User specified Id* – Allows specifying a name for the node.
- *Create connection to 2D domain (SX)* – Connect 1D nodes to the 2D domain.
  - *Force cell z to at or below node z (invert/ground level)(SXZ)*
  - *Lower 2D cell elevation (SXL)* – Specified to connect the 1D node to the 2D domain and lower the 2D cell by the given value.

- *Don't alter cell Z*
    - *Method to choose additional cells*
- *Method to determine number of connected 2D cells* – Provides two options: "Automatic Plus" or "Specified".
    - *Number of Additional*
    - *Number of Cells*

Generic Node Options

- *Nodal area curve*
    - "Overwrite existing storage"
    - "Additional to existing storage"
- *Set Channel Invert* – Sets the inverts of connected channels. Default is "-99999".
- *Manhole* – Allows manually defining manholes. Clicking the **Manhole Attributes** button will open the *Manhole Attributes* dialog.

Inlet Options

- *Additional Nodal* – Adds the value specified as additional nodal area.
- *Ground Level*
- *Invert (bottom)* – The bottom elevation of the node.
- *Number of inlets*
- *Manhole* – Allows manually defining manholes. Clicking the **Manhole Attributes** button will open the *Manhole Attributes* dialog.

Circular Inlet Options

- *Shape Diameter*

- *Contraction Coefficients Width*
- *Losses*
    - *Entry*
    - *Exit*

Rectangular Inlet Options

Shape

- *Width*
- *Height*

Contraction Coefficients

- *Width*
- *Height*

Losses

- *Entry*
- *Exit*

Weir Inlet Options

- *Shape Width* – Width of the weir in meters.
- *Weir coefficient Multiplier* –

From Database Inlet Options

- *Inlet database name* – Contains a list of available inlets from a database.
- **Inlet Database** – Opens the *Inlet Database* dialog.

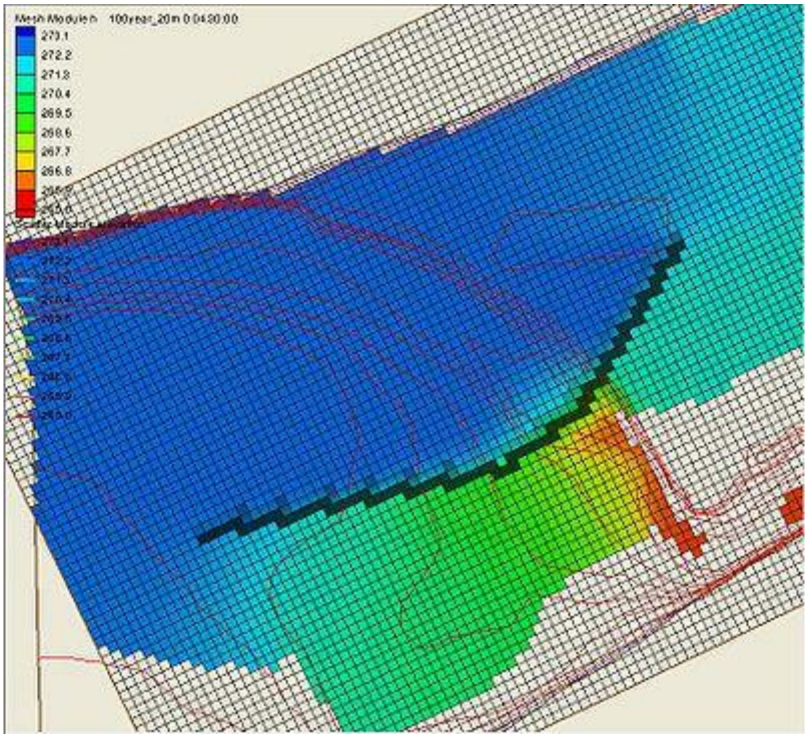
Related Topics

- TUFLOW 1D Network Coverage

TUFLOW ZShape

The TUFLOW ZShape coverage allows modifying terrain with arcs and setting up triggers to change those modifications, such as at a specific time during the simulation or when water depth exceeds a certain amount. An example application is a levee collapsing when flood water overtops it.

TUFLOW also supports polygon ZShapes. However, SMS does not support polygon ZShapes at this time. If needing to create static modifications to terrain using polygons, use the 2D Z Lines/Polygons (Simple) Coverage .



The ZShape Coverage

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

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

Arc Properties Dialog

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

- *Override Z Values* – ZShape data is only stored and exported if this checkbox is enabled. It tells TUFLOW to override the current terrain Z values with the new values from the arc.
- *Thickness* – Arcs can be Thin Lines, Thick Lines, or Wide Lines. Thin lines have a width of 0, thick lines have a thickness less than or equal to 1.5 times the 2D cell size, and wide lines are any width larger than that. Thin lines follow some special rules that affect other options in the dialog, such as Option and Trigger Type.
- *Option* – The ZShape Options are All, Add, Min, Max, and Offset. Offset is only available for variable thin line Z Shapes, which cannot use the "Add", "Min" or "Max" options. "Add" is unavailable for any variable Z Shape.
    - Offset is not listed by the same name in the TUFLOW documentation and is an alias in SMS for not specifying "NO MERGE". This causes the points of the arc to not be written, and the Z of the arc is determined by the Offset field.
- *Offset* – Raises the entire arc by this amount. Negative values are appropriate.

    - When offset is specified as the Option for variable thin lines, this field specifies the height to adjust terrain to (instead of adjusting it up or down by this amount) when triggered.
    - For all other variable arcs, this the the amount to adjust the terrain by when the trigger activates.
- *Use Trigger* – Toggles the arc between being a static zshape and a variable zshape. Variable z shapes use triggers to change terrain when a specific event occurs. All variable Z Shapes have the Duration option to span the change over a certain period of time. Other options will change with the Trigger Type.
    - *Trigger Type* – The type of event that will trigger the Z modification.
        - "Specified time" – Triggers at a specific time during the simulation, measured in hours.
        - "Water level at point" – Uses a Trigger Point to measure water level, and triggers when the water level reaches or exceeds the value in the Water level field.
        - "Water level difference" – Uses two trigger points. Triggers when the difference between water levels measured at both points exceeds the value in Water level difference. A difference in either direction is treated the same in TUFLOW, and there is no need for negative numbers.
        - "Thin line water level" – Similar to Water level at point, but for thin lines. Does not use a trigger point, instead measuring the water levels on either side of the thin line.
        - "Thin line water level difference" – Triggers when the difference between water levels on either side of the thin line reaches or exceeds the value in water level difference.
    - *Point1* and *Point2* – Both point selection boxes list the trigger points available in the coverage. These fields are only enabled for trigger types that use them.
    - *Specified time, Water level, Water level difference* – These fields become available based on the trigger type selected.
    - *Duration* – If set to 0, changes are instantaneous once triggered. Otherwise the change will be interpolated over this many hours.
- *Restore Z Shape* – This option can be used to have TUFLOW restore the original elevations at a specified time.
    - *Repeat restoration* – The z-shape will be restored indefinitely.
    - *Restore interval* – The time (hrs) between when the variable z-shape is finished and when to start restoring the points back to their original values.
    - *Restore period* – The time (hrs) needed to restore the points back to their original values.

Trigger Points

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

Trigger Point Dialog

The *Trigger Point* dialog is used to turn a feature point into a ZShape trigger point. Enabling the *Specify Trigger* checkbox and giving the point a name will cause it to appear in the Point1 and Point2 selection boxes in the arc attributes dialog for use as a trigger point.

If renaming a trigger point that is being used by triggers, or deleting it by unchecking *Specify Trigger* , SMS will notify that there are one or more arcs/polygons being updated with the new change.

Related Topics

- TUFLOW

7. Appendix

Bugfixes SMS

The SMS Intermediate Release Bugfixes page for released versions of SMS:

SMS 12.2 Release Bugfixes

SMS 12.2.1 - October 10, 2016

- 09185 1D Summary Table Crash
- 09186 Can't close summary table dialog with X`
- 09145 SMS Exporting HY8 Culvert Arc Nodestrings Incorrectly
- 09142 Revert subset hangs or crashes
- 09138 Working in subset mode confuses ADCIRC boundary conditions
- 09144 ADCIRC nodestring with multiple instances of same node not detected and fixed
- 09139 ADCIRC BC types not correctly maintained when merging meshes
- 09057 Interpolating nodes puts mesh into disarray
- 09121 SMS DMI allows multiple coverages of same type to be added to a simulation if added by a single drag
- 09122 Inconsistency of terminology when unlinking elements from a SRH-2D simulation
- 09146 Opening ADCIRC mesh in geographic coords does not set the projection if the data spans the international date line
- 09075 Check for Updates Not Working
- 09172 Crash opening project with SRH model

SMS 12.2.0 - September 10, 2016

- Initial Release!!

SMS 12.1 Release Bugfixes

SMS 12.1.8 - October 19, 2016

General

- 09075 Check for Updates Not Working
- 09145 SMS Exporting HY8 Culvert Arc Nodestrings Incorrectly
- 09172 Crash opening project with SRH model
- 09053 Default Help Option is said to be Online but Actually takes you to Local Help

Spectral Data

- 09124 Crash When Using Display Tools in Spectral Energy Dialog

Projection

- 09146 Opening ADCIRC mesh in geographic coords does not set the projection if the data spans the international date line

Plots

- 09114 Isues with observation plots
- 09020 Observation Plot Doesn't Look As Expected

Mesh

- 09139 ADCIRC BC types not correctly maintained when merging meshes

- 09057 Interpolating nodes puts mesh into disarray
- 08824 Deleting points of a mesh in non-display projection forgets what it is doing

Map

- 09127 Can't make observation arcs visible

Grids

- 09113 Bad grid created from Generic Grid generator coverage when set to "Mesh Centered" type

Feature Objects

- 09142 Revert subset hangs or crashes
- 09138 Working in subset mode confuses ADCIRC boundary conditions
- 09144 ADCIRC nodestring with multiple instances of same node not detected and fixed

Dynamic Model

- 09042 No default display of snapped arcs

SMS 12.1.7 - August 13, 2016

ADCIRC

- 08955 Edit fields for Harmonic parameters in files tab of ADCIRC model control
- 08937 Spatial Attribute changes not saved

BOUSS2D

- 09008 Crash When Attempting to Save, Export, and Launch BOUSS-2D Simulation

Contours

- 08957 Can't populate contours with proper range of values

Datasets

- 08974 Dataset conversion from vector to scalar not working for Quadtree from Toolbox

DISPLAY OPTIONS

- 09046 Raster Display Doesn't Update Properly when Changing Point Size and Maximum Points
- 08962 Incorrect update of CMS-Wave cell attribute display
- 08939 Display doesn't refresh after pan

Dynamic Model

- 09029 Can't run or specify location of aswip
- 09027 DMI crash when referencing nonexistent dialog

Feature Objects

- 08956 Select by Dataset Value does not work on UGrid Z dataset
- 08952 ADCIRC Mesh elevations revert back to old values when reopening SMS and project
- 08954 Nodestrings with Unassigned Boundary Conditions aren't Being Saved

General

- 08979 Text of the Coverage to Smooth Button is Overflowing
- 09024 SMS Steering Menu Item
- 09070 User gets error message when canceling out of ImportWizard
- 09041 SMS is Trying to Read in Files that it Shouldn't
- 08960 Open File Format appears instead of an error message
- 09069 SMS Coverage Duplicate Issues
- 08942 Can't access right-click menu for datasets

- 08963 PTM particles remain in display after deletion
- 09040 Arc BC data not saved
- 09087 The Time Step time is not being displayed when the time is 12:00:00 AM

Quadtree

- 08949 Can’t edit quadtree scalar datasets
- 08975 Quadtree transform fails to transform refined cell points
- 08882 Quadtree Grid menu items

Raster

- 08921 Raster Frame and Display
- 08914 Crash trying to open .las file as "Raster"

Simulation

- 08988 Crash When Creating New WAM Simulation

Spectral Data

- 08972 The direction of the wave components converting from spectra to CGWAVE is not spectra dependent and is wrong
- 08971 Converting spectra to CGWAVE wave components does not clear spectral list
- 08940 Can't work with high number of spectral indices

SMS 12.1.6 - June 2, 2016

ADCIRC

- 08899 ADCIRC solution with small timestep won't read in
- 08813 Incorrect NWS value written to control file

BOUSS2D

- 08768 B2D Wave maker minimum period (tmin) is not filled in anymore
- 08767 B2D Warning of time range is wrong

Datasets

- 08795 Map Elevation command doesn't use specified dataset
- 08745 Floodway tool Crash
- 08794 Scatter Visibility affects dataset used for interpolation
- 08782 Data Calculator behavior type vs. paste
- 08757 UGrid datasets generate their own dset contour options

Display

- 08880 CGWAVE nodestrings are numbered with bogus IDs
- 08865 Color of snap preview does not change after hitting OK

Export

- 08797 DMI - If a curve is associated with a "set", it is not exported
- 08765 BOUSS-2D damping dataset not computed completely

Feature Objects

- 08766 Wave maker no longer shows direction
- 08885 Weir from bug 8798 leaves elements in the interior
- 08881 ADCIRC node strings not numbered correctly

- 08798 Crash Creating ADCIRC Weir
- 08777 Canceling In Select Dataset Dialog Changes Functional Surface Options

General

- 08866 The Line Attributes dialog is not modal
- 08868 XMDF File Not Loaded In Correctly
- 08800 Functional Surface Won't Use User Defined Dataset

Interpolation

- 08755 Interpoate from Quadtree to Scatter vector crashes

Map

- 08829 Incorrect polygon fill display

Mesh

- 08830 Preview Mesh leaves a gap in the mesh.
- 08840 Issues with Functional Surface on Mesh
- 08787 Map-2D Mesh with multiple rasters

Particles

- 08856 Crash When Writing Particles Subset If the File Name Has not Been Set

Plots

- 08763 Observation plots not consistent
- 08764 Observation plot from Mesh is too long
- 08762 DEM Observation plot of data doesn't show up

Project Explorer

- 08769 SMS gets confused about the mesh module
- 08746 Missing features for TUFLOW 1D

Projection

- 08799 Missing Cross Section data in Summary Table
- 08854 Crash After Reproject All

Raster

- 08838 Incorrect Raster Elevations

Scatter Data

- 08907 Saving an empty scatter set corrupts project.
- 08908 Dialog indicates spacing is in meters when it is really in degrees
- 08848 Crash Saving Project After Failed Interpolation
- 08837 ADH Model Checker SI Units
- 08847 Error When Using an Existing Dataset for Extrapolation

SRH-2D

- 08748 Can't Open SRH solution after model run

TUFLOW

- 08846 Crash Saving TUFLOW Project

SMS 12.1.5 - April 20, 2016

CMCARDS

- 08739 Loading CMS-Flow project from SMS 11.2 does not associate the projection correctly

CMS Flow

- 08668 CMS Project Crash
- 08695 CMS-Flow Registration Item Disrupts SRH Coverages

Map

- 08682 Selecting Polygons Takes a Long Time
- 08690 SMS Does not Preserve "Bank" attribute type in the 1D Hydraulic Centerline Coverage

General

- 08697 Splash Screen Causes SMS to Hang
- 08702 SMS Main Menu, and Hot Key for Help Goes to Internet Only

GSSHA

- 08642 GSSHA Upstream Culvert

Import

- 08711 Opening multiple surveys at a single time gives error message

Mesh

- 08730 Map->2D Mesh is too slow "Setting nodal z values..."

Models

- 08575 Observation Data Cannot be Defined for stream arcs with an arc type of "General Stream"

Plots

- 08581 Drogue plot shifts when zooming out

Quadtree

- 08673 Problems Rotating on Quadtree/CGrid with Grid Frame
- 08672 Able to Create Quadtree When Quadtree Not Enabled

Scatter Data

- 08720 Converting coverage to spacing scatter gives wrong values
- 08719 Converting coverage to scatter hangs

SRH-2D

- 08747 Data sets don't appear under Mesh Data after running SRH-2D.
- 08689 SMS Not Saving Populated WSE Value

SMS 12.1.3 - March 03, 2016

- 08653 Contour options are not initialized on UGrid dsets after duplicate operation
- 08593 Empty spreadsheet double cells crash xms
- 08649 Crash opening .dep file
- 08596 Polygon Fill Display Incorrect
- 08635 Snap Preview won't stay off
- 08622 DMI range type edit boxes change values on blur
- 08623 DMI double clicking anywhere with polygon select tool enabled opens polygon attributes menu
- 08637 DMI Building New Polygons Messes Up Dependencies on Old Polygons
- 08597 Mesh Paving Crash
- 08604 Renumbering a mesh with gaps while color fill contours is on causes a crash
- 08595 UGrid Functional Surface doesn't display correctly when new active dataset selected
- 08607 ADCIRC files not copied to run directory when executing ADCIRC

- 08606 SMS does not read in ADCIRC data correctly
- 08615 Vector to Scalars Command Generates Dataset with Incorrect Velocities
- 08608 SRH-2D material coverage created from FESWMS coverage has bad attributes for polygons

SMS 12.1.2 - January 24, 2016

- 08544 - Dataset compare feature is not working
- 08527 - #file_name keyword is not accessible when process_on_condition is used in input_file
- 08571 - PBL output grids not displayed correctly
- 08536 - Crash After saving FESWMS Project and Reopening Saved Files
- 08567 - Opening CMS-Wave simulation with the Display Projection set to Geographic creates incompatible display case
- 08555 - Crash opening CMS-Flow project
- 08478 - Contours change when they shouldn't
- 08549 - Contour update issue on quadtrees after interpolating datasets from one to the Z of the other
- 08542 - Summary table from 2dMesh uses elevation value for all datasets
- 08550 - Cannot edit multiple nodes simultaneously on a vtk mesh
- 08547 - Min and max not set when interpolating one quadtree dataset to Z of another
- 08493 - Quadtree Created in the Wrong Direction after Changing Coverage Types
- 08566 - Date/Time stamps for the year 2000 - 2009 are read incorrectly when written with 8 digit format
- 08563 - Cells in a table become blank
- 08560 - SMS crashes when clicking the top left cell of the Linear BC table.
- 08561 - Wording correcting in film loop error
- 08557 - Merging coverages causes duplicate node ids
- 08526 - Order, i_order, and j_order attributes have no effect on process_each_polygon operations on cgrids
- 08481 - Redistribute Verices without cubic spline causes points in bad places
- 08545 - Confirmation setting not used when deleting coverages with the "Del" key
- 08404 - Saving and Reopening Coverages Removes Certain Characters
- 08492 - Wrong Popup When Generating a CGrid with No Grid Frame

SMS 12.1.1 - December 07, 2015

CMS Flow

- 08476 - Error Popup when Opening Old CMS-Flow Project File in 12.1

Coverage

- 08519 - Merged coverage has inappropriate atts that cause a crash when cleaning
- 08501 - Crash Deleting All Coverages

Feature Object

- 08513 - Double-Clicking on point in DMI Coverage does not bring up dialog

Grids

- 08508 - DMI unable to conditionally retrieve cell i and j based on bound coverage attribute
- 08511 - Points Snap to Incorrect CGRID Cells During CMS-WAVE2 DMI Export

Map

- 08496 - Strange Behavior and Crash When Merging Polygons

- 08495 - Crash When Merging Polygons
- 08452 - Redistribution of arc that is not in display projection causes problems

Mesh

- 08483 - Incorrect mesh preview in polygon attributes
- 08499 - Incorrect Mapping of BC arc to mesh
- 08491 - Crash committing mesh subset
- 08502 - SMS deletes duplicate elements without reporting what elements are being deleted

Opening File

- 08494 - Crash or Hang when opening project
- 08507 - Cross sectional database with "." in the name gets trimmed

Project

- 08522 - Crash Opening SRH Tutorial

Project Explorer

- 08330 - Project Explorer Items not updating their display properly

Quadtree

- 08493 - Quadtree Created in the Wrong Direction after Changing Coverage Types

Raster

- 08488 - Observation plot options allow user to select images if raster is loaded, but then says it is invalid

Saving

- 08473 - Crash during save of generic model

Scatter Data

- 08482 - Crash loading project file merging dynamic database (MergeAfterAllFilesReadIn)

SMS 12.1.0 - Release November 17, 2015

- Initial Release!!

SMS 12.0 Release Bugfixes

SMS 12.0.7 - Release October 22, 2015

- **Contours**
  - 8278 - Functional surface contour dialog name incomplete
- **Generic Model**
  - 8223 - ADH Hotstart folder created when switching models
- **GIS**
  - 8092 - Crash on Exit when working with GIS data
- **HY-8**
  - 8250 - SMS not specifying the location of the HY-8 executable in the preferences
- **Import**
  - 8261 - Update for Import Options in Tutorials
- **Mesh**
  - 7912 - Scalar Paving Meshing Results not Accurate
  - 8254 - GIS right click menus not finished

  - 8248 - Can't Create mesh nodes when only SRH Interface enabled
  - 8296 - Global parameters problems in .2dm file
  - 8280 - SMS doesn't detect the mesh nodes (or the mesh at all)
  - 8268 - Projection not matching when trying to enter mesh subset edit
  - 8224 - Incorrect Z Values assigned to nodes when performing Map->2D Mesh
- **NOAA**
  - 8270 - Tidal Data commands for the Web menu do nothing
- **Quadtree**
  - 8360 - Quadtree hides the contour legend and arrow become transparent
- **SRH-2D**
  - 8307 - SRH2D Obstruction Arc Direction Causes Termination of SRH2D
  - 8263 - SRH-2D Time Series Option Displaying when it should not be displayed
  - 8216 - Observation Profile Plot not Updating to Show the Active Timestep
  - 8210 - Simple Z lines not writing to the .tgc file

SMS 12.0.6 - Release August 21, 2015

ADCIRC

- 8207 SMS crashes when loading fort.45.

BOUSS2D

- 8214 STWAVE mentioned in model checker description instead of BOUSS2D
- 8191 SMS automatically cycles through time steps of BOUSS2D WSE Animation output dataset

Cartesian Grid

- 8095 Generic CGrids are not saved with a project

GenCade

- 8209 Inlets (Reservoir Model and Jetties) dialog typo
- 8194 GenCade coverages lose attributes when merged

Generic Model

- 8223 ADH Hotstart folder created when switching from ADH to Generic model

Import Options

- 8244 Name of 'Z' dataset not correct

Mesh

- 8229 Crash computing mesh spacing
- 8153 Long Processing Time for Map->2D Mesh

Plots

- 8216 Observation Profile Plot not Updating to Show the Active Timestep
- 8196 Crash Toggling through Timesteps with Observation Profile 2D Plot

SRH-2D

- 8171 SRH-2D Obstruction Coverage Issues
- 8173 SRH-2D and HY8 Runaway Processes

STWAVE

- 8201 STWAVE sim file import does not read projection

TUFLOW

- 8210 Simple Z lines not writing to the .tgc file

WAM

- 8114 WAM spectral site menu command Spectral Energy... doesn't work

General

- 8208 SMS crashes when reopening the current open project.
- 8083 PTM Waves and Breaking Input File Option cannot be removed from the model

SMS 12.0.5 - Release August 4, 2015

ADCIRC

- 8188 SMS crashes when opening multiple files including fort.64 fort.64 and other ADCIRC files.
- 8177 Fixed errors in ADCIRC LTEA tutorial to work with recent changes to the LTEA meshing toolbox.
- 8104 SMS was not creating the unit 23 file even when it was set to be created.

ADH

- 7984 Fixed issues with the sidewall boundary conditions.

BOUSS-2D

- 8117 BOUSS-2D Generating arcs along land boundaries not using elevation scalar set.

CGWAVE

- 8184 Unable to read in the solutions .cgo file.

General

- 8108 Can't delete items from project.
- 8180 Save tabular data mixing of meshes and datasets.
- 8107 Project corrupted after saving in certain cases.
- 8157 Delete key unresponsive when deleting arcs and other map objects.
- 8130 Deleting datasets that are changed outside of SMS caused datasets in the project tree to be deleted
- 8123 Removed unnecessary messages from Delete All operation.
- 8125 Reading in a Materials file with foreign characters does not read in properly.
- 8026 Removed unnecessary message about remote desktop from startup.

Images/GIS

- 8164 Can't save project with DGN data.
- 8138 MIF/MID importing errors.

Map

- 8159 Creating arcs caused node numbering to change, and added gaps.
- 8158 Can't find feature point by ID.
- 8100 Incorrect Extracted Values at Observation Arcs.
- 8033 Right-Click Menu not defined for dynamic maps.

2D-Mesh

- 8168 Mesh generation toolbox was missing in the menu for ADCIRC.
- 8162 Unable to deselect feature objects while multi-selecting in 2D mesh polygon properties dialogue.
- 8152 Incorrect snap preview.
- 8097 Mesh created with overlapping elements.

- 8116 SMS Not exporting BC node strings correctly.
- 8098 Crash optimizing triangulation of SMS generated mesh.
- 8099 SMS saves bad mesh after refining elements.
- 8091 Crash Interpolating scatter to mesh.

Scatter Data

- 8151 Can't merge scatter sets after using to much memory.
- 8154 Incorrect extracted plot from scatter in certain cases.

Simulations

- 8110 Model checker sometimes doesn't recognize the mesh and materials that are in the project.

SRH-2D

- 8186 user defined weir coefficient input boxes displayed for wrong options.
- 8161 has no upstream/downstream arc assignment when using paired boundary.
- 8167 Model control label is "Total Simulation Time" when it should be "Simulation End Time".
- 8160 material display options not working as expected on SRH-2D boundary condition meshes.

TINs

- 8137 Selecting triangles of a TIN causes SMS to crash in certain cases.

WAM

- 8061 Model check not functioning properly.
- 8062 Tutorial model does not run successfully.

SMS 12.0.4 - Release June 16, 2015

ADCIRC

- 8031 Loading ADCIRC Meshes Causes Multiple Meshes with the Same Name

CMS

- 8068 CMS Model Check and Sediment Datasets

Dynamic Model Interface

- 7978 dataset not saved in DMI

Generic Model

- 7991 Generic Mesh Model, can't uncheck Activate parameter group
- 7989 Generic Mesh Model does not save parameter group correlation for BC

2D Grid

- 7994 Display of Cartesian Grid Data is Incorrect

General

- 8060 Scale/Printing Issue
- 8080 Displayed snapping location is wrong when first opening the project
- 8029 Zoom to Selection Not working when display features are turned off
- 8067 Printing crash
- 8025 SMS does not remember the path to the HY8 executable if the registry is already set
- 8050 Slow vector display
- 8048 Plot From Observation Profile Not Appearing In Graph
- 8015 projection changes on load

- 8041 Non-admin user with partial SMS license can't set file paths
- 8008 Incorrect Profile Plot
- 7834 Various issues found while working on East End Bay project
- 7998 Reading project file hangs SMS

Images/GIS

- 8030 Shapes→Map and can create duplicate coverage names

Map

- 7591 Redistribute arc using source/target fails with bias distribution on source
- 8024 Spectral coverage "Projection.." right-click menu item does nothing

2D-Mesh

- 8079 Rectangular patch not clearing previous mesh
- 8052 Datasets not included in Mesh→Scatter Operation
- 8028 Saving Mesh Subsets
- 8023 snapping elements not correct
- 7940 BC lost when editing mesh subset
- 7918 Z direction of multiple meshes flips

SRH-2D

- 8032 SRH Not Outputting at Specified Interval
- 8071 Incorrect display of SRH boundary condition labels
- 8073 Changing one SRH-2D BC coverage changes other SRH-2D BC coverage
- 8064 SRH-2D Model Checker Crashes
- 8034 Time series curves linked on duplicate SRH BC coverages
- 7461 SRH Variable WSE tutorial files don't work
- 8016 SRH-2D Duplicate Coverage Errors
- 7999 The SRH-2D DMI writes multiple pressure cards in srhhydro file incorrectly

STWAVE

- 7345 Can't run STWAVE through SMS

SMS 12.0.3 - Release May 20, 2015

ADCIRC

- 7917 Single node value not editable for ADCIRC nodal attribute

ADH

- 7961 SMS hangs when opening ADH solution file
- 7878 ADH materials can't be deleted

BOUSS Runup/Overtopping

- 7942 Checkboxes in Runup/Overtopping Model Control Dialog Aren't Being Initialized

Dynamic Model Interface

- 7978 dataset not saved in DMI

General

- 8008 Incorrect Profile Plot
- 7834 Various issues found while working on East End Bay project

- 7998 Reading project file hangs SMS
- 7963 The case of the mysterious red rectangle
- 7951 Vertical Units Undefined
- 7949 Crash performing Model Check
- 7934 Unidentified Label in Node Options
- 7938 File Import Wizard Hangs SMS

Generic Model

- 7991 Generic Mesh Model, can't uncheck Activate parameter group
- 7989 Generic Mesh Model does not save parameter group correlation for BC

2D Grid

- 7994 Display of Cartesian Grid Data is Incorrect
- 7979 Cell sizes change with reprojection

Map

- 8024 Spectral coverage "Projection.." right-click menu item does nothing
- 7975 Incorrect BC arc labeling
- 7964 Crashing Stamping Feature
- 7915 Double-Click arc group crash
- 7933 Coverage created from scatter boundary has issues

2D Mesh

- 8023 snapping elements not correct
- 7918 Z direction of multiple meshes flips
- 7940 BC lost when editing mesh subset
- 7985 Certain BC not available when exactly two nodestrings selected
- 7941 Crash refining elements
- 7944 Nodal BC symbols not showing up unless the nodes are also displayed

Scatter

- 7955 Deleting scatter point converts dataset to Z
- 7937 Trimming a scatter set destroys the data values (only 1 dataset)

Simulation

- 7952 Renaming Mesh doesn't update name in simulation link

SRH-2D

- 8016 SRH-2D Duplicate Coverage Errors
- 7999 The SRH-2D DMI writes multiple pressure cards in srhhydro file incorrectly
- 7973 SRH BC not displayed
- 7974 SRH Geometry Nodestrings not written correctly for SRH Model
- 7966 SRH Project in 11.2 doesn't read correctly into 12.0
- 7957 Opening saved SRH-2D project creates duplicate datasets.
- 7950 Model Check for SRH says there is no mesh
- 7889 Attempting to Open a SRH-2D Causes a Crash

STWAVE

- 7784 STWAVE Numeric Model Crashes Midway Through Running

- 7378 STWAVE Tutorial Crash
- 7199 STWAVE Crash when Use time steps is specified

TUFLOW-FV

- 7925 Can't run TUFLOW-FV from SMS
- 7924 Can't read TUFLOW FV Vector Solution

SMS 11.2 Release Bugfixes

SMS 11.2.16 - Release March 04, 2016

- Update for Security in Installer

SMS 11.2.15 - Release November 25, 2015

- **ADCIRC**
  - 08436 Fixed an Issue where you couldn't run ADCIRC in Parallel
- **ADH**
  - 08439 Crash when setting ADH Initial Conditions
- **General**
  - 08330 Project Explorer Items not updating their display properly
  - 08329 Image visibility not saved with project
  - 08331 OK Button Under Other Dialog Items in the Map -> 1D Grid Dialog

SMS 11.2.14 - Release October 22, 2015

- **CMS Wave**
  - 8389 - CMS-Wave export does not save the "txt" file
  - 8392 - Save as "CMS-Wave sim" asks for name that it doesn't use
  - 8391 - CMS-Wave simulation reftime is defaulted incorrectly
- **General**
  - 8331 - OK Button Under Other Dialog Items in the Map -> 1D Grid Dialog
  - 8398 - Synthetic storm dialog does not have the correct icon in the dialog title bar
- **GIS**
  - 8316 - When opening a *_message.mif file into the GIS module, the font color cannot be changed
- **Grids**
  - 8378 - CMS-Wave Spec file has wrong value for Hs
- **Materials**
  - 8257 - Materials Data Loss
- **Mesh**
  - 8399 - Deleted curves keep adding to 2dm file
  - 8347 - Map->Mesh invalid for area property coverage
  - 8335 - Reloading a project after deleting a mesh in subset mode in the same SMS session causes problems
  - 8317 - Crash in Nodestring right-click menu
- **Project Explorer**
  - 8329 - Image visibility not saved with project

- **Spectral Data**
  - 8414 - Click and Drag Box not Showing up for Zoom, and Select in Spectral Energy Dialog
  - 8387 - SMS prompts for spectral coverage destination not precise
  - 8380 - Spectral datasets not deleted correctly when editing

SMS 11.2.13 - Release September 22, 2015

BOUSS2D

- 08214 - STWAVE mentioned in model checker description instead of BOUSS2D

CGrid

- 08228 - Crash Loading Project File

Contours

- 08278 - Functional surface contour dialog name incomplete

Datasets

- 08244 - Name of 'Z' dataset not correct

Generic Model

- 08223 - ADH Hotstart folder created when switching models

Mesh

- 08296 - Global parameters problems in .2dm file
- 08280 - SMS doesn't detect the mesh nodes (or the mesh at all)
- 08268 - Projection not matching when trying to enter mesh subset edit
- 08224 - Incorrect Z Values assigned to nodes when performing Map->2D Mesh

NOAA

- 08270 - Tidal Data commands for the Web menu do nothing

Plots

- 08258 - Profile Plot crash with transient data

SRH-2D

- 08210 - Simple Z lines not writing to the .tgc file

Other

- 08083 - PTM Waves and Breaking Input File Option cannot be removed from the model
- 08191 - SMS automatically cycles through time steps of BOUSS2D WSE Animation output dataset
- 08201 - STWAVE sim file import does not read projection
- 08194 - GenCade coverages lose attributes when merged
- 08207 - SMS crashes when loading fort.45.
- 08208 - SMS crashes when reopening the current open project.
- 08209 - Inlets (Reservoir Model and Jetties) dialog typo (GenCade)
- 08188 - SMS crashes when opening multiple files.

SMS 11.2.12 - Release August 4, 2015

ADCIRC

- 8177 Can't run model in LTEA tutorial
- 8104 SMS not creating unit 23 file for ADCIRC

General

- 8188 SMS crashes when opening multiple files.
- 8108 Can't delete items from project
- 8107 Project corrupted after saving
- 8130 Deleting Datasets that are changed outside of SMS
- 8138 MIF/MID importing errors
- 8158 Can't find feature point by ID
- 8123 Unnecessary messages during Delete All operation
- 8125 Reading in a Materials file with German characters does not read in the German characters.
- 8026 When first starting SMS an annoying message about remote desktop appears

Map

- 8168 Mesh Generation Toolbox Menu Item
- 8159 Creating Arcs changes feature point numbering

2D Mesh

- 8099 SMS saves bad mesh after refining elements

Scatter

- 8151 Can't merge scatter sets
- 8154 Incorrect extracted plot from scatter
- 8091 Crash Interpolating Scatter to Mesh

TIN

- 8137 Selecting triangles of a TIN causes SMS to crash

WAM

- 8061 WAM Model Check not functioning
- 8062 WAM Tutorial Model Does Not Run Successfully

SMS 11.2.11 - Release June 16, 2015

ADCIRC

- 8031 Loading ADCIRC Meshes Causes Multiple Meshes with the Same Name

CMS

- 8055 CMS-Flow model check and model wrapper errors
- 8068 CMS Model Check and Sediment Datasets

General

- 8060 Scale/Printing Issue
- 8029 Zoom to Selection Not working when display features are turned off
- 8067 Printing crash
- 8054 SMS not writing projection card to cmcards file
- 8050 Slow vector display
- 8048 Plot From Observation Profile Not Appearing In Graph
- 8015 projection changes on load
- 8008 Incorrect Profile Plot
- 7834 Various issues found while working on East End Bay project

Generic Model

- 8046 Error Message Appears Stating "No Defined Boundary Condition Options Found For This Entity.
- 7991 Generic Mesh Model, can't uncheck Activate parameter group
- 7989 Generic Mesh Model does not save parameter group correlation for BC

2D Grid

- 7994 Display of Cartesian Grid Data is Incorrect

Map

- 7591 Redistribute arc using source/target fails with bias distribution on source

2D Mesh

- 8079 Rectangular patch not clearing previous mesh
- 8052 Datasets not included in Mesh→Scatter Operation
- 8028 Saving Mesh Subsets
- 7918 Z direction of multiple meshes flips

SRH-2D

- 7468 SRH Restart File Option Not Saving
- 8032 SRH Not Outputting at Specified Interval
- 7461 SRH Variable WSE tutorial files don't work
- 7977 SRH restart file location not saved

STWAVE

- 7345 Can't run STWAVE through SMS

TUFLOW

- 8072 Changing visibility of CGrid causes errors in TUFLOW

SMS 11.2.10 - Release May 12, 2015

ADCIRC

- 7917 Single node value not editable for ADCIRC nodal attribute

ADH

- 7961 SMS hangs when opening ADH solution file

BOUSS-2D

- 7885 BOUSS2D not saving the damping grid file (may exist in 11.2 as well)

BOUSS Runup

- 7942 Checkboxes in Runup/Overtopping Model Control Dialog Aren't Being Initialized

DMI

- 7872 DMI output time table crashes when deleting a row

General

- 7981 Projection not set in Reproject dialog
- 7979 Cell sizes change with reprojection
- 7913 SMS not reading "merged" nodes from the *.map file
- 7910 Zoom and Pan Performance Issues
- 7901 When ok doesn't mean ok
- 7857 Data not saved to h5 file
- 7854 Saving Materials Coverage as a Shapefile does not include Material names or IDs

Generic Model

- 7991 Generic Mesh Model, can't uncheck Activate parameter group
- 7989 Generic Mesh Model does not save parameter group correlation for BC

GIS

- 7982 SMS Tutorial: GIS
- 7871 Shapes → Feature Objects Does not create new coverage

2D Grid

- 7994 Display of Cartesian Grid Data is Incorrect

2D Mesh

- 7985 Certain BC not available when exactly two nodestrings selected
- 7918 Z direction of multiple meshes flips
- 7863 Merging meshes causes BC to be lost
- 7850 SMS Hangs Merging Meshes
- 7852 Mesh projection blocks interpolation - late

PTM

- 7851 PTM Model wrapper update

Scatter Data

- 7955 Deleting scatter point converts dataset to Z
- 7937 Trimming a scatter set destroys the data values (only 1 dataset)
- 7888 Interpolate from scatter to scatter crash

SRH-2D

- 7977 SRH restart file location not saved
- 7962 Specified SRH model location doesn't save
- 7957 Opening saved SRH-2D project creates duplicate datasets.
- 7916 PreSRH2D not running
- 7889 Attempting to Open a SRH-2D Causes a Crash
- 7859 SRH-2D interface deletes the contents of the HY8 file (creates a blank file) when linking

TUFLOW

- 7925 Can't run TUFLOW-FV from SMS
- 7924 Can't read TUFLOW FV Vector Solution

SMS 11.2.9 - Release March 13, 2015

ADCIRC

- 7813 Crash loading ADCIRC unit 62 file

ADH

- 7757 Pre-ADH Error with ITL and NTL
- 7750 Can't delete initial depth folder after converting from ADH to SRH or other model

BOUSS-2D

- 7804 Incorrect units in BOUSS2D Wavemaker
- 7835 SMS Tutorial Error: Listboxes Lack Checkboxes in BOUSS2D Solution View Dialog
- 7726 SMS crashes when saving, exporting, and running BOUSS2D

CAD

- 7816 DXF→Map Crash

CMS

- 7693 CMS Project Error and Crash

General

- 7837 Review of help text for menu items
- 7827 Plot Wizard with VTK Mesh Crash
- 7805 SMS reverts to quadratic elements after using linear elements
- 7815 Convert Observation Points to Scatter Not Available
- 7814 Dummy Observation Points Set Incorrectly for Fort.62
- 7795 Contour range values not initialized correctly
- 7780 Duplicate Initial Depth Dataset
- 7786 Project File Crashes SMS
- 7766 Warning Window Constantly Popping Up About Contours Legend Box
- 7752 Contour options checked and changed when they are not on - causes annoyance and slow down
- 7723 Display of Project Explorer tree toggles not correct
- 7729 File | Save crashes SMS
- 7705 Gap in vector display
- 7716 Attempting to Open .sol File Without Opening a Project First Causes SMS to Crash
- 7623 Plot window film loop broken

Generic Model

- 7787 11.2 - Opening 2dm File Sets the Incorrect Value for Parameter
- 7769 Opening Project Has Issue With Generic Mesh Template
- 7754 Default "Generic Model Definition" gets created and is in the way
- 7753 Generic Model Coverages can become corrupted on read of "map" file.
- 7722 Generic template not saved to new folder
- 7687 Material Properties not saved in generic model

GIS

- 7808 GIS to Feature Objects Wizard Closes Without Ever Going to Step 2
- 7809 Not Selecting a Coverage type In the GIS to Feature Objects Wizard Causes a Crash

Map

- 7781 SMS Hangs when cleaning feature objects
- 7745 Offset arc elevation is set to 0.0
- 7741 Merging coverages does not maintain elevation of arcs
- 7704 Build Polygon command alters material type for some existing polygons

2D Mesh

- 7817 Merge Mesh Crash
- 7800 Nodestrings Not Updating Projection
- 7751 Message about mesh being out of date is not accurate
- 7696 Unable to delete VTK mesh dataset
- 7548 Merging Overlapping Meshes Causes Geometry Problems

PTM

- 7770 SMS Not Saving Particle Data Simulation

Scatter Data

- 7740 Conversion from selected vertices to scatter points not working

Spectral Energy

- 7768 Spectral Energy Angle Bin Size
- 7785 SMS Doesn't Save Reference Time When Importing Spectral Data

SRH-2D

- 7715 Model Check Not Running Before Exporting and/or Launching SRH-2D
- 7695 Can't use Intl Feet for SRH

SMS 11.2.8 - Release January 19, 2015

CMS

- 7656 CMS-Wave model control issues
- 7646 Can't set number of digits for CMS-Wave date/times
- 7644 cms flow save points cell id not correct when exporting

General

- 7665 Units/Projection Mixed-Up
- 7496 Dataset standard deviation is wrong
- 7621 View is modified and the z direction flips
- 7633 Displaying wrong mannings N value
- 7632 Monitor points not shown
- 7326 Freeze assigning BC
- 7617 Display projection set funny
- 7558 Crash Reading in BC File

Generic Model

- 7687 Material Properties not saved in Generic Model
- 7620 Extra "mesh" folder and 2dm file created
- 7592 Map files with generic model data not handled correctly

2D Grid

- 7622 Grid frames displayed double
- 7619 Z-mag for cgrid appears wrong

Map

- 7704 Build Polygon Command Alters Material Type for Some Existing Polygons

2D Mesh

- 7548 Merging Overlapping Meshes Causes Geometry Problems
- 7660 Nodestring Interal Error
- 7645 quadratic mesh nodes aren't flagged as midside when re-loaded in Sms

Raster

- 7618 Interpolate from Raster to Mesh Subset

RiverFlo-2D

- 7682 SMS assigns all mesh nodes same XYZ coordinate when running RiverFlo-2D with v11.2 template

- 7657 Can't run Generic Models RiverFlo-2D after loading solution

Scatter Data

- 7547 Create Mesh Quality Scatter Set Popup Item Causes SMS to Crash
- 7636 Can't save scatter file
- 7624 Merging Scatter Sets Crash
- 7625 Contours not updating after triangulation is changed
- 7641 scalar value not updated
- 7627 Scatter Merge priority lost when selecting maintain triangulation
- 7626 Crash when merging scatter sets
- 7635 Holes in scatter triangulation

SRH-2D

- 7695 Can't Use Intl Feet for SRH-2D
- 7467 Right-clicking on an SRH-2D coverage can give access to other coverage attributes

STWAVE

- 7668 STWAVE coordinate system not written out correctly

SMS 11.2.7 - Release Novemeber 13, 2014

DMI

- 7476 Crash when double-clicking in DMI coverage
- 7538 Crash when displaying materials on a duplicated mesh in DMI

General

- 7481 Projection not saved for a default coverage with no data
- 7485 Bad projection after prompt to reproject
- 7497 Projection error in log when the projection should just change
- 7509 Work in object projection function causes scatter set to disappear
- 7532 Flowtrace from single timestep of a transient dataset fails to generate animation
- 7533 Sample timesteps from a transient dataset in the Dataset Toolbox allows 0.0 time step resulting in infinite loop
- 7557 Opening project file (.sms) gives projection messages it shouldn't

Generic Model

- 7525 Generic Model BC Group Classifications not preserved
- 7526 Model specific data saved into the Generic Model Template
- 7527 Generic Model reuses the curves of a material type
- 7528 switching to generic model should also prompt for model (if more than 1)
- 7529 duplicating a generic model mesh should also duplicate its boundary conditions
- 7549 Generic Model Toggle writing out GP VAL wrong
- 7550 generic model multiple material groups not working

Map

- 7377 Offset arcs function not functioning correctly
- 7486 Rectangular Patch Crash
- 7487 Merging meshes creates overlapping elements
- 7563 SMS 11.2 Crash on showing polygon fill when coverage has no materials

2D Mesh

- 7442 Extract Weir Elevations for selected nodestring crashes and is unclear as to purpose
- 7517 Slow mesh merge
- 7519 loss of boundary conditions in mesh merge
- 7522 Crash Using Linear ↔ Quadratic on Merged Mesh
- 7530 should not allow for template deletion if it is in use by a mesh
- 7541 switch current model should always be enabled and new right-click command to create new mesh
- 7571 Rips occur when deleting nodes and retriangulating voids is on

SRH-2D

- 7477 SRH-2D *.srhmat file for a copied mesh(simulation) has no materials
- 7484 SRH material properties are scrambled when building polygons
- 7493 SRH-2D (Dynamic Model) display text not changing to specified settings
- 7554 SRH-2D material file is written wrong when material polygons with holes exist

TUFLOW

- 7218 TUFLOW AD BC values don't save

SMS 11.2.6 - Release October 3, 2014

ADCIRC

- 7437 ADCIRC File Save Error

BOUSS-2D

- 7414 Saving BOUSS2D Native Files Not WOrking for Second Simulation
- 7472 Reading a BOUSS2D simulation results in a bad damping cellstring

General

- 7334 Incorrect Checkbox Display in Project Explorer
- 7406 Nodestrings from Model Native Files are Corrupted
- 7415 Unable to Interpolate between Two Nodes
- 7407 H5 Gets Saved with Bad Connectivity
- 7445 Unable to Read in H5 File
- 7427 SMS files with no projection have mismatch problem
- 7433 Reproject all Does Not Change Display Projection
- 7428 If a New Project is Read in, The Projection is set before the old data is check on or cleared out
- 7439 Projection not set with "extra.h5" file when it should be
- 7379 Simulation Icon Problem

GIS

- 7479 Mapping GIS to Features takes hours

Map

- 7458 Cannot Reproject Grid Frame In Coverage

2D Mesh

- 7403 Merging Meshes Crash
- 7444 Nodes in Mesh that lie on Boundary of a Feature Arc are Selected When Boundary is Out Flag is on
- 7430 Mesh has odd projection

- 7460 Hang/Crash merging meshes

Raster

- 7474 Projection issues with raster

SRH-2D

- 7465 The arcs of an SRH-2D Materials coverage are not displayed
- 7470 Import SRHHyrdro File Doesn't Import Mannings N for Materials
- 7471 Import SRHHydro File Has Issues with Exporting and Boundary Conditions

TUFLOW

- 7332 SMS crashes when loading in TUFLOW .sup file and changing display options

SMS 11.2.5 - Release September 8, 2014

FESWMS

- 7270 FESWMS projection units not read in correctly

General

- 7366 SMS doesn't read the files associated with a B2D Project quite right
- 7375 Zoom to selection does not function as expected
- 7344 SMS saves out vector dataset in two locations
- 7347 Project Tree Missing Items after Reading in Project
- 7333 Crash opening Map file
- 7322 Open Project Crash

Generic Model

- 7339 Command to delete a generic model definition not in 11.2

Map

- 7327 SMS Polygon Attributes Dimmed when they should be active

2D Mesh

- 7329 Bad mesh boundary from Map → 2D Mesh
- 7398 Merge meshes results in bad mesh
- 7330 Can't merge meshes

Scatter Data

- 7368 Can't Triangulate Scatter Sets

SRH-2D

- 7338 Assigning materials from an SRH-2D materials coverage to an SRH-2D mesh allows bad option - crash
- 7337 SRH-2D time series curve file has extra lines that cause SRH-Pre to fail
- 7318 Dragging SRH coverage into a Map Module Folder hangs

SMS 11.2.4 - Release August 13, 2014

FESWMS

- 7271 Fixed Loading FESWMS solution corrupts ceiling dataset

General

- 7290 FixedDiscrepancies in the Wet Area displayed in the Echo Window and the Status Bar
- 7288 Fixed Selection Echo Window not displaying history
- 7286 Fixed duplicate datasets after running model

- 7280 Fixed a Time Series Issue

Map

- 7284 Fixed creating material property coverage activates additional coverage type
- 7325 Fixed a Crash when Editing Arc Attributes
- 7249 Fixed a Build Polygons Failure
- 7241 Fixed Interpolation for Map→2D Grid
- 7266 Fixed an Issue where Scatter Contour Display was not Updating when Deleting Triangles

2D Mesh

- 7289 Fixed Error "No mesh exists" when there is a mesh
- 7291 Fixed Nodestring Info not showing in the Selection Echo Window
- 7256 Fixed a Problem with Delete with all nodes selected on a Mesh
- 7265 Fixed the Distance between mesh nodes calculator

Scatter Data

- 7235 Fixed a Problem with Scatter Triangle and Contour Display not updating

SRH-2D

- 7202 Fixed a problem with an SRH Error finding .dat file
- 7212 Fixed an SRH-2d Pre fail when reading a rating curve boundary

SMS 11.2.3 - Release July 15, 2014

ADCIRC

1) Fixed a problem where ADCIRC elevation forcing time interval

CMS

1) Fixed a problem where CMS-WAVE Telescoping Grid were Unable to Read Vector Data

General

1) Fixed a problem where Contour label options are always labeled as meters
2) Fixed Unexpected End of File When Reading in Project
3) Fixed a Resample timestep error
4) Fixed a problem where Saving display projection in "save settings" doesn't always work
5) Fixed a problem where Opening the same *.atcf file two times does not create distinct coverages

2D Grid

1) Fixed Interpolating CGrid vector data to scatter

Images/GIS

1) Fixed a problem where Static Image not linked to the project file

PTM

1) Fixed a problem where Can't open large PTM *.h5 file

Scatter

1) Fixed a problem where Scatter→Mesh Activity not mapped

SRH-2D

1) Fixed SRH-2D Error in finding HDF5 file
2) Fixed a problem where Global renumber of SRH-2D ugrid scrambles material assignments

TUFLOW

1) Fixed a problem where TUFLOW AD BCs disappear

SMS 11.2.2 - Release June 02, 2014

This is a version release for SMS 11.2

CMS

1) Fixed a Problem where CMS-WAVE Telescoping Grid Unable to Read Vector Data
2) Fixed a Problem where Saving CMS-Wave Project Incomplete

General

1) Fixed a Problem where Contour label options are always labeled as meters
2) Fixed a Problem where Deleted disjoint nodes don't go away
3) Fixed a Crash when Extracting datasets in the Assign BC dialog
4) Fixed a issue where Dragging a project into SMS during start up causes problems

Images/GIS

1) Fixed a Problem where Static Image not linked to the project file

2D Mesh

1) Fixed a Crash when clicking 'OK' in Rectangular Patch Options
2) Fixed a Problem where No Materials Data dialog for generic mesh elements. (11.2 32 bit)
3) Fixed Mesh → Scatter Crash

SRH-2D

1) Fixed a Problem where SRH-2D Error in finding HDF5 file

TUFLOW

1) Fixed TUFLOW Active/Inactive Cell Display Confusion
2) Fixed a Problem with cell-by-cell activity in TUFLOW

Older Bugfixes

For older versions of SMS that are no longer actively supported, see the individual bugfix articles:

Bugfix Article	Release Notes
Bugfixes SMS 10.0	SMS 10.0 Intermediate Release
Bugfixes SMS 10.1	SMS 10.1 Intermediate Release
Bugfixes SMS 11.0	SMS 11.0 Intermediate Release
Bugfixes SMS 11.1	SMS 11.1 Intermediate Release

Related Topics

- What's New in SMS
- Downloads
- Installing and Setting up SMS
- System Requirements
- License Agreement

FHWA:2010 Webinars

Federal Highways Administration and Aquaveo have partnered to present a series of webinars on using SMS/WMS. This page contains links to past webinars that can be watched.

January: Introduction to WMS: How to Build a Model using the Hydrologic Modeling Wizard

Slide Presentation (50 minutes)

Software Demonstration (20 minutes – please note that the sound for this recording will take a few seconds to come up.)

February: Introduction to SMS models FESWMS and TUFLOW

Watch (45 minutes)

March: Using coordinate systems in SMS and WMS

Watch (50 minutes)

April: Data Collection in SMS

Watch (53 minutes)

May: Data Collection in WMS

Watch (1 hr 8 min)

June: Editing Scattered Data in SMS

Watch (47 min)

July: Using the Hydrologic Modeling Wizard in WMS

Watch (55 min)

August: Flow Modeling using SMS and TUFLOW

Watch (1 hr 27 min)

September: HEC-RAS Conceptual Modeling in WMS

Watch (1 hr)

October: Setting up FESWMS Models in SMS

Watch (1 hr)

November: Mapping Floodplains in WMS

Watch (1 hr)

December: FESWMS(and TUFLOW), Modeling structures in 1D and 2D

Watch

Back to XMS

Dialog Help

This is a special page that relates SMS dialogs to wiki pages. SMS reads this page when a user hits the *Help* button in a SMS dialog, and opens the wiki at the page indicated below. Blank Dialog IDs use uniquely generated numbers because the dialog is derived or shared.

Wiki Page | [**Dialog Number** | **Dialog ID**][**QDlgClass**]

1) Color Options | 101 | IDD_COLOROPTIONS
2) Find 2D Mesh Node Dialog | 102 | IDD_FINDNODE
3) Mesh Module Display Options | 103 | IDD_MESHDISPLAYOPTIONS
4) Scatter Module Display Options | 104 | IDD_SCATDISPOPTS
5) CGWAVE BC Nodestrings | 105 | IDD_DISPOPTS_CGNSTR
6) Mesh Module Information | 106 | IDD_MESHINFO
7) IDD_ASKME | 107 | IDD_ASKME
8) RMA2 Material Properties | 108 | IDD_RMA2_ROUGHNESSBYDEPTH
9) Curvilinear Grid Display Options | 109 | IDD_POINT_DISPLAY_ATTS
10) Mesh Quality | 110 | IDD_ELEMENTQUALITY
11) RMA2 Boundary Conditions | 111 | IDD_DISPOPTS_R2NSTR
12) Nodal Boundary Conditions | 112 | IDD_RMA2_NODEBC
13) Materials Data | 113 | IDD_MATERIALS
14) New Palette | 114 | IDD_NEWPALETTE
15) 2D Mesh Polygon Properties | 115 | IDD_DISPOPTS_FENODE
16) Display Options | 116 | IDD_SYMBOL_DISPLAY_ATTS
17) FESWMS BC Nodestring Display Options | 117 | IDD_DISPOPTS_FENSTR
18) RMA2 Boundary Conditions | 118 | IDD_DISPOPTS_R2NODE
19) ADH Material Properties | 119 | IDD_ADH_MP_PROPS
20) ADH Material Properties | 120 | IDD_ADH_MP_REFINE_AND_TRANS
21) Filtering | 121 | IDD_CSATTSPROPFILTER
22) ADH Model Control Global Material Properties | 122 | IDD_ADH_MC_MATERIALS
23) ADH Model Control | 123 | IDD_ADH_MC
24) TABS Attribute Dialog | 124 | IDD_RMA2_GWNCARD
25) IDD_CHOICE | 125 | IDD_CHOICE
26) Mesh to Map | 127 | IDD_MESH_TO_MAP
27) BOUSS-2D Probe Solutions | 128 | IDD_BOUSS2D_UTILITIES
28) Redistribute Vertices | 129 | IDD_REDISTVERT
29) IDD_CS_CATEGORYTYPES | 130 | IDD_CS_CATEGORYTYPES
30) Shapefiles | 132 | IDD_CLASSIFY_MAPPING
31) Editing Cross Sections | 133 | IDD_CSATTS
32) Merging | 134 | IDD_CSATTSPROPMERGE
33) FESWMS Model Control Dialog | 135 | IDD_FESWMSGATE
34) Geo-Referencing | 136 | IDD_CSATTSPROPGEOEDIT
35) Geo-Referencing | 137 | IDD_CSATTSPROPGEOREF
36) Line Properties | 138 | IDD_CSATTSPROPLINE
37) Point Properties | 140 | IDD_CSATTSPROPPOINT
38) Observation Coverage | 141 | IDD_OBSERVE
39) TUFLOW Boundary Conditions | 142 | IDD_TUFLOW_BC2D_POLYATTS
40) GIS Module Menus | 143 | IDD_ARCGISSELECTBYLOCATION
41) GIS Module Menus | 144 | IDD_ARCGISSELECTIONOPTIONS
42) GIS Module Menus | 145 | IDD_ARCGISTRANSPARENCY
43) Vector Display Options | 146 | IDD_VECTORDISPOPTS

44) Calibration Display Options | 147 | IDD_FEATDISPOPTS_OBSERVE
45) Mesh Module Display Options | 148 | IDD_FEATDISPOPTS_MESH
46) Plot Window Right-Click Menu | 149 | IDD_AXISTITLES
47) IDD_MODELWRAPPER | 150 | IDD_MODELWRAPPER
48) View Data File | 151 | IDD_CHOOSETEXTEDITOR
49) Display Options | 153 | IDD_LEGENDOPTIONS
50) ADCIRC | 154 | IDD_EXTRACTCOASTLINE
51) IDD_FEATDISPOPTS_WAVE | 155 | IDD_FEATDISPOPTS_WAVE
52) Tidal/Harmonics Tab | 156 | IDD_TIDAL_ELLIPSES
53) Cartesian Grid Module Display Options | 157 | IDD_FEATDISPOPTS_CGRID
54) Generic 2D Nodal BC, Nodestring and Element Display Options | 158 | IDD_FEATDISPOPTS_GENERIC
55) ADCIRC Boundary Conditions | 161 | IDD_DISPOPTS_ADNSTR
56) Info Options | 162 | IDD_SELECTIONECHO
57) Functional Surfaces | 164 | IDD_FSURFACEOPTS
58) Film Loop Drogue Plot Options | 166 | IDD_FILMLOOP_DROGUES
59) Film Loop Display Options | 167 | IDD_FILMLOOP_DISPLAY
60) Film Loop Flow Trace Options | 168 | IDD_FILMLOOP_FWTRACE
61) Film Loop General Options | 170 | IDD_FILMLOOP_GENERAL
62) Cartesian Grid Module Display Options | 171 | IDD_CGRIDDISPOPTS
63) File Import Wizard | 182 | IDD_TXTWIZ1
64) File Import Wizard | 183 | IDD_TXTWIZ2
65) RMA4 BC Nodestrings | 184 | IDD_RMA4_NODESTRBC 184
66) Mesh Module Display Options | 185 | IDD_DISPOPTS_ADNODE
67) Film_Loop_Multiple_Views | 186 | IDD_FILMLOOP_ADDVIEW
68) Wave Gages | 187 | IDD_GENESIS_WAVELIST
69) IDD_TAB_BIN_DEFINE | 188 | IDD_TAB_BIN_DEFINE
70) Mesh Module Display Options | 189 | IDD_DISPOPTS_2DM_ENTITY
71) Profile Customization Dialog | 190 | IDD_ASSIGNPROFILE
72) Scatter Options | 191 | IDD_SCATTEROPTIONS
73) RMA2 1D Control Structures | 192 | IDB_RMA2_NEW1DGEO
74) Time Settings Options | 193 | IDD_PLOTWIZ2_TIMESERIES
75) File Menu | 195 | IDD_SELECTIONINFOOPTS
76) IDD_TAB_FILTERS | 201 | IDD_TAB_FILTERS
77) IDD_TAB_STATION | 203 | IDD_TAB_STATION
78) IDD_FPROP | 204 | IDD_FPROP
79) GenCade Graphical Interface | 205 | IDD_GENESIS_CHOOSEGRID
80) CGWAVE Math Details: Boundary Conditions | 206 | IDD_CGWAVE_SOL_OPTS
81) Cartesian Grid Module | 207 | IDD_CGRIDINFO
82) Scatter Module | 208 | IDD_SCATTERMODULEINFO
83) Map Module | 209 | IDD_MAPMODULEINFO
84) Import Spectra | 212 | IDD_SPECTRAL_IMPORT
85) BOUSS-2D Probes | 213 | IDD_BOUSS2D_PROBES

86) IDD_TWO_COLUMN_PROPERTIE | 215 | IDD_TWO_COLUMN_PROPERTIE
87) Feature Stamping | 218 | IDD_STAMP_COVER
88) Arc Attributes Dialog | 219 | IDD_MAPPING_ARCATTS
89) Mesh to Scatterpoint | 220 | IDD_MESH_TO_SCAT
90) Generic Model Graphical Interface | 221 | IDD_GEN_ASSIGNPARAM
91) Generic Model Graphical Interface | 224 | IDD_GEN_DEFINE_BC_TAB
92) 2D Mesh Polygon Properties | 226 | IDD_2DMESH_POLY_PROP
93) IDD_RMA2_JUNCTION | 228 | IDD_RMA2_JUNCTION
94) Particle Module Display Options | 229 | IDD_PARTICLEDISPOPTS
95) RMA4 Material Properties | 230 | IDD_RMA4_MATERIALPROPERTIES1
96) Particle Module Create Datasets | 231 | IDD_PARTICLE_DATASETS
97) Particle Report | 232 | IDD_PARTICLEREPORT
98) Merge Scatter Sets | 233 | IDD_SCATTER_MERGE_SCATSETS
99) Data Calculator | 235 | IDD_DATA_CALCULATOR
100) Tool Right-Click Menus | 236 | IDD_SELECT_FEATURE_ARC
101) Projections | 237 | IDD_COORD_WIZ
102) Projections | 238 | IDD_COORD_SYSTEM
103) Feature Stamping | 239 | IDD_STAMP_ARC
104) IDD_PROMPT_FLT | 240 | IDD_PROMPT_FLT
105) FESWMS Material Properties | 241 | IDD_FESWMS_ROUGHNESS
106) FESWMS Model Control Dialog | 242 | IDD_FESWMS_WAVEPARAMS
107) FESWMS Material Properties | 243 | IDD_FESWMS_TURBULENCE
108) Registering an Image | 244 | IDD_REGIMAGE_GM2
109) XY Series Editor | 245 | IDD_XYSERIES
110) Feature Stamping | 246 | IDD_STAMP_WIDTH_SLOPE
111) IDD_FATE_LAYERMATS | 247 | IDD_FATE_LAYERMATS
112) FESWMS Sediment Control | 248 | IDD_FESWMS_SED_PARAMS
113) Timing Tab | 249 | IDD_FESWMS_ITERATIONS
114) Feature Stamping | 250 | IDD_STAMP_POINT
115) FATE | 251 | IDD_FATE_FILEINDATE
116) Feature Stamping | 252 | IDD_STAMP_FEATURES
117) FESWMS Hydraulic Structures | 253 | IDD_DISP_CTRLSTRUCT_OPTS
118) FESWMS Sediment Control | 256 | IDD_FESWMS_SED_CONTROL
119) FESWMS BC Nodestrings | 258 | IDD_FESWMS_BC_STRING
120) What Is SMS | 259 | IDD_ABOUT
121) Sediment Boundary Conditions | 260 | IDD_FESWMS_BC_SED
122) General | 261 | IDD_PREFS_GENERAL
123) Toolbars | 262 | IDD_PREFS_TOOLBARS
124) Preferences | 263 | IDD_PREFS_MODELS
125) Images | 264 | IDD_PREFS_IMAGES
126) Project Explorer | 265 | IDD_PREFS_PROJECT_EXPLORER
127) ADCIRC Boundary Types | 266 | IDD_ADCIRC_PERIODIC_NFLOW
128) Grid Frame Properties | 267 | IDD_CGRID_GRIDFRAME_DLG

172) Model Checker | 368 | IDD_GRID_CHECK_ISSUES
173) IDD_PRINT_SCREEN_BLOCK | 371 | IDD_PRINT_SCREEN_BLOCK
174) IDD_EXTRUDE_2D_MESH | 373 | IDD_EXTRUDE_2D_MESH
175) ADCIRC Model Control | 374 | IDD_ADCIRC_3DPARAMS
176) Particle Module Compute Grid Datasets | 375 | IDD_PARTICLE_COMPUTE_GRID_DATASETS
177) IDD_EXTRUDED_2DMESH_DISPLAYOPTIONS | 376 | IDD_EXTRUDED_2DMESH_DISPLAYOPTIONS
178) Time Step Window | 377 | IDD_TIMESTEP_PICKER
179) ADCIRC Model Control | 378 | IDD_ADCIRC_BAROCLINIC_COEFFICIENTS
180) Particle Module Compute Grid Datasets | 379 | IDD_PARTICLE_COMPUTE_GRID_DATASETS_TIME
181) Display Options | 385 | IDD_DISPLAY_OPTIONS
182) PTM Sediment File | 387 | IDD_PTM_SELECT_XMDF_PATH
183) Contour Options | 388 | IDD_CONTOURS_SPECIFIC
184) TUFLOW Boundary Conditions | 390 | IDD_TUFLOW_BC
185) TUFLOW Boundary Conditions | 391 | IDD_TUFLOW_BC_EVENTS
186) ADH Model Control Output | 392 | IDD_ADH_MC_OUTPUT
187) TUFLOW 2D Flow Constriction Shape Coverage | 399 | IDD_TUFLOW_FLOW_CONSTRICTION_PROPS
188) Dredge Source Model | 400 | IDD_DREDGE_EQUIPMENT
189) MK Placement | 401 | IDD_FATE_MK_PLACEMENT
190) FATE | 402 | IDD_FATE_COMP_REPORTS
191) IDD_DREDGE_ATTS_PAGE | 403 | IDD_DREDGE_ATTS_PAGE
192) ADH Hot Start File | 404 | IDD_ADH_HOTSTART
193) IDD_FATE_GETIJ | 405 | IDD_FATE_GETIJ
194) Dredging Coverage | 406 | IDD_DREDGE_COV_OPTIONS
195) Dredging Sediment Characteristics | 407 | IDD_DREDGE_GENERATE_PTM
196) IDD_DREDGE_ARC_ATTS | 408 | IDD_DREDGE_ARC_ATTS
197) IDD_DREDGE_POLY_ATTS | 410 | IDD_DREDGE_POLY_ATTS
198) Dredging Sediment Characteristics | 411 | IDD_DREDGE_SEDIMENTS
199) STFATE Clouds to PTM Sources | 412 | IDD_FATE_CLOUD_MASTER
200) Dredge Source Model | 413 | IDD_DREDGE_CALCPRODRATE_MECH
201) IDD_DREDGE_CALCDELTATC | 414 | IDD_DREDGE_CALCDELTATC
202) Dredge Source Model | 415 | IDD_DREDGE_CALCPRODRATE_HYD
203) IDD_DREDGE_CALC_SLURRY_CONC | 416 | IDD_DREDGE_CALC_SLURRY_CONC
204) Time Series | 417 | IDD_TIME_SERIES
205) Compass Plot | 420 | IDD_COMPASSPLOT_PROPERTIES
206) Grid Frame Properties | 422 | IDD_CGRID_GRIDFRAME
207) Map → 2D Grid | 423 | IDD_CGRID_MAP_TO_GRID
208) Steering | 424 | IDD_STEERWIZ_ADCIRC_CMSWAVE
209) RMA2 Spindown | 426 | IDD_STEERWIZ_SPINDOWN
210) FESWMS Model Control Dialog | 427 | IDD_FESWMS_CEILING
211) ADH Bed Layers Assignment | 428 | IDD_ADH_BED_LAYERS
212) Annotation Objects | 430 | IDD_ANNONEWANCHORINGDIALOG

256) RMA2 Model Control Dialog | 479 | IDD_RMA2_MODELCONTROL1
257) RMA2 Model Control Dialog | 480 | IDD_RMA2_MODELCONTROL3
258) Line/Arrows | 481 | IDD_LINEARROW_ATTS
259) Annotations | 482 | IDD_RECTOVAL_ATTS
260) IDD_FEATDISPOPTS_RIVHYD | 483 | IDD_FEATDISPOPTS_RIVHYD
261) GenCade Model Control Dialog | 484 | IDD_GENESIS_MODEL4
262) GenCade Model Control Dialog | 485 | IDD_GENESIS_MODEL1
263) GenCade Structures | 486 | IDD_GENESIS_SPDS
264) GenCade Model Control Dialog | 487 | IDD_GENESIS_MODEL2
265) Grid Frame Properties | 488 | IDD_1DGRIDFRAME
266) GenCade Structures | 489 | IDD_GENESIS_OFFSETGRID
267) 1D River Hydraulics Data Browser | 490 | IDD_RIVHYD_DATABROWSER
268) IDD_REPORT | 491 | IDD_REPORT
269) GenCade Arc Attributes | 492 | IDD_GENESIS_ARCATTS
270) Film Loop Multiple Views | 493 | IDD_FILMLOOP_VIEWS
271) Cartesian Grid Find Cell | 494 | IDD_FIND_CELL
272) FESWMS Spindown | 495 | IDD_SPINDOWN_STEERING
273) IDD_STEERING_ADST | 496 | IDD_STEERING_ADST
274) ADH Extract WSE | 497 | IDD_EXTRACTBC
275) Wind Tab | 499 | IDD_ADCIRC_FLEET_WIND
276) Wind Tab | 500 | IDD_ADCIRC_BIN_WIND
277) ADCIRC Model Control | 503 | IDD_WETTING_DRYING
278) Display Options | 504 | IDD_GENERALLEGEND
279) Scale Bars | 505 | IDD_SCALEBAROPTIONS
280) North Arrows | 506 | IDD_NORTHARROW
281) Film Loop Display Options | 507 | IDD_CLOCKOPTIONS
282) Contour Options | 508 | IDD_CONTOURS_GENERAL
283) IDD_OCEAN_CONDITIONS | 509 | IDD_OCEAN_CONDITIONS
284) Plot Window | 510 | IDD_PLOT_VIEWVALUES
285) Contour Legend Options | 511 | IDD_CONTOURLEGEND
286) IDD_AUTOMATICLABELOPTIONS | 512 | IDD_AUTOMATICLABELOPTIONS
287) 1D Grid Display Options | 513 | IDD_ONEDGRID_DISPOPTS
288) IDD_SPD_PERMUTATIONS | 514 | IDD_SPD_PERMUTATIONS
289) Get Online Maps | 515 | IDD_GET_DATASET
290) Arc Attrbutes | 516 | IDD_CGRID_ARC_OPTS
291) Grid Frame Properties | 518 | IDD_GRID_FRAME_PROP_GENERAL
292) IDD_TIMESTEP | 519 | IDD_TIMESTEP
293) RMA2 Revisions | 521 | IDD_RMA2REV
294) Generic Model Graphical Interface | 523 | IDD_GENERIC_MODEL_WRAPPER
295) RMA2 Material Properties | 526 | IDD_MATERIAL_PROPERTIES
296) RMA2 Material Properties | 527 | IDD_RMA2MAT_TAB1
297) RMA2 Material Properties | 528 | IDD_RMA2MAT_TAB2
298) RMA2 Material Properties | 529 | IDD_RMA2MAT_TAB3

342) Scalar to Vector | 580 | IDD_DATA_SCAL_TO_VEC
343) Projections | 581 | IDD_COORD_CONVERT
344) Projections | 582 | IDD_COORDS
345) Data Transform | 583 | IDD_COORD_TRANSFORMATION
346) IDC_TXT_NC1 | 584 | IDC_TXT_NC1
347) Projections | 585 | IDD_COORD_SINGLEPT
348) Vector to Scalar | 586 | IDD_DATA_VEC_TO_SCAL_COPY
349) Map Module Display Options | 587 | IDD_FEATDISPOPTS_MAPPING
350) Weir | 588 | IDD_FESWMS_GLOBALPIER
351) Mapping Coverage | 589 | IDD_MAPPING_POLYATTS
352) Mapping Coverage | 590 | IDD_MAPPING_OPTIONS
353) 2D Mesh Node Options Dialog | 591 | IDD_NODESTRING_OPTS
354) Registering SMS | 593 | IDD_REGISTER
355) IDD_SECURITY_STRING | 594 | IDD_SECURITY_STRING
356) IDD_NETWORK_SETTINGS | 595 | IDD_NETWORK_SETTINGS
357) Weir | 596 | IDD_FESWMS_PIER_DEFINITION
358) IDD_NEW_TABLE_COLUMN | 597 | IDD_NEW_TABLE_COLUMN
359) View | 598 | IDD_EDIT_VIEW
360) Generic Model | 599 | IDD_GENERIC_GLOBAL_PARAMETERS
361) Area Property Polygon Attributes Dialog | 600 | IDD_LAND_OCEAN_POLYATTS
362) IDD_GENERIC_PARAMDEF | 601 | IDD_GENERIC_PARAMDEF
363) Cartesian Grid Data Menu | 602 | IDD_GRID_TO_MAP
364) IDD_GENGPARAM_TYPEOPTS | 603 | IDD_GENGPARAM_TYPEOPTS
365) IDD_EXDATASET | 604 | IDD_EXDATASET
366) GIS Conversion and Editing | 606 | IDD_DEM_IMPORT
367) GIS Conversion and Editing | 607 | IDD_DEM_SMOOTHING_OPTS
368) Feature Object Modification: All | 608 | IDD_CLEAN_FEATURE_OBJECTS
369) Refine Point Dialog | 609 | IDD_MAP_REFINEATTS
370) Material Properties | 610 | IDD_GENERIC_MATERIALPROPS
371) Feature Object Commands | 611 | IDD_DOMAIN_OPTIONS
372) Area Property Polygon Attributes Dialog | 612 | IDD_LAND_POLYATTS
373) Shapefiles | 613 | IDD_EXPORT_SHAPEFILE
374) Generic Model Graphical Interface | 614 | IDD_GEN_DEFINEMODEL
375) CGWAVE Boundary Conditions Dialog | 615 | IDD_CGWAVE_BC
376) Generic Model Graphical Interface | 616 | IDD_GEN_ASSIGNBC
377) Refine Attributes Dialog | 618 | IDD_ADCIRC_PT_ATTS
378) Select By Dataset Value | 619 | IDD_FUNCTION_VALUE
379) Rectangular Patch | 620 | IDD_MESH_RECTANGULAR_PATCH
380) RMA2 Model Control Dialog | 621 | IDD_RMA2_CONTROL
381) Managing Cross Sections | 622 | IDD_RIVHYD_XSECTION_ATTS
382) Grid Smoothing | 623 | IDD_CGRID_SMOOTHINGOPTS
383) FATE | 624 | IDD_FATE_MATERIAL_PROPERTIES
384) Triangular Patch | 625 | IDD_TRIPATCH_OPTS

385) PTM Model Control | 626 | IDD_PTM_CREATE_INPUT_FILES
386) Grid Frame Properties | 627 | IDD_GRIDFRAME
387) FATE | 628 | IDD_FATE_VESSEL_LOADS
388) Smooth Dataset | 629 | IDD_DATA_SMOOTHING
389) ADCIRC Model Control | 630 | IDD_ADCIRC_MODELCONTROL_GENERAL
390) Files Tab | 631 | IDD_ADCIRC_MODELCONTROL_FILES
391) Tidal/Harmonics Tab | 632 | IDD_ADCIRC_MODELCONTROL_TIDAL_HARMONIC
392) Wind Tab | 633 | IDD_ADCIRC_MODELCONTROL_WIND
393) FATE | 634 | IDD_FATE_GENERAL
394) IDD_FATE_COEFFICIENTS | 635 | IDD_FATE_COEFFICIENTS
395) Sediment Options Tab | 636 | IDD_ADCIRC_MODELCONTROL_SEDIMENT
396) IDD_WAVE_CALCULATOR | 637 | IDD_WAVE_CALCULATOR
397) Files Tab | 638 | IDD_ADCIRC_RADIATION_STRESS_FILES
398) Tidal/Harmonics Tab | 639 | IDD_ADCIRC_NEW_HARMONIC_ANALYSIS
399) ADCIRC Boundary Conditions | 640 | IDD_ADCIRC_ARC_ATTRIBUTES
400) ADCIRC Boundary Types | 641 | IDD_ADCIRC_NORMAL_FLOW_PARAMATERS
401) IDD_TUFLOWNTWK_DIALOG | 642 | IDD_TUFLOWNTWK_DIALOG
402) IDD_TUFLOW_CULVWEIR | 643 | IDD_TUFLOW_CULVWEIR
403) BOUSS-2D Probes | 644 | IDD_BOUSS2D_PROBE_OPTIONS
404) FATE | 645 | IDD_FATE_VESSELS
405) FATE | 646 | IDD_FATE_DENSITY
406) IDD_FATE_TRACER_CONSTITUENT | 647 | IDD_FATE_TRACER_CONSTITUENT
407) IDD_FATE_OUTPUT | 648 | IDD_FATE_OUTPUT
408) Timing Tab | 649 | IDD_ADCIRC_MODELCONTROL_TIME
409) IDD_FATE_PLACEMENTS | 650 | IDD_FATE_PLACEMENTS
410) BOUSS Runup / Overtopping | 651 | IDD_OVERTOP_CALCULATOR
411) IDD_FATE_CURRENT | 652 | IDD_FATE_CURRENT
412) IDD_FATE_SPREADSHEET | 653 | IDD_FATE_SPREADSHEET
413) BOUSS-2D Probe Solutions | 654 | IDD_SPEC_PLOT
414) 2D Mesh Nodes Menu | 655 | IDD_NODE_INTERP_OPTS
415) CGWAVE Model Control | 656 | IDD_CGWAVE_MODEL_CONTROL
416) PTM Feature Point Attributes Dialog | 657 | IDD_PTM_POINT_ATTS
417) IDD_TUFLOW_WEIR1D | 658 | IDD_TUFLOW_WEIR1D
418) PTM Polygon Attributes Dialog | 659 | IDD_PTM_LINE_OR_POLY_ATTS
419) FESWMS Sediment Control | 663 | IDD_FESWMS_SED_LOCALBED
420) File Import Filter Options | 664 | IDD_FILE_IMPORT_FILTER_OPTS
421) Export Tabular File | 665 | IDD_METAFILE_OPTS
422) IDD_FESWMS_PRINT | 666 | IDD_FESWMS_PRINT
423) FESWMS Spindown | 667 | IDD_FESWMS_IC
424) FESWMS Model Control Dialog | 668 | IDD_FESWMS_SED_VOL_FLOW
425) Recording Stations | 670 | IDD_ADCIRC_RECORDING_STATIONS
426) Select/Delete Data... | 672 | IDD_MAP_SELECT_DELETE_WITH_POLY
427) Dataset Right-Click Menus | 673 | IDD_METADATA_DATASET

428) SMS | 675 | IDB_SMS_LOGO
429) IDB_ARROW | 676 | IDB_ARROW
430) Model Checker | 677 | IDD_QUICK_CHECK
431) Switch Current Model | 678 | IDD_CURRENT_MODEL
432) FESWMS Sediment Control | 679 | IDD_FESWMS_GLOBAL_BED
433) FESWMS Sediment Control | 680 | IDD_FESWMS_LOCAL_PARAMS
434) TUFLOW Menu | 681 | IDD_TUFLOW_CELL_ATTS
435) Export Tabular File | 682 | IDD_EXPORT_TABULAR_FILE
436) Exporting Profile Dialog | 683 | IDD_FORMAT_COLUMN_DATA
437) FESWMS BC Nodestrings | 684 | IDD_FESWMS_BC_NODE
438) Create Datasets | 685 | IDD_CREATE_DATA_SETS
439) Parameters Tab | 686 | IDD_FESWMS_PARAMETERS
440) IDD_AUTOSAVE | 687 | IDD_AUTOSAVE
441) Q&A ADCIRC | 688 | IDD_ADCIRC_FLOW_FROM_HYDROGRAPH
442) Weir | 689 | IDD_FESWMS_WEIR
443) Time Settings | 690 | IDD_DATASET_TIME_INFO
444) Time Settings | 691 | IDD_GLOBAL_TIME_SETTINGS
445) FESWMS | 692 | IDD_FESWMS_GENERAL
446) Map to 2D Scatter Points | 693 | IDD_MAP_TO_SCAT
447) FESWMS Model Control Dialog | 694 | IDD_FESWMS_CHLINK
448) TABS Attribute Dialog | 695 | IDD_RMA2_ARC_ATTS
449) FESWMS Arc Attributes Dialog | 696 | IDD_FESWMS_ARC_ATTS
450) TABS Attribute Dialog | 697 | IDD_RMA2_POINT_ATTS
451) FESWMS Point Attributes Dialog | 698 | IDD_FESWMS_POINT_ATTS
452) Generic 2D Nodal BC, Nodestring and Element Display Options | 699 | IDD_GEN2DM_POINT_ATTS
453) ADCIRC Weirs and Island Barriers | 700 | IDD_ADCIRC_BARRIER_WEIR
454) IDD_WEBSERVICE_GETTOKEN | 701 | IDD_WEBSERVICE_GETTOKEN
455) STWAVE Boundary Conditions | 702 | IDD_SPECTRAL_EVENTS
456) Registering an Image | 703 | IDD_EDITIMAGE_GM
457) Output Control | 704 | IDD_TUFLOW_OUTPUTCONTROL
458) Time | 705 | IDD_TUFLOW_TIME_CONTROL
459) Wetting/Drying | 706 | IDD_TUFLOW_WETDRY_CONTROL
460) Restart Files | 707 | IDD_TUFLOW_RESTART_CONTROL
461) Water Level | 708 | IDD_TUFLOW_WATERLEVEL_CONTROL
462) ADH Material Properties | 709 | IDD_ADH_MP_BC
463) ADH Model Control Model Parameters | 710 | IDD_ADH_MC_PARAMETERS
464) TUFLOW Coverages | 711 | IDD_TUFLOW_CREATE_CS_ARCS
465) Arc Properties Dialog | 712 | IDD_TUFLOW_1D_2D_ARC_ATTS
466) TUFLOW Coverages | 713 | IDD_FEATDISPOPTS_TUFLOW_NETWORK
467) TUFLOW Coverages | 714 | IDD_TUFLOW_CREATE_WLL_ARCS
468) TUFLOW Network Node SX Additions | 715 | IDD_TUFLOW_NTWK_NODE_ATTS
469) General | 721 | IDD_TUFLOW_CTRL1D_GENERAL
470) Network | 722 | IDD_TUFLOW_CTRL1D_NETWORK

471) TUFLOW Clean Options | 723 | IDD_TUFLOW_CLEAN_FEAT
472) TUFLOW Grid Options | 726 | IDD_TUFLOW_GRID_OPTIONS
473) TUFLOW Boundary Conditions | 727 | IDD_TUFLOW_BC_CONTROL
474) TUFLOW Material Properties | 728 | IDD_TUFLOW_MATERIAL_CONTROL
475) TUFLOW Coverages | 729 | IDD_TUFLOW_GEOM_MOD_COV_OPTS
476) Misc | 730 | IDD_TUFLOW_MISC_CONTROL
477) Long Wave Input Toolbox | 731 | IDD_CGWAVE_LONG_WAVE_TOOLBOX
478) GIS Module Display Options | 732 | IDD_GISDISPOPTS
479) IDD_MATERIAL_LAYERS | 733 | IDD_MATERIAL_LAYERS
480) Polygons | 734 | IDD_ELEMENT_MAT_ZONES
481) Grid Nesting | 735 | IDD_STWAVE_NEST_GRIDS
482) Generic 2D Nodal BC, Nodestring and Element Display Options | 736 | IDD_GEN2DM_ARC_ATTS
483) Extracting Cross Sections | 737 | IDD_EXTRACTCS
484) ADCIRC Boundary Conditions | 738 | IDD_ADCIRC_NODALATTS
485) PTM Model Control | 739 | IDD_PTM_MODELCONTROL_FILES
486) PTM Model Control | 740 | IDD_PTM_MODELCONTROL_TIME
487) PTM Model Control | 741 | IDD_PTM_MODELCONTROL_COMPUTATIONS
488) PTM Model Control | 742 | IDD_PTM_MODELCONTROL_OUTPUT
489) PTM Model Control | 743 | IDD_PTM_MODELCONTROL_WAVES
490) Particle Module Compute Grid Datasets | 744 | IDD_PARTICLE_COMPUTE_GRID_DATASETS_BINS
491) ADH Material Properties | 747 | IDD_ADH_MP_EMPTY
492) Smooth Arc (Right-click Menu)|Smooth Arc (Right-click Menu) | 748 | IDD_ARC_SMOOTH_TOOL
493) Generic Model Graphical Interface | 749 | IDD_GENERIC_MODEL_PARAMETERS
494) Dredging Sediment Characteristics | 750 | IDD_DREDGE_SED_PROP
495) Dredging Sediment Characteristics | 751 | IDD_DREDGE_SED_SS
496) IDD_DREDGE_EMPTY | 752 | IDD_DREDGE_EMPTY
497) Annotation Objects | 753 | IDD_ANNOANCHORINGDIALOG
498) ADH Boundary Condition Assignment | 754 | IDD_ADH_BC_FLOW_NSTR
499) ADH Boundary Condition Assignment | 755 | IDD_ADH_BC_FLOW_NODE
500) ADH Boundary Condition Assignment | 756 | IDD_ADH_BC_PRESSURE
501) ADH Boundary Condition Assignment | 757 | IDD_ADH_BC_TRANSPORT
502) ADH Boundary Condition Assignment | 758 | IDD_ADH_BC
503) Curvilinear Grid Display Options | 759 | IDD_CURVDISPOPTS
504) RMA4 Element Loading | 760 | IDD_RMA4_ELEM_MASS_LOAD
505) IDD_DISPOPTS_R2ELEM | 761 | IDD_DISPOPTS_R2ELEM
506) CMS-Wave Cell Attributes Dialog | 762 | IDD_CMSWAVE_CELLATTS
507) Converting Coverages | 763 | IDD_ARC_EXTRACT_TOOL
508) Dataset Toolbox | 764 | IDD_DST_TIME_SAMPLE
509) Dataset Toolbox | 765 | IDD_DST_COASTAL_WAVELENGTH_CELERITY
510) Dataset Toolbox | 766 | IDD_DST_DERIVATIVE
511) Dataset Toolbox | 767 | IDD_DST_ACTIVITY
512) Dataset Toolbox | 768 | IDD_DST_SCALAR_TO_VECTOR
513) Dataset Toolbox | 769 | IDD_DST_VECTOR_TO_SCALAR

514) Dataset Toolbox | 770 | IDD_DST_COASTAL_GRAVITY_WAVES
515) Dataset Toolbox | 771 | IDD_DST_COASTAL_ADVECTIVE
516) Dataset Toolbox | 772 | IDD_DST_FILTER
517) Dataset Toolbox | 773 | IDD_DST_GEOMETRY
518) IDD_NOGEOREFINFO | 774 | IDD_NOGEOREFINFO
519) Scatter Breakline Options | 777 | IDD_IMPORT_BREAKLINE
520) Offset Arc (Right-click Menu) | 779 | IDD_ARC_OFFSET_TOOL
521) PBL | 780 | IDD_PBL_MODELCONTROL
522) Hurricane Path Pertubations Dialog | 781 | IDD_ADCIRC_PERTURBATIONS
523) Output | 782 | IDD_EFDC_MC_OUTPUT
524) Geometry | 783 | IDD_EFDC_MC_GEOMETRY
525) General | 784 | IDD_EFDC_MC_GENERAL
526) Time | 785 | IDD_EFDC_MC_TIME
527) EFDC Model Control Advanced Cards | 786 | IDD_EFDC_MC_ADVANCED_CARDS
528) PTM Particle Filters | 789 | IDD_PARTICLE_FILTER_OPTS
529) TM Particle Filters | 790 | IDD_IMPORT_HURDAT
530) Dataset Toolbox | 793 | IDD_EDITABLE_DATASETS
531) Spatial Inputs Tab | 795 | IDD_WAM_MC_GRID_SPATIAL
532) TUFLOW AD | 796 | IDD_TUFLOW_CONSTITUENT_CONTROL
533) TUFLOW AD | 797 | IDD_TUFLOW_AD
534) Dredging Scheduling | 798 | IDD_DREDGE_SCHEDULING_ATTS
535) Estimating generated Parcels | 799 | IDD_PTM_ESTIMATE_GENERATED_PARCELS
536) Curvilinear Grid Display Options | 801 | IDD_DISPOPTS_CURV_NODESTR
537) SED-ZLJ | 805 | IDD_SEDZLJ_NODE_ATTS
538) SED-ZLJ | 805 | IDD_SEDZLJ_CORE_PROPS
539) SED-ZLJ | 806 | IDD_SEDZLJ_EROSION_RATES
540) SED-ZLJ | 814 | IDD_SEDZLJ_SED_CLASSES
541) SED-ZLJ | 815 | IDD_SEDZLJ_CORE_EROSION_RATES
542) Damping / Porosity Coverage | 818 | IDD_RUNUP_DAMPING_POROSITY
543) Probes Coverage | 820 | IDD_RUNUP_PROBES
544) BOUSS Runup / Overtopping | 821 | IDD_RUNUP_NEW_SIM
545) Transect Profile | 822 | IDD_RUNUP_TRANSECT_PROFILE
546) Solution View | 823 | IDD_RUNUP_SOLUTION
547) ADH Material Properties | 826 | IDD_ADH_FLOW_REFINEMENT
548) IDD_DIALOG1 | 827 | IDD_DIALOG1
549) CSTORM-MS | 827 | IDD_CSTORM_SPATIAL
550) CSTORM-MS | 828 | IDD_CSTORM_TIMELINE
551) IDD_GENERIC_MODEL_CURV_AXIS | 829 | IDD_GENERIC_MODEL_CURV_AXIS
552) CSTORM-MS | 831 | IDD_CSTORM_OVERVIEW
553) Generic Model Graphical Interface | 832 | IDD_GENERIC_MATERIALPROPS_TAB
554) GenCade Structures | 834 | IDD_PLOTWIZ2_INLETTS
555) GenCade Structures | 835 | IDD_PLOTWIZ2_SHORELINE
556) Auto-Create Probes | 836 | IDD_RUNUP_AUTO_PROBES

557) Auto-Create Probes | 837 | IDD_RUNUP_AUTO_PROBE_ELEVS
558) IDD_FEAT_FIND | 838 | IDD_FEAT_FIND
559) Interpolation | 839 | IDD_INTERPOPTS_RASTER
560) CSTORM-MS | 840 | IDD_CSTORM_THREADS
561) ADH Model Control Advanced | 845 | IDD_ADH_MC_ADVANCED_CARD
562) Display Options | 3220 | IDD_LINE_DISPLAY_ATTS
563) IDD_TOG_WIDECOLUMN | 3576 | IDD_TOG_WIDECOLUMN
564) IDD_STEERWIZ_CHOOSE | 4009 | IDD_STEERWIZ_CHOOSE
565) IDD_SHOW_TEXT | 4302 | IDD_SHOW_TEXT
566) Dataset Toolbox | 4451 | IDD_DATASET_TOOLBOX
567) Annotations#Text | 4941 | IDD_TEXT_ATTS
568) Dataset Toolbox | 4942 | IDD_DST_GRID_SPACING
569) Cartesian Grid Module Right-Click Menus | 4943 | IDD_GRID_CREATE_TRANSFORMED_GRID
570) TUFLOW ZShape | 4944 | IDD_TUFLOW_SHAPEZ
571) TUFLOW ZShape | 4944 | IDD_TUFLOW_ZSHAPE_POLY_ARC_OPTS
572) TUFLOW Inlet Database | 4945 | IDD_TUFLOW_INLET_DBASE
573) TUFLOW ZShape | 4946 | IDD_TUFLOW_ZSHAPE_POINT_OPTS
574) TUFLOW 2D Geometry Components | 4947 | IDD_TUFLOW_COMPONENT_SCATTER_PROPS
575) TUFLOW 2D Flow Constriction Shape Coverage | 4948 | IDD_TUFLOW_COV_FCSH_LAYERED
576) TUFLOW 2D Flow Constriction Shape Coverage | 4948 | IDD_TUFLOW_COV_FCSH
577) TUFLOW 2D Flow Constriction Shape Coverage | 4949 | IDD_TUFLOW_FCSH_LAYERED_POINTS
578) TUFLOW 2D Flow Constriction Shape Coverage | 4949 | IDD_TUFLOW_FCSH_POINTS
579) IDD_SELECT_TREE_ITEM_WITH_MATERIAL | 4950 | IDD_SELECT_TREE_ITEM_WITH_MATERIAL
580) Arcs | 4951 | IDD_ARC_SURVEY_PATH_TOOL
581) IDD_COORD_CLARIFY | 4952 | IDD_COORD_CLARIFY
582) ADH Vessel Coverage | 4953 | IDD_ADH_VESSEL_COV_ATTS
583) ADH Vessel Coverage | 4954 | IDD_ADH_VESSEL_POINT_OPTS
584) Graphics | 4955 | IDD_PREFS_GRAPHICS
585) Quadtree Generator Coverage | 4956 | IDD_TELESCOPE_POLYATTS
586) ADCIRC Wind Coverage | 4957 | IDD_ADCIRC_STORM_COV_ATT
587) ADCIRC Wind Coverage | 4958 | IDD_ADCIRC_STORM_NODE_ATT
588) Split Feature Arcs Utility | 4959 | IDD_ARC_FILTER_SEGS_TOOL
589) PTM Feature Point Attributes Dialog | 4960 | IDD_PTM_POINT_ATTS1
590) CSTORM-MS | 4960 | IDD_CSTORM_MODEL_CONTROL
591) ADH Model Control Iterations | 4961 | IDD_ADH_MC_ITERATIONS
592) ADH Bed Layers Assignment | 4963 | IDD_ADH_BED_LAYERS1
593) ADH Consolidation | 4963 | IDD_ADH_CONSOLIDATION
594) IDD_ADH_DREDGE_LOSS | 4964 | IDD_ADH_DREDGE_LOSS
595) IDD_CURVDISPOPTS1 | 4965 | IDD_CURVDISPOPTS1
596) Mesh Module Display Options | 4965 | IDD_MESHDISPOPTS
597) Coverage | 4966 | IDD_FEATDISPOPTS_DREDGING
598) CSTORM-MS | 4967 | IDD_FEATDISPOPTS_CSTORM

599) General Tab | 4968 | IDD_WAM_MC_GRID_GENERAL
600) Output Tab | 4969 | IDD_WAM_MC_GRID_OUTPUT
601) General Tab | 4970 | IDD_WAM_MC_SIM_GENERAL
602) Spectra Tab | 4971 | IDD_WAM_MC_SIM_SPECTRA
603) LTFATE | 8210 | IDD_LTFATE_MC_GENERAL
604) LTFATE | 8211 | IDD_LTFATE_MC_OBSERVATIONS
605) LTFATE | 8212 | IDD_LTFATE_MC_VISCOSITY
606) LTFATE | 8213 | IDD_LTFATE_MC_WETTING
607) LTFATE | 8214 | IDD_LTFATE_MC_WIND
608) LTFATE | 8215 | IDD_LTFATE_MC_COMPUTE
609) LTFATE | 8216 | IDD_LTFATE_MC_OUTPUT
610) LTFATE | 8217 | IDD_LTFATE_MC_FRICTION
611) LTFATE | 8218 | IDD_LTFATE_MC_PARAMETERS
612) IDD_PLOTWIZ2_PTMGAGE | 8219 | IDD_PLOTWIZ2_PTMGAGE
613) LTFATE | 8220 | IDD_LTFATE_SEDZLJ_OPT
614) LTFATE | 8221 | IDD_LTFATE_OBSERVATION_STATION
615) ADH Model Control Iterations | 8222 | IDD_ADH_MC_ITERATIONS1
616) ADH Model Control Advanced | 8222 | IDD_ADH_MC_ADVANCED
617) STWAVE Model Control | 8223 | IDD_STWAVE_ITERATORS
618) STWAVE Model Control | 8223 | IDD_STWAVE_ITERATIONS
619) Raster Module | 8224 | IDD_RASTER_MODULEINFO
620) Synthetic Storm Coverage | 8225 | IDD_FEATDISPOPTS_SYNTHETIC_STORM
621) Map to 1D Grid | 8226 | IDD_MAP_TO_1DGRID
622) Startup | 8227 | IDD_PREFS_STARTUP
623) TUFLOW Manholes | 8228 | IDD_TUFLOW_CTRL1D_MANHOLE
624) Dataset Toolbox | 8229 | IDD_DST_ANG_CONV
625) QDlgSavePointManager | QDlgSavePointManager
626) QDlgCellAtt | QDlgCellAtt
627) QDlgSolutionView | QDlgSolutionView
628) QDlgRelaxGrid | QDlgRelaxGrid
629) Data Calculator | QDlgDsetCalculator
630) Conversions Scalar/Vector | QDlgDsetScalarToVector
631) Conversions Scalar/Vector | QDlgDsetVectorToScalar
632) Fleet Wind Files | QDlgOpenFleetWindFile
633) PTM Trap Output | QDlgTrapOutputFilterRange
634) Scatter Interpolation | QDlgDsetInterp
635) QDlgAssignNdstrBc | QDlgAssignNdstrBc
636) QDlgLTFATEMcWindData | QDlgLTFATEMcWindData
637) QDlgSedimentBc | QDlgSedimentBc
638) QDlgTrimWithMask | QDlgTrimWithMask
639) Select With Poly | QDlgSelectDeleteWithPoly
640) QDlgUndoFeatureControl | QDlgUndoFeatureControl
641) Initial Values | QDlgInitialValues

642) ADCIRC Spatial Attributes | QDlgSpatialAttributes
643) ADCIRC Spatial Attributes | QDlgSpatialAttsCustom
644) Extract Elevations | QDlgExtractWeirElevs
645) QDlgFort141Options | QDlgFort141Options
646) Switch Current Model | QDlgModelTemplateSelector
647) SRH-2D Material Properties | QDlgTextureChooser
648) QDlgGetDateTime | QDlgGetDateTime
649) SMS:Scatter_Menu#Merge_Report | QDlgReport
650) Project Explorer | QDlgSelectTreeItem
651) Raster Values as Elevation | QDlgRasterSetOptions
652) Generate Contour Breaklines | QDlgAutoBreaklines
653) Scatter Data Menu | QDlgScatFilterOptions
654) QDlgObjectEditor | QDlgObjectEditor
655) TUFLOW Manholes | QDlgManholeAtts
656) QDlgBatchRun | QDlgBatchRun
657) QDlgConversionReport | QDlgConversionReport
658) QModelWrapperTUFLOW | QModelWrapperTUFLOW
659) ADH Bed Layers Assignment | QDlgAdhBedLayerTable
660) QDlgAdhDistributionChart | QDlgAdhDistributionChart
661) ADH Sediment Library Control | QDlgAdhSedLibraryCtrl
662) Arcs | QDlgAlignArc
663) ADlgCgridSpectralCoverageAtts | ADlgCgridSpectralCoverageAtts
664) Arcs#Create Contour Arcs | QDlgCreateContourArcs
665) CSHORE | QDlgModelControlCShore
666) Grid Nesting | QDlgWAMToSTWAVEOptions
667) QDlgCgridSpectralCoverageAtts | QDlgCgridSpectralCoverageAtts
668) SRH-2D Material Properties | QDlgMaterialProperties
669) SRH-2D Model Control | DynSrhModelControl
670) SRH-2D Boundary Conditions | DynSrhNdstrBc
671) Merge 2D Meshes | QDlgMesh2dMergeOptions
672) CHS Web Data Files | QDlgImportChs
673) ADCIRC Weirs and Island Barriers | QDlgRemoveWeirs
674) ADCIRC Weirs and Island Barriers | QDlgAddWeir
675) CMS-Flow Model Control | DynCMSFlowModelControl
676) CMS-Flow Observation Cells | DynCMSFlowSavePoints
677) Boundary Conditions | DynCMSFlowBC
678) Summary Table | QDlgSummaryParams
679) Activity Classification Coverage | QDlgActivityCov
680) Select Arc Type | QDlgSelectBcType
681) Map Display Options | QDlgBc
682) Nesting Dialog | QDlgNesting
683) SRH-2D Populate Dialog | QDlgPopulateCurve
684) Interpolation Source and Options Dialog | QDlgInterpSourceAndOptions

7.1. Dynamic Model Interface

Dynamic Model Interface Schema

The dynamic model interface is available in SMS version 11.2 and later. It is primarily a tool for developers. The dynamic model interface provides a way to quickly generate an interface for a numeric model.

Dynamic Dialogs

Dynamic dialogs are a quick way to generate dialogs without having to compile code. To add or delete a widget on a dynamic dialog, the user simply modifies the XML document. When the modified document is loaded, the new/modified dialog exists. The XML document defines the layout, behavior and the different dialog controls.

The dialog is divided into 2 sections: the tree view and the widget view. The tree is on the left side of the dialog and contains groups and items that represent data. Clicking on a group or item will result in the widget view being updated to match the selected tree or group item.

- Order of the tree items and groups will match the order defined in the XML file.
- Clicking on group will display all children items in the widget view.
- Clicking on a single item will display associated widgets in widget view.
- Tree item will also display the values in a non-editable field.
- Unique_name use format: file::unique_name#value
- Multiple widgets, when displayed on the right side, can be expanded/collapsed.
- Can have nested groups.
- Keywords are words SMS has reserved in the schema and can not be used as unique names. All keywords will start with a "#".

Available Keywords

This is a list of available keywords. Additional keywords are defined with the various custom control widgets. Note that in changing from version 1 to version 2, all tags that were `<custom_control_XXXX>` were changed to `<control_XXXX>`.

General Keywords	
#card_name	
#geom_name	name of geometry as it shows up in the project explorer
#project_name	
#value	
#units	
#xmdf_path	path inside the XMDF file
#file_name	
#file_path	
#sms_path	path in sms project explorer once inside a geometric item
#count	used for a widget in a table for counting the rows/columns
#geom_guid	
Arc Keywords	

#arc_id	
#arc_count	
#arc_point_count	
Coverage Keywords	
#area_property	
#activity_classification	
#cgrid_generator	
#location	
#mapping	
#mesh_generator	
#observation	
#quadtree_generator	
#spectral	
executable_progress_update Keywords	
#progress_amount	
#progress_max	
Material Keywords	
#material_id	
#material_name	
#material_count	
#unassigned	
Point Keywords	
#point_count	
#point_id	
#point_x	
#point_y	
#point_z	
Polygon Keywords	
#polygon_id	
#polygon_count	
#polygon_point_count	
Projection Keywords	
#horizontal_datum	NAD83, NAD27, LOCAL

#horizontal_system	UTM, STATE_PLANE, GEOGRAPHIC, LOCAL
#horizontal_units	FEET, METERS, DEGREES
#horizontal_zone	3104, etc.
#vertical_datum	
#vertical_units	

Elements

Sorted alphabetically by element name.

Elements A - C

<arc_att>	
Info:	Used to specify what attributes should be used for the arcs of a coverage.
Version(s):	2
Attributes:	none
Children:	menu_item , snap
Used by:	declare_coverage
Example:	
<attribute_set>	
Info:	This represents an item in the project explorer that is some sort of grouping.
Version(s):	4
Attributes:	none
Children:	takes , menu_item
Used by:	model
Example:	See the model example.
<card>	
Info:	Determines the card name and format when the item is exported. For more examples see <export_format> .
Version(s):	1
Attributes:	none
Children:	card_name , export_format , export_location , dependency , anything beginning with "process_each_", export_group , export_optional , use_paramter .
Used by:	item (version 1), file_def (version 2)

Example:	<item text = "Formulation">
	`<card>`
	`<card_name>FORMULATION</card_name>`
	`<export_format>card  "formulationUnits"</export_format>`
	`</card>`

<card_name>	
Info:	The name of the card which is used in the card file.
Version(s):	1
Attributes:	none
Children:	none
Used by:	card
Example:	
<check_box>	
Info:	Widget that displays text that is checked/unchecked.
Version(s):	1
Attributes:	default , export_text_checked , export_text_unchecked , text , unique_name .
Children:	dependency , text_style
Used by:	item

Example:	<check_box text = "Calculate Sediment Transport"
	`default = "checked"`
	`unique_name = "togCalcSedimentTransport"`
	`export_text_checked = "ON"`
	`export_text_unchecked = "OFF">`
	`<dependency>…</dependency>…`
	`</ check_box >`

<color>	
Info:	The color of an item, expressed in red, green, and blue values ranging from 0 to 255. All 0 values for red, green and blue are black. Added in SMS 11.2.
Version(s):	1
Attributes:	red , green , blue
Children:	none
Used by:	text_style

Example:	This would change the text to a bright red color.
	`<text_style>`
	`<color red = "255" green="0" blue="0"></color>`
	`<text_style>`

<column>	
Info:	Definition of the column in the table. A widget defined in a column will be the widget used for each cell in the column.
Version(s):	1
Attributes:	text , read_only , optional
Children:	dependency , text_box , combo_box , edit_box , all elements starting with "control_",

	check_box
Used by:	table

Example:	Column1 is read only, Column 2 isn't read only and is also optional.
	This means the column can have empty fields and a warning message won't be displayed.
	`< table >`
	`. . .`
	`<column text = "Column1" read_only>`
	`</column>`
	`< column text = "Column2 optional></ column>`
	`</table>`

<combo_box>	
Info:	Widget that displays list of options. Only 1 can be selected. If no default is specified and the optional tag is present, then an empty option will be added to the combo box. If there is no default tag, and no optional tag, then the first item will be default. In version 2, if a combo box has display_options, and is part of a dialog that is used as an arc attribute, then a limited set of display options will appear for the options of the combo box. The display option for each option of the combo box will only be line thickness and color. The display option is NOT saved at any point and will be reset every time SMS is opened.
Version(s):	1
Attributes:	optional , unique_name , default
Children:	option , optional , dependency , text_style , display_options (version 2)
Used by:	item , row , column

Example:	Creates a combo box with hours, minutes and seconds. Hours is the default item.
	`<combo_box unique_name="cbxTransportUnits">`
	`<option text = "hours" default></option>`
	`<option text = "minutes"></option>`
	`<option text = "seconds"></option>`
	`<dependency>…</dependency>…`
	`</ combo_box >`

<command_args>	
Info:	Defines the command line arguments to run a particular executable.
Version(s):	3
Attributes:	use_file
Children:	none
Used by:	executable_command
Example:	See executable example.
<comment>	

Info:	Defines a character or sequence of characters that defines the start of a comment on a line. Comments are always terminated by an end of line.
Version(s):	2
Attributes:	none
Children:	none
Used by:	use_file_def

Example:	The following example has a file where the exclamation point starts the comment.
	`<file_def>`
	`...`
	`   <comment>!</comment>`
	`…`
	`</file_def>`

<condition>	
Info:	Evaluates two or more objects using GREATER_THAN, GREATER_THAN_EQUALS, LESS_THAN, LESS_THAN_EQUALS, EQUALS, AND, OR, NOT, CHECKED, UNCHECKED, EMPTY. If condition is not met, then message is displayed (if model_check) or widget is hidden/dimmed (if dependency). String literals, such as an entry in a combo-box, must be enclosed in double-quotes (ex. "Combo box entry").
Version(s):	1
Attributes:	none
Children:	none
Used by:	model_check , dependency , text_style

Example:	`<table>`
	`<column  text="A">`
	`     <edit_box    unique_name="colA">`
	`     </edit_box>`
	`   </column>`
	`  …// other columns also defined`
	
	`  <model_check problem_text="Column D or E is required">`
	`      <condition>( (colA EQUALS 0.0 OR colB LESS_THAN 0.0) AND NOT(colD NOT EQUALS EMPTY OR colE NOT EQUALS "Some value"))</condition>`
	`  </model_check>`
	`</table>`

<contains>	
Info:	Holds all the groups and items of a group.
Version(s):	2
Attributes:	none
Children:	group , item
Used by:	group

Example:	
<control_curve>	
Info:	Displays a curve push button. When pushed, the xy curve values can be updated and a curve is displayed.
Version(s):	2
Attributes:	flags , unique_name , max_row_count
Children:	x_column , y_column , dependency
Used by:	item , row , column

Example:	This example creates a curve limited to 10 rows.
	`<item>`
	`   <control_curve  unique_name="myCurve1"`
	`                            max_row_count="10"`
	`                            flags = "XY_USEDATE">`
	`       <x_column   text="Time"></x_column>`
	`       <y_column   text="Velocity">`
	`       </y_column>`
	`   </control_curve>`
	` …`

<control_dataset>	
Info:	Displays an edit box, and push buttons for select, delete and create. Once selected, the dataset string is placed into the edit box. The dataset_type can be scalar or vector. Possible keyword outputs are file_name (which is the name of the dataset without path), geom_name, and file_path (full path c:\somewhere) and sms_path. By default, all datasets of geometries that are part of the simulation (or part of the parameters if one or more use_parameter is used) can be selected. The the geometry attribute is set to "all", then the datasets from all geometries currently loaded into SMS can be selected from, regardless of relation to the simulation and assuming no other attribute prevents it. If "all" is used, then the dataset values will be interpolated to the geometry in the simulation. The time_type attribute can be set to "all" (default), "transient", or "steady state". When "transient" is used, only datasets with multiple timesteps may be selected. When "steady state" is used, only timesteps without timesteps may be selected. The select_time attribute can be set to "all" (default), "single", or "range". If "single" is used, then only a single timestep of the dataset will be selected. If "range" is used, then both a starting and ending timestep can be selected. By default, all timesteps of a dataset are used. (For possible future use.)The interpret_time attribute can be set to "true" or "false" (default). When true, this attribute tells SMS to interpolate dataset values at the timesteps in the selected dataset to values at timesteps calculated from a start time, end time, and delta time that is specified by the user.
Version(s):	1 (<custom_control_XXXX>), 2
Attributes:	geometry , time_type , select_time , interpret_time , unique_name , dataset_type , default ,

	name
Children:	push_button , dependency , dataset_name , text_style , use_parameter (version 2), text , export_text
Used by:	item , row , column , use_parameter

Example:	This example creates the custom control with only the select and delete buttons.
	`<item>`
	`    <control_dataset   dataset_type="scalar">`
	`        <push_button>select<push_button>`
	`        <push_button>delete</push_button>`
	`    </control_dataset>`
	 …

<control_date>	

Info:	Displays date and time selector, where a date can be selected. Possible output is value, which will be the date formatted as specified in SMS preferences. Possible outputs are year, month, day, hour, minute, second. Keywords: • #DAY_DIGIT • #DAY_DIGIT_ZERO • #DAYS_IN_YEAR • #DAY_SHORT_NAME • #DAY_LONG_NAME • #MONTH_DIGIT • #MONTH_DIGIT_ZERO • #MONTH_SHORT_NAME • #MONTH_LONG_NAME • #YEAR_2 • #YEAR_4 • #HOUR • #HOUR_ZERO • #HOUR_24 • #HOUR_24_ZERO • #MINUTE • #MINUTE_ZERO • #SECOND • #SECOND_ZERO • #AM_PM • #AM_PM_CAPS
Version(s):	1 (<custom_control_XXXX>), 2
Attributes:	unique_name
Children:	dependency , text_style
Used by:	item , row , column

Example:	`<item text =”Start date>`
	`<item>`
	`    <control_dataset   dataset_type="scalar">`
	`        <push_button>select<push_button>`
	`        <push_button>delete</push_button>`
	`    </control_dataset>`

	...
<control_density>	
Info:	Displays an edit box and combo box. Combo box can contain kg/m^3, gr/cm^3, lb/ft^3. Possible outputs are value and units. Keywords: • #MPV_MG_PER_L • #MPV_G_PER_L • #MPV_KG_PER_CU_M • #MPV_G_PER_CU_CM • #MPV_LB_PER_CU_FT
Version(s):	1 (<custom_control_XXXX>), 2
Attributes:	default , unique_name
Children:	option , dependency , range , text_style
Used by:	item , row , column
Example:	Displays kg/m^3 and lb/ft^3 in combo box. If kg/m^3 selected and exported, returns the text "kg m".

```
<item>
   <text>Density </text>
   <control_density>
       <option      text="kg/m^3"
                         export_text = "kg m"
                         unit_keyword="#MPV_KG_PER_CU_M">
       </option>
       <option    text="lb_ft^3"
                       export_text ="lbs ft"
                       unit_keyword = "#MPV_LB_PER_CU_FT">
        </option>
    </control_density>
 ...
```

<control_duration>	
Info:	Displays an edit box and combo box. Combo box contains days, hours, minutes, seconds. Possible outputs are value and units. Keywords: • #TIME_SECONDS • #TIME_MINUTES • #TIME_HOURS • #TIME_DAYS • #TIME_WEEKS • #TIME_YEARS
Version(s):	1 (<custom_control_XXXX>), 2

Attributes:	default , unique_name
Children:	option , dependency , range , text_style
Used by:	item , row , column

Example:	Displays only minutes and hours in combo box. If hours selected and exported, returns the text "hrs".
	`<item text ="Transport Time Step">`
	`<control_duration>`
	`<option text="minutes"`
	`export_text = "min"`
	`unit_keyword = "#TIME_MINUTES">`
	`</option>`
	`<option text = "hours"`
	`export_text = "hrs"`
	`unit_keyword = "#TIME_HOURS"`
	`default>`
	`</option>`
	`</control_duration>`
	…

<control_executable>	
Info:	Displays a button for running a simulation from a dialog. Unlike the normal simulations found in the project explorer, these simulations are generally hidden from the user. As such, the options that can be used are significantly less than those of a normal simulation. The model attribute defines the name of the model to be executed. The model may be one that is already defined by another XML file. Dialogs can be specified for data that will be kept for the model, but will not be shown to the user. The simulation and files specified with this control may not be the same as one already specified in another XML file.
Version(s):	3
Attributes:	model , unique_name
Children:	option , dependency , range , text_style
Used by:	unique_name , dependency, text_style, use_parameter , simulation , dialogs , files

Example:	This example declares a control_executable that runs "MyModel".
	`<control_executable model = "MyModel"`
	`unique_name="exeMyModel">`
	`<declare dialogs>`
	`<declare dialog name="Arc BC">`
	`<pages>`
	`<page_def name="ArcBC"`
	`display="NO_NAV">`
	`<contains>`
	`<item>`
	`<edit_box`
	`unique_name="edtMyModelText">`
	`</edit_box>`

	``` </item> </contains> </page_def> </pages> </dialog> </dialogs> <simulation> <executable name ="MyModel"> <use_parameter>HydroFile</use_parameter> <execute_command> <command_args>" \"%s\"", edtMyModelText</command_args> <execute>"%s", #executable_name</execute> </execute_command> </executable> <input_files> <input_file> <use_file_def>myFile</use_file_def> <export_location>"%s.srhhydro", #project_name</export_location> <declare_parameter>HydroFile</declare_parameter> <input_file> </input_files> </simulation> <files> <declare file_def name=MyFile type=CARD_ASCII> <comment>! this is myfile</comment> </declare file_def> </files> </control_executable> ```
<control_feature_selector>	
Info:	Push button that opens a canvas window that allows users to select one or more points, arcs or polygons that are in the main window.
Version(s):	3
Attributes:	feature , unique_name
Children:	range
Used by:	item , row , column
Example:	Allows the user to select 1 to 10 arcs ``` <item> <control_feature_selector feature = "arc"> unique_name="mySelector"> <range>1-10</range> </control_feature_selector> … ```
<control_file_selector>	

<table>
<tr><td>Info:</td><td>Push button that opens a file open dialog to select files. Possible outputs are file_path.</td></tr>
<tr><td>Version(s):</td><td>1 (<custom_control_XXXX>), 2</td></tr>
<tr><td>Attributes:</td><td>default , create_file , filter , unique_name</td></tr>
<tr><td>Children:</td><td>dependency , text_style</td></tr>
<tr><td>Used by:</td><td>item , row , column</td></tr>
<tr><td>Example:</td><td>Opens a file dialog, filtering on files with *.h5 and *.cmcards extensions
<pre>
<item>
   <text>File:</text>
   <control_file_selector
           filter = "Cmcards file (.cmcards)">
    </control_file_selector>
…
</pre></td></tr>
<tr><td colspan="2"><control_length></td></tr>
<tr><td>Info:</td><td>Displays an edit box and combo box. Combo box can contain meters, cm, mm, um, ft, in. Possible outputs are value and units.
Keywords:
#LEN_MM
#LEN_CM
#LEN_M
#LEN_KM
#LEN_INCH
#LEN_FT
#LEN_YD
#LEN_MILE
#LEN_UM</td></tr>
<tr><td>Version(s):</td><td>1 (<custom_control_XXXX>), 2</td></tr>
<tr><td>Attributes:</td><td>default , unique_name</td></tr>
<tr><td>Children:</td><td>option , dependency , range , text_style</td></tr>
<tr><td>Used by:</td><td>item , row , column</td></tr>
<tr><td>Example:</td><td>Displays only cm, and mm in combo box. If cm is selected and exported, returns the text "centimeter".
<pre>
<item   text = "Length">
   <control_length   unique_name="myLength">
       <option>
          text = "cm"
          export_text="centimeter"
          unit_keyword = "#LEN_CM">
       </option>
       <option>
</pre></td></tr>
</table>

```
        text = "mm"
        export_text = "mm">
        unit_keyword="#LEN_MM"
        default
      </option>
  </control_length>
 …
```

<control_nesting>

Info:	Push button that opens a special dialog to create nesting points. The dialog allows the user to select a parent and child grid and a coverage to put the nesting sites in.
Version(s):	3
Attributes:	unique_name
Children:	dependency
Used by:	item , row , column
Example:	Creates the nesting dialog widget

```
<item>
   <control_nesting   unique_name="ccNesting">
    </control_nesting>
…
```

<control_set>

Info:	Table like widget that allows for multiple points, polygons, or arcs to be joined together as a set. An example usage would be to join two arcs in creating a weir. By default the <control_set> contains two columns, object and role. The object column identifies the arc, point, or poly id. The role column is a user defined combo box that allows for customization of types and behavior. See <roles>. Addition columns can be added by simply adding a <column> tag with a widget. See tables. When exporting you can use the #object or #role keyword for retrieving data from those particular columns.
Version(s):	3
Attributes:	min (default = 0), max (default = 2.147 billion), increment (default is 1), unique_name
Children:	column , roles
Used by:	item , row , column
Example:	Example of a culvert widget with a extra "Name" column. The culvert requires 2 arcs.

```
<control_set  min  =  "2"                       max  =  "2"
unique_name  =  "tblSetCulverts">      <roles  behavior  =
"SWAP">         <role    text  =  "upper">            <range>1-
1</range>         </role>         <role   text  =  "lower">
<range>1-1</range>                   </role>        </roles>
<column    text  =  "Name">         <edit_box
unique_name="edtMyArcsName"
type="text">           </edit_box>        </column></control_set>
```

User allows for either 2 or 4 arcs on a weir.

```
<control_set  min  =  "2"  max  =  "4"  increment  =  "2"  >….
```

<control_velocity>

Info: Displays an edit box and combo box. Combo box can contain m/sec, m/min, m/hours, etc.... Also could be m/sec, cm/sec, ft/sec, etc... The option tag specifies which options are displayed. The edit box will display doubles, within the given range, if provided. Possible outputs are value and units.

Keywords:

- #VEL_M_PER_S
- #VEL_KM_PER_H
- #VEL_FT_PER_S
- #VEL_MPH
- #VEL_KNOTS
- #VEL_CM_PER_S
- #VEL_MM_PER_S

Version(s): 1 (<custom_control_XXXX>), 2

Attributes: default , unique_name

Children: option , dependency , range , text_style

Used by: item , row , column

Example: Displays m/sec and cm/sec in combo box. If m/sec selected and exported, returns the text "m sec".

```
<item>    <text>Velocity  </text>    <control_velocity
unique_name="ccVelocity">        <range>0-100</range>
<option   text="m/sec"                       export_text  =  "m
sec"                     unit_keyword  =  "#VEL_M_PER_S">
</option>        <option    text  =  "cm/sec"
export_text  =  "cm  sec"xt                        unit_keyword
=  "#VEL_CM_PER_S">         </option>     </control_velocity>  …
```

<control_volume_flow>

Info: Displays an edit box and combo box. Combo box can contain m^3/sec, m^3/min, m^3/hours, etc.... Also could be m^3/sec, cm^3/sec, ft^3/sec, etc... The option tag specifies which options are displayed. The edit box will display doubles, within the given range, if provided. Possible outputs are value and units.

Version(s): 2

Attributes: default , unique_name

Children: option , dependency , range , text_style

Used by: item , row , column

Example: Displays m^3/sec and cm^3/sec in combo box. If m^3/sec selected and exported, returns the text "m^3 sec".

```
<item    text="Volume  flow">    <control_volume_flow
unique_name  ="ccVolumeFlow">                   <range>0-
100</range>        <option            text  =  "m^3/sec"
export_text  =  "m^3  sec"            unit_keyword  =
"#VFLOW_CU_M_PER_SEC">        </option>        <option
text  =  "cm^3/sec"            export_text  =  "cm^3  sec"
unit_keyword  =  "#VFLOW_CU_CM_PER_SEC">          </option>
</control_volume_flow>  …
```

<count_filter>

Info:	Specifies the range on which the process_on_count is valid.
Version(s):	2
Attributes:	none
Children:	none
Used by:	process_on_count
Example:	

Elements D - H

<dataset_name>	
Info:	Determines the name of the dataset to be created. Uses the standard printf and sprintf format (www.cplusplus.com/reference/clibrary/cstdio/printf/). The keywords #row_number, #column_number, #row_name, and #column_name are used when the dataset is in a table.
Version(s):	1
Attributes:	none
Children:	none
Used by:	control_dataset
Example:	<control_dataset> `<dataset_name>"Five percent Layer %d", #row_number</dataset_name> …</control_dataset>`
<declare_coverage>	
Info:	Defines a coverage type for the model.
Version(s):	2
Attributes:	z_is_elev , bind_to , name , use_icon
Children:	point_att , arc_att , polygon_att , material_att , menu_item
Used by:	model
Example:	This defines a coverage that can be created. `<declare_coverage z_is_elev = "true" name="MyCov"> <point_att> <menu_item text = "Assign BC…" use_dialog = "NodeAtt"> </menu_item> </point_att> <arc_att> <menu_item text = "Assign BC…" use_dialog= "NodestringBC"> </menu_item> </arc_att> <material_att use_dialog= "MaterialProp"</material_att></coverage>`
<declare_dialogs>	
Info:	Contains all the dialogs used in a model.
Version(s):	2
Attributes:	none
Children:	declare_dialog
Used by:	model
Example:	

<declare_dialog>	
Info:	Used to define a dialog.
Version(s):	2
Attributes:	name , help_button_url , help_button_wiki
Children:	pages
Used by:	dialogs
Example:	
<declare_file>	
Info:	A way to declare a file that will be used in the future.
Version(s):	3
Attributes:	name , file_type
Children:	comment , identifier , card , export_format , export , process_each_row , process_each_coverage , export_group , process_on_condition , xmdf_group , xmdf_data , xmdf_geometry , xmdf_dataset , section
Used by:	files
Example:	
<declare_pages>	
Info:	Signals the beginning of page creation for a dialog. One or more pages may be created using declare_page.
Version(s):	3
Attributes:	none
Children:	declare_page
Used by:	dialog

Example:	<declare_dialog name ="Model Control">
	`<declare_pages>        <declare_page   text="Page1">`
	`….        </declare_pagef>        <declare_page   text  ="Page2">`
	`….        </declare_page>    </declare_pages>`

<declare_page>	
Info:	This defines the page or tab of a dialog. If there is only one page_def in a dialog, then no tabs will appear.
Version(s):	3
Attributes:	text, display
Children:	contains
Used by:	declare_pages
Example:	
<declare_parameter>	
Info:	A way to declare data that will be used in the future.

Version(s):	3
Attributes:	none
Children:	none
Used by:	input_file, anything beginning with "process_each", takes
Example:	
<dependency>	
Info:	Dependencies allow widgets to be hidden, dimmed (grayed out), or shown based on the state/value of another widget. The [<dependency>]() tag can be placed in any widget, combo-box option, item, or group. A user needs to specify the condition that this object is dependent upon and what the value(s) need(s) to be in order for the object to be shown/not dimmed. If the condition is false, by default the control is hidden. To have the control dimmed, use the [<dim>]() tag.
Version(s):	1
Attributes:	dim
Children:	condition
Used by:	card , check_box , text_box , combo_box , table , any element that begins with "custom_control", edit_box , item , group
Example:	**Example 1** We only want our edit_box displayed if the check_box is checked (true). If check_box is not checked, we want to dim (show but is grayed out) the edit_box. <check_box unique_name = "MyBox">…<edit_box> <dependency dim> <condition>MyBox EQUALS CHECKED</condition> </dependency> **Example 2** We want our text_box displayed if the edit_box has a value between 0-3 or if the combo box has the value minutes or days. <edit_box unique_name="MyEdit_box">…<combo_box unique_name = "MyComboBox"> <option text = "hours"></option> <option text ="minutes"></option> …<text_box> <dependency> <condition>(MyEdit_box GREATER_THAN_EQUALS 0 AND MyEdit_box LESS_THAN_EQUALS 3) OR (MyComboBox EQUALS "minutes" OR MyComboBox EQUALS "days")</condition> </dependency></text_box>
<display_options>	
Info:	Specifies that the options in the combo box are to be used as arc/nodestring boundary condition display options. This element should only be used once in a combo_box and once in a edit_box per dialog.
Version(s):	2
Attributes:	type , text
Children:	none
Used by:	combo_box
Example:	<combo_box unique_name="cbxArcDeleteMe"> <display_options type= "point" text ="Save Points Name"></display_options> <option text="Hydro" default></option> <option text="Sediment"></option></combo_box>
<dynamic_model>	

Info:	Used to define a dynamic model.
Version(s):	1
Attributes:	file_type – always "dynamic model", version
Children:	model
Used by:	none
Example:	
<edit_box>	
Info:	Widget that displays text or numbers. If <type> not specified, default to double. The type can also be text, or integer.
Version(s):	1
Attributes:	default , optional , type , unique_name
Children:	range , dependency , text_style , display_options
Used by:	item , row , column
Example:	<edit_box type="double" default="1.0" unique_name="edtBox5" optional> `<range>0, 2.2</range> <dependency>…</dependency>…</ edit_box >`
<end_card>	
Info:	Defines that the card end with a single iteration of the enclosing <process_each_XXX>
Version(s):	1
Attributes:	none
Children:	none
Used by:	anything starting with process_each
Example:	
<executable>	
Info:	Represents a single executable used by the model.
Version(s):	2
Attributes:	executable_order , text , default_executable_name32 , default_executable_name64
Children:	use_parameter , executable_command , executable_progress_update
Used by:	simulation

Example:

Example 1

```
<executable  name  ="MyModel"   executable_order  =  "1"
default_executable_name32="MyModel32.exe"
default_executable_name64="MyModel64.exe">
<use_parameter>HydroFile</use_parameter>      <execute_command>
<command_args>"  \"%s\"",  edtMyModelText</command_args>
<execute>"%s",  #executable_name</execute>
</execute_command>  </executable>   <input_files>
<input_file>          <use_file_def>myFile</use_file_def>
<export_location>"%s.srhhydro",  #project_name</export_location>
<declare_paremeter>HydroFile</declare_parameter>
<input_file>     </input_files>
```

Example 2 In the following example, one instance of the numeric model is executed per coverage. A file will be exported per coverage and no command line arguments are used when launching the model.

```
<executable  name  =  "SRH  2D"    executable_order="1"
</executable>   <use_parameter>MyCaseFile</use_parameter>
<executable_command>        <command_args>"\"%s\"",
#file_name</command_args>        <execute>"%s  -  %s",
#executable_name,  #geom_name</execute>
</executable_command></executable><input_files>     <input_file>
<process_each_coverage>
<use_file_def>caseFile</use_file_def>
<export_location>"%s/%s",  #geom_name,
#project_name</export_location>
<declare_paramter>MyCaseFile</declare_parameter>
</process_each_coverage>    </input_file></input_files>
```

Example 3 In the following example, only one instance of the numeric model is executed. A file will be exported per coverage and no command line arguments are used when launching the model.

```
<executable  name  ="SRH  2D
executable_order="1""</executable>   <executable_command>
<execute>"%s",  #executable_name</execute>
</executable_command></executable><input_files>   <input_file>
<use_file_def>caseFile</use_file_def>     <process_each_coverage>
<export_location>"%s/%s",  #geom_name,
#project_name</export_location>    </process_each_coverage>
</input_file></input_files>
```

<executable_command>	
Info:	Defined command line arguments for the executable.
Version(s):	3
Attributes:	none
Children:	command_args , execute
Used by:	executable , anything starting with process_
Example:	See executable example.

<executable_progress_update>	
Info:	Updates the status bar of the model based on output from the model.

Version(s):	3
Attributes:	amount , max
Children:	update_text
Used by:	executable
Example:	For a model that outputs "CASE 1 of 35","CASE 2 of 35" etc.. `<executable name="My Model Main"> <executable_progress_update progress_amount="0" progress_max ="35"> <update_text>"CASE %d of %d" #progress_amount, #progress_max </update_text> </executable_progress_update> </executable>`
<execute>	
Info:	Defines when to run an executable. The text of this element is the text that will be displayed as the process name when running the model.
Version(s):	2
Attributes:	none
Children:	none
Used by:	executable_command
Example:	See executable example.
<export_format>	
Info:	Placed inside the <card> to determine how the card format will be displayed when exporting to a text file. Uses the standard printf and sprintf format (www.cplusplus.com/reference/clibrary/cstdio/printf/). "#card_name" should be used when displaying the card. When using an element that starts with "custom_control", a user can use the # character to get to the value or units.
Version(s):	1
Attributes:	ignore_on_read
Children:	none
Used by:	card , export_each_row (version 1), export_column (Version 1), export_each_coverage (version 1), export_each_polygon (version 1), export_each_arc (version 1), export_each_point (version 1), anything starting with process_each_

Example:	**Example 1**
	`<edit_box unique_name="MyEditBox"></edit_box>…<card> <card_name>FIFTH_GRAIN</card_name> <export_format>"%s %lf\n", #card_name, MyEditBox</export_format></card>`
	If the edit_box has the value of 15, this would print out: FIFTH_GRAIN 15
	Example 2 If export_format was changed to this: <export_format>"%s \"%lf\" // comment\n", #card_name, MyEditBox </export_format>
	FIFTH_GRAIN "15" // comment
	`<control_length unique_name="MyLength">`
	Example 3
	`...</control_length>…<card> <card_name>ADAPTATION_LENGTH_TOTAL</card_name> <export_format>"%s %lf, %s\n", #card_name, MyLength#value, MyLength#units</export_format></card>`

	ADAPTATION_LENGTH_TOTAL 25 cm
<export_group>	
Info:	A way of grouping optional exports.
Version(s):	2
Attributes:	ignore_on_read , required
Children:	Any element beginning with "process_" and required (version 2), xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry , export_optional , section
Used by:	Any element beginning with "process_".
Example:	
<export_location>	
Info:	Relative or absolute path of the export file (.cmcards). Can be used multiple times if exporting to more than 1 file location. Keywords can be #PROJECT_NAME and #GEOMETRY.
Version(s):	1, 2
Attributes:	none
Children:	none
Used by:	group , card , input_file (version 2), output_file (version 2). If it used just within group then everything within the element will use the same export location unless specified in the child group or item card.
Example:	**Example 1** <export_location>\\Ideal__Inlet.cmcards</export_location> **Example 2** <export_location>\\#PROJECT_NAME_#GEOMETRY.cmcards</export_location> <export_location>"\\%s__%s.cmcards", #PROJECT_NAME, #GEOMETRY</export_location>
<export_optional>	
Info:	Indicator of text that might be included. If it has default as a child, then SMS will write out the text included in this element.
Version(s):	2
Attributes:	ignore_on_read , default
Children:	Any element beginning with "process_", export_format , default (version 2), separator , xmdf_dataset, xmdf_data, xmdf_group, xmdf_geometry, section
Used by:	Any element beginning with "process_".
Example:	
<files>	
Info:	
Version(s):	2
Attributes:	
Children:	file_def
Used by:	model_control_executable

Example:	
<group>	
Info:	Tree group item that contains one or more items or groups.
Version(s):	1
Attributes:	text
Children:	item (version 1), dependency , text_style , contains (version 2)
Used by:	link_to (version 1), page_def (version 2), another <group> if nested (version 1), <contains> (version 2)
Example:	<group text = "Timing"> <item> … tree item stuff </item></group>

Elements I - O

<input_file>	
Info:	Defines a file to be used for input for an executable.
Version(s):	2
Attributes:	none
Children:	use_file_def , export_location , process_each_row , process_each_coverage , process_each_material , declare_parameter
Used by:	input_files
Example:	See executable example.
<input_files>	
Info:	A container to place all of the input files into for a simulation.
Version(s):	3
Attributes:	none
Children:	input_file
Used by:	simulation
Example:	
<item>	
Info:	Tree item contains one or more widgets (combo_box , text_box , edit_box , etc). A tree item can contain multiple cards (version 1), but those cards must be mutually exclusive.
Version(s):	1
Attributes:	text
Children:	card , dependency , text_box , edit_box , check_box , combo_box , table , any element starting with "control", text_style , new_line
Used by:	group
Example:	<item> <text>Transport Time Step</text> <edit_box>…. </edit_box> <card>…</card>

<location>	
Info:	Used to specify which point locations to export. Point locations are defined as: "corner", "mid", "center", "disjoint" and "all". Point locations can have a slightly different meaning based on their context as described below. Exporting coverage arc points: • mid = middle points of arc • corner = 2 end points of an arc • disjoint = not valid • center = not valid Exporting coverage polygon points: • mid = non nodes in the polygon • corner = nodes in the polygon • disjoint = not valid • center = not valid Exporting grid/quad points: • mid = midside points on the cell • corner = corner points on the cell • disjoint = not valid • center = center point on cell Exporting mesh points: • disjoint = disjoint points only • corner = end points of arcs, nodes in polygons • mid = mid points in arcs, mid points in polygons • center = not valid
Version(s):	3
Attributes:	none
Children:	none
Used by:	process_each_point , snap
Example:	// writes out snapped coordinates of the 2 end points of each arc

```
<process_each_coverage>     <use_parameter>myCov</use_parameter>
<process_each_arc  source  =  "snap">         <process_each_point
source  =  "snap">              <location>corner</location>
<export_format>"%Lf  %Lf  %Lf\n",  #point_x,  #point_y,
#point_z              </export_format>
</process_each_point>
</process_each_arc></process_each_coverage>//  writes  out  all
points  of  each  arc<process_each_coverage>
<use_parameter>myCov</use_parameter>     <process_each_arc
source  =  "snap">         <process_each_point  source  =
"snap">              <location>corner</location>
<location>mid</location>             <export_format>"%Lf  %Lf
```

	```%Lf\n", #point_x, #point_y, #point_z </export_format> </process_each_point> </process_each_arc></process_each_coverage> // writes disjoint nodes in a mesh<process_each_geometry> <use_parameter>myMesh</use_parameter> <process_each_point source = "geometry"> <location>disjoint</location> <export_format>"%Lf %Lf %Lf\n", #point_x, #point_y, #point_z </export_format> </process_each_point></process_each_geometry>// writes all midpoints in a grid<process_each_geometry> <use_parameter>myGrid</use_parameter> <process_each_point source = "geometry"> <location>mid</location> <export_format>"%Lf %Lf %Lf\n", #point_x, #point_y, #point_z </export_format> </process_each_point></process_each_geometry>```
<material_att>	
Info:	Used to specify what attributes should be used for materials of a coverage.
Version(s):	2
Attributes:	use_dialog
Children:	none
Used by:	declare_coverage
Example:	See coverage example.
<max_rows>	
Info:	Optional element used to limit the number of rows the user can specify in a control_curve. If this element is not used, the max_rows is unlimited.
Version(s):	2
Attributes:	none
Children:	none
Used by:	control_curve
Example:	This example creates a curve button with the date/time flag. ```<item> <control_curve> <unique_name>myCurve1</unique_name> <max_rows>10</max_rows> <x_column> <text>Time</time> </x_column> <y_column> <text>Velocity</text> </y_column> </control_curve> …```
<menu_item>	
Info:	Item to be placed in a menu. The text is the text displayed in the menu, use_dialog describes the dialog to be launched, dependency describes what needs to be true in order for the menu item to appear. The double_click attribute (if true) indicates that this is the menu item to be launched on a double click event. For example, if a menu_item is in an arc_att, and has double_click = "true", then when an arc is double clicked, the dialog specified by the menu item is launched.
Version(s):	2
Attributes:	double_click , use_dialog , text
Children:	dependency

Used by:	simulation , attribute_set (version 3), declare_coverage , point_att , arc_att , polygon_att , material_att
Example:	See the model example.
<msg>	
Info:	Used to display text to the user when a model check condition has failed.
Version(s):	(Version 3 replaced with problem_text, description_text and fix_text)
Attributes:	none
Children:	none
Used by:	model_check
Example:	< table > <column text="A"> <edit_box unique_name = "colA"> </edit_box> </column>…// other columns also defined<model_check problem_text = "Column D or E is required." description_text = "This model requires column D." fix_text = "To fix do the following: …... "> <condition>((colA OR colB) AND NOT(colD OR colE))</condition> </model_check>< /table >
<model>	
Info:	Defines the model interface that is being created.
Version(s):	2
Attributes:	name , version
Children:	simulation , declare_coverage , dialogs , files
Used by:	dynamic_model
Example:	A simple model that has one coverage type, one dialog, and one file definition. This interfaces with version 4 of "MyModel". <dynamic_model filetype="dynamic model"> version="2"> <model name="MyModel" version="4"> <simulation> … </simulation> <declare_coverage name= "MyCov">1</coverage> ... </coverage> <declare_dialogs> … </declare_dialogs> <file_def> …
<model_check>	
Info:	Used to validate data of multiple combinations when the **OK** button is clicked. Displays an error message if the logic in the <condition> is false. Can use AND, OR, LESS_THAN, GREATER_THAN, EQUALS and NOT.
Version(s):	1
Attributes:	problem_text , description_text , fix_text
Children:	condition , process_each_coverage , process_each_geometry
Used by:	item (version 1), model_checks (version 2)
Example:	< table > <column text="A"> <edit_box unique_name="colA"> </edit_box> </column>…// other columns also defined< /table ><model_check problem_text = "Column D or E is required." description_text = "This model

	`requires column D."                 fix_text = "To fix do the following: …...   ">    <condition>( (colA EQUALS 0.0 OR colB EQUALS 0.0) AND NOT(colD EQUALS 0.0 OR colE EQUALS 0.0))</condition></model_check>`
<model_checks>	
Info:	Contains all model checks
Version(s):	2
Attributes:	none
Children:	model_check
Used by:	
Example:	<declare_dialogs> `<declare_dialog> ... < table > <column text="A"> <edit_box unique_name="colA"> </edit_box> </column>…// other columns also defined < /table > ... <declare_dialog><declare_dialogs><model_checks><model_check problem_text = "Column D or E is required." description_text = "This model requires column D." fix_text = "To fix do the following: …... "> <condition>((colA EQUALS 0.0 OR colB EQUALS 0.0) AND NOT(colD EQUALS 0.0 OR colE EQUALS 0.0))</condition> </model_check><model_checks>`
<new_line>	
Info:	Creates a new line before adding the next widget in a tree item.
Version(s):	1
Attributes:	none
Children:	none
Used by:	item
Example:	In the dialog there would be: `Breaking Efficency:   [edit box]         (new_line called) Friction Efficency:` `<item>   <text>Non-cohesive bedload entrainment</text> <text_box>           <text>Breaking Efficency:</text> </text_box>         <edit_box  unique_name="edtBreakingEff"> </edit_box>         <new_line></new_line>         <text_box text="Friction Efficency:">         </text_box>`
<option>	
Info:	Widget that displays list of options. Only 1 can be selected.
Version(s):	1
Attributes:	default , text , unit_keyword , display_options_hide
Children:	export_text , dependency , text_style
Used by:	combo_box , any element that starts with "custom_control" and has units
Example:	Creates a combo box with hours, minutes and seconds. Minutes is the default item. `<combo_box  unique_name="cbxTransportUnits"> <option  text="hours">        </option>        <option text="minutes"                          default> </option>        <option  text="seconds">        </option>`

	`<dependency>…</dependency>…</  combo_box  >`
<output_file>	
Info:	Defines a file to be created by the executable.
Version(s):	1
Attributes:	none
Children:	use_file_def, export_location , export_table , export_each_coverage , export_each_material
Used by:	executable
Example:	See executable example.
<output_files>	
Info:	A container to place all of the output files into for a simulation.
Version(s):	3
Attributes:	none
Children:	output_file
Used by:	simulation
Example:	See executable example.

Elements P - S

<page_def>	
Info:	This defines the page or tab of a dialog. If there is only one page_def in a dialog, then no tabs will appear.
Version(s):	2 (depreciated use <delcare_page>)
Attributes:	name , display
Children:	contains
Used by:	dialog
Example:	
<point_att>	
Info:	Used to specify what attributes should be used for the points of a coverage.
Version(s):	2
Attributes:	none
Children:	menu_item , snap
Used by:	coverage
Example:	
<polygon_att>	
Info:	Used to specify what attributes should be used for the polygon of a coverage.
Version(s):	2
Attributes:	none

Children:	menu_item , snap
Used by:	coverage
Example:	
<process_each_arc>	
Info:	Indicator to loop through each arc in the given context.
Version(s):	2
Attributes:	ignore_on_read , order , i_order , j_order , source
Children:	card_name , export_format , separator , process_each_point , end_card , use_parameter , declare_parameter , executable_command , section , export_group , export_optional , process_on_condition , process_each_neighbor , xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry
Used by:	card , process_each_coverage , process_each_polygon
Example:	
<process_each_coverage>	
Info:	Indicator to loop through each coverage in the given context.
Version(s):	2
Attributes:	ignore_on_read , order , i_order , j_order , source
Children:	card_name , export_format , separator , process_each_point , process_each_polygon , process_each_arc , end_card , use_parameter , declare_parameter , process_each_set , executable_command , export_group , export_optional , process_on_condition , process_each_material , xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry , count_range, section , location
Used by:	card
Example:	
<process_each_material>	
Info:	Indicator to loop through each material in the given context.
Version(s):	2
Attributes:	ignore_on_read , order , i_order , j_order , source
Children:	card_name , export_format , separator , end_card , use_parameter , declare_parameter, process_each_polygon , export_group , export_optional , process_on_condition , process_each_polygon , section , count_range, xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry
Used by:	card , process_each_coverage , process_each_polygon
Example:	
<process_each_neighbor>	
Info:	Indicator to loop through each neighbor in the given context.
Version(s):	3
Attributes:	corner_skip , feature , ignore_on_read , order , i_order , j_order , interior_polygon , null_id , neighbor_per_edge , source

Children:	card_name , export_format , separator , end_card , use_parameter , declare_parameter , process_each_polygon , export_group , export_optional , process_on_condition , process_each_arc , process_each_polygon , process_each_point , section , xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry
Used by:	card , process_each_coverage , process_each_polygon
Example:	
<process_each_point>	
Info:	Indicator to loop through each point in the given context.
Version(s):	2
Attributes:	ignore_on_read , order , i_order , j_order , source, dataset_widget
Children:	card_name , export_format , separator , end_card , use_parameter , declare_parameter , executable_command , location , export_group , export_optional , process_on_condition , section , process_each_ neighbor , xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry
Used by:	card , process_each_coverage , process_each_polygon , process_each_arc
Example:	
<process_each_polygon>	
Info:	Indicator to loop through each polygon in the given context.
Version(s):	2
Attributes:	ignore_on_read , order , i_order , j_order , source
Children:	card_name , export_format , separator , process_each_point , process_each_arc , end_card , use_parameter , declare_parameter , executable_command , export_group , export_optional , process_on_condition , section , process_each_neighbor , process_each_material , xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry
Used by:	card , process_each_coverage
Example:	
<process_each_row>	
Info:	Indicator to loop through each row in a table. Has the widget attribute that must be used. The widget attribute needs to be set to the name of a widget which belongs to the table to be iterated through. To iterate through the rows of a curve widget, use the name of the curve widget and a keyword, such as "#x".
Version(s):	2
Attributes:	ignore_on_read , widget, order , i_order , j_order
Children:	card_name , export_format , separator , export_group , export_optional , process_on_condition , section , process_each_polygon , process_each_material , process_each_arc , process_each_point , executable_command , xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry
Used by:	card , process_each_coverage , process_each_polygon , process_each_arc , process_each_point
Example:	<process_each_row widget = "someColumnWidget">
<process_on_condition>	

Info:	Indicator to perform a certain action only if the condition is satisfied.
Version(s):	2
Attributes:	ignore_on_read
Children:	process_each_coverage , process_each_geometry , process_each_polygon , process_each_arc , process_each_point , card_name , export_format , separator , condition , use_file_def , xmdf_dataset , xmdf_data , xmdf_group , use_parameter , end_card , xmdf_geometry , export_location , export_group , export_optional , process_on_condition , section , process_each_row , process_each_material , process_each_neighbor , process_on_count , process_model
Used by:	card , process_each_coverage , process_each_polygon , process_each_arc , process_each_point
Example:	
<process_on_count>	
Info:	Indicator to perform a certain action only on certain iterations of a loop.
Version(s):	2
Attributes:	ignore_on_read
Children:	process_on_condition , process_each_geometry , process_each_material , section , export_optional , export_group , process_each_coverage , process_each_polygon , process_each_arc , process_each_point , card_name , export_format , separator , count_filter , xmdf_dataset , xmdf_data , xmdf_group , xmdf_geometry
Used by:	card , process_each_coverage , process_each_polygon , process_each_arc , process_each_point
Example:	
<push_button>	
Info:	The button to be used for a control_dataset. Can be SELECT, CREATE, or DELETE.
Version(s):	1, 2
Attributes:	none
Children:	none
Used by:	control_dataset
Example:	See control_dataset example.
<range>	
Info:	Determines the range of acceptable values in an edit_box . Can use the keywords GREATER_THAN, GREATER_THAN_EQUALS, LESS_THAN and LESS_THAN_EQUALS. Numbers assumed inclusive unless otherwise specified.
Version(s):	1
Attributes:	none
Children:	none
Used by:	edit_box , any element that starts with "control" except: control_file_selector, control_dataset and control_date

Example:	**Example 1**
	`<edit_box> <range>0-1</range></ edit_box >`
	Example 2
	`<range>0 - 5.6546</range>`
	Example 3
	`<range>GREATER_THAN 0</range>`
	Example 4
	`<range>GREATER_THAN 0.0 - LESS_THAN 5.0</range>`
	Example 5 Both examples are equivalent
	`<range>0.0 - LESS_THAN 5.0</range><range>GREATER_THAN_EQUALS 0.0 - LESS_THAN 5.0</range>`

<required>	
Info:	Used to indicate that something from the exported group must be used.
Version(s):	2
Attributes:	none
Children:	none
Used by:	export_group
Example:	
<role>	
Info:	A single entry in a combo box option for <roles>. Used to define the displayed text and range. The range specifies how many objects that can have this role.
Version(s):	3
Attributes:	none
Children:	text , range
Used by:	roles
Example:	Example of a culvert widget with a extra "Name" column `<control_set unique_name="tblSetCulverts"> <roles behavior = "SWAP"> <role text="upper"> <range>1-1</range> </role> <role text="lower"> <range>1-1</range> </role> </roles> <column text = "Name"> <edit_box unique_name="edtMyArcsName" type="text"> </edit_box> </column></control_set>`
<roles>	
Info:	Defines the behavior of each role.
Version(s):	3
Attributes:	behavior
Children:	role
Used by:	control_set
Example:	Example of a culvert widget with a extra "Name" column `<control_set unique_name="tblSetCulverts"> <roles`

	`behavior = "SWAP"> <role text="upper"> <range>EQUALS 1</range> </role> <role text = "lower"> <range>EQUALS 1</range> </role> </roles> <column text="Name"> <edit_box unique_name="edtMyArcsName" type="text"> </edit_box> </column></control_set>`
<row>	
Info:	Typically you would only use row if you had a fixed table and wanted to display row text or to specify specific rows as read only. Row tags are placed inside a < table > .
Version(s):	1
Attributes:	text , read_only
Children:	optional , dependency , text_box , combo_box , edit_box , all elements starting with "custom_control", check_box , text_style
Used by:	table
Example:	Row 1 is read only, row 2 isn't `< table > … <row text = "Row1" read_only></row> <row text = "Row2"></row>< /table >`
<separator>	
Info:	Determines how to separate text that is being exported into a text file.
Version(s):	1
Attributes:	none
Children:	none
Used by:	export_each_row
Example:	See <export_table>
<simulation>	
Info:	The simulation object that represents what will run and be exported upon launching a model. There can only be one per model.
Version(s):	2
Attributes:	use_icon
Children:	takes , executable , menu_item , input_files , output_files
Used by:	model
Example:	See the model example.
<snap>	
Info:	Snaps the arc, point or polygon in a coverage to the underlying geometry (mesh, grid, etc) that is defined by the <declare_coverage> bind_to attribute
Version(s):	3
Attributes:	none
Children:	snap_exterior , snap_interior , location
Used by:	arc_att , point_att , polygon_att
Example:	Snaps an arc boundary condition to the interior of the mesh when the arc type is Monitor

	Line and to the exterior when the arc is a Inlet-Q. `<declare_coverage name= "Boudary Conditions" bind to ="MESH2D"> <arc_att> <menu_item double_click = "true" text = "Assign Linear BC" link_to = "Linear BC" </menu_item> <snap> <snap_exterior> <condition>cbxLineType EQUALS "Inlet-Q" </condition> </snap_exterior> <snap_interior> <condition>cbxLineType EQUALS "Monitor-Line" </condition> </snap_interior> </snap> </arc_att></coverage>`
<snap_exterior>	
Info:	Snaps the arc, point or polygon in a coverage to the closest exterior point, arc, or polygon of the underlying geometry.
Version(s):	3
Attributes:	none
Children:	condition
Used by:	snap
Example:	See <snap>
<snap_interior>	
Info:	Snaps the arc, point or polygon in a coverage to the closest interior point, arc, or polygon of the underlying geometry.
Version(s):	3
Attributes:	none
Children:	condition
Used by:	snap
Example:	See <snap>.

Elements T - Z

<table>	
Info:	Table widget with columns and rows. Will most likely have multiple column and row tags (version 3). Using the attribute fixed_row_count means that the user can't add or delete rows. Not using the fixed_row_count , an insert and delete button will be included at the bottom of the table.
Version(s):	1
Attributes:	read_only , fixed_row_count (version 3), max_row_count (version3), min_row_count (version 3), min_height (version3), unique_name
Children:	column , row , dependency , text_style
Used by:	item
Example:	(version 3) Read only table with 2 columns and 3 rows. The number of rows are fixed. `< table unique_name = "myTable" read_only fixed_row_count="3"> <column text = "hours"</column> <column text = "minutes"</column></ table >`
<take_coverage>	

Info:	Represents a coverage that can be taken in the project explorer.
Version(s):	2
Attributes:	limit
Children:	declare_parameter, condition , use_coverage
Used by:	simulation , attribute_set(version 3)
Example:	See takes example.
<take_mesh2d>	
Info:	Represents a coverage that can be taken in the project explorer.
Version(s):	2
Attributes:	limit, linear
Children:	declare_parameter, condition
Used by:	simulation , attribute_set(version 3)
Example:	See takes example.
<takes>	
Info:	Represents what can be dragged under another object in the project explorer.
Version(s):	2
Attributes:	none
Children:	declare_parameter, take_coverage , take_mesh2d , take_grid2d
Used by:	simulation , attribute_set(version 3)
Example:	**Example 1** This example shows a simulation that can take a coverage and a 2D mesh which uses linear elements. `<takes> <take_coverage limit = "1"> <use_coverage>My Cov</use_coverage> <condition>""</condition> </take_coverage> <take_mesh2d limit = "1" linear = "true"> <condition>""</condition> </take_mesh2d> <declare_parameter>meshCov</declare_parameter></takes>` **Example 2** This example shows taking one type of coverage OR another type of coverage. `<takes> <take_coverage limit = "1"> <use_coverage>My Cov</use_coverage> <use_coverage>My Other Cov</use_coverage> <condition>""</condition> </take_coverage> <declare_parameter>monitorCov</declare_parameter></takes>`
<text_box>	
Info:	Widget that displays text that is not editable.
Version(s):	1
Attributes:	alignH, alignV, text, unique_name
Children:	dependency , text_style
Used by:	item , row , column
Example:	<text_box unique_name = "edtD50Units" alignH= "CENTER" text = "mm"> `</text_box>`

<text_size>	
Info:	Sets the point size of the text. Ranges from 8 to 20. Added in SMS 11.2.
Version(s):	1, 2
Attributes:	none
Children:	none
Used by:	text_style

Example:	This would set the size of the text to 20pt. when the value in the edit_box is less than 0.0.
	`<edit_box>   <unique_name>edtA</unique_name>`
	`<default>1.0</default>   <text_style>`
	`<text_size>20</text_size>       <condition>edtA  LESS_THAN`
	`0.0</condition>   <text_style></edit_box>`

<text_style>	
Info:	Holds the style options for the text. Added in SMS 11.2.
Version(s):	1
Attributes:	bold, italic, strike_through, text_size, underline
Children:	color, condition
Used by:	group , item , text_box , combo_box , edit_box , all elements starting with "custom_control", check_box , options

Example:	This would bold the text when the value in the edit_box is less than 0.0.
	`<edit_box>  <unique_name>edtA</unique_name>`
	`<default>1.0</default>  <text_style    bold    text_size="12">`
	`<condition>edtA  LESS_THAN  0.0</condition>`
	`<text_style></edit_box>`

<title_format>	
Info:	This indicates what the title of an object in an h5 file is. Uses the standard printf and sprintf format ([www.cplusplus.com/reference/clibrary/cstdio/printf/ www.cplusplus.com/reference/clibrary/cstdio/printf/]). Works the same as export_format .
Version(s):	3
Attributes:	none
Children:	none
Used by:	h5_group, h5_geometry, h5_dataset, h5_data
Example:	
<unique_link_name>	
Info:	A unique name given to a link when using link_to_xml. This is needed to identify the unique_name elements from the file, especially when a linked to XML file is used multiple times. Hence, when using link_to_xml, this tag is required. No spaces or punctuation are allowed in the name.
Version(s):	1, 2
Attributes:	none
Children:	none

Used by:	link_to_xml
Example:	See link_to_xml example.
<update_text>	
Info:	Used to match the output text from a model to determine the progress amount.
Version(s):	3
Attributes:	none
Children:	none
Used by:	executable_progress_update
Example:	Lets say our model outputs "CASE 1 of 35" then "CASE 2 of 35" etc.. <executable name="My Model Main"</executable> <executable_progress_update amount="0" max ="35"> <update_text>"CASE %d of %d" #progress_amount, #progress_max </update_text> </executable_progress_update> </executable>
<use_coverage>	
Info:	Use a coverage that has previously been defined in the attribute type in <declare_coverage>
Version(s):	2
Attributes:	none
Children:	none
Used by:	declare_coverage , take_coverage
Example:	See the coverage example and the takes example.
<use_file_def>	
Info:	A way to access file data that has been previously declared with <declare_file_def>.
Version(s):	2
Attributes:	control_file
Children:	output_file , input_file
Used by:	file
Example:	
<use_parameter>	
Info:	A way to access file data that has been previously declared with <declare_parameter>.
Version(s):	3
Attributes:	none
Children:	none
Used by:	anything beginning with "process_each", card , control_dataset
Example:	
<x_column>	
Info:	Used to define the attributes in the x column such as text (heading).

Version(s):	2
Attributes:	text
Children:	none
Used by:	control_curve
Example:	This example creates a curve button with the date/time flag. `<item>    <control_curve> <unique_name>myCurve1</unique_name> <max_rows>10</max_rows>        <x_column> <text>Time</time>        </x_column>        <y_column> <text>Velocity</text>                                 </y_column> </control_curve>  …`
<xmdf_data>	
Info:	Writes data to the XMDF file. The attributes rows, columns and layers indicate what information should be written for each row, column or layer respectively. By default, the attributes are set to "#specified". If no attributes are set then each export_format will be assumed to be a new row. The attribute columns can only be set if the rows attribute is set. The attribute layers can only be set if both the rows attribute and the columns attribute is set. The datatype attribute can be set to "float", "double", "integer", or "text". If the datatype attribute is not set or is set to "text" then export_format can be used. Otherwise, only export can be used. If more than 1 dimension is used, then the datatype cannot be "text". If 3 dimensions are used, then the datatype cannot be "integer".
Version(s):	3
Attributes:	datatype
Children:	export_format , title_format , export
Used by:	xmdf_group , file_def, and anything beginning with "process_each".
Example:	This writes out 6 different widgets' values, one on each row. `<xmdf_data datatype = "text"> <export_format>"%s", widget1</export_format> <export_format>"%s", widget2</export_format> <export_format>"%s", widget3</export_format> <export_format>"%s", widget4</export_format> <export_format>"%s", widget5</export_format> <export_format>"%s", widget6</export_format></xmdf_data>` This writes out a table with each row being a different timestep and each column a different point. Each export would be written to different layers. `<xmdf_data rows = "#timestep" columns = "#point" datatype = "double"> <export>#component1</export> <export>#component2</export></xmdf_data>`
<xmdf_dataset>	
Info:	Writes out all of the data of a dataset in an SMS friendly format to an xmdf file. The attribute null_value is optional. If used, it specifies a numeric value that will be treated by the model as inactive. By default, SMS will write an activity array. The attribute location specifies where data should be taken from. The attribute widget specifies to only write the dataset referenced by the named widget. If no title_format is used, then the name of the dataset will be used.
Version(s):	3
Attributes:	null_value , location , widget

Children:	title_format
Used by:	xmdf_group , xmdf_geometry , process_each_coverage .
Example:	This writes out the hardbottom dataset as specified by the widget edtHardBottomWidget that exists in some dialog. Any value that is inactive is written as -999.0. `<xmdf_dataset widget = "edtHardBottomWidget" location = "all" null_value = "-999.0"> <title_format>"Hardbottom"</title_format></xmdf_dataset>`
<xmdf_dimension>	
Info:	
Version(s):	3
Attributes:	dimension
Children:	
Used by:	xmdf_data
Example:	**Example 1:** The following shows how to write out the text rows of a single column of a table into a single column XMDF dataset. `<xmdf_group> <title_format>"Test"</title_format> <xmdf_data> <title_format>"Table"</title_format> <process_each_row widget="myColumn1"> <xmdf_dimension dimension="rows"> <export_format>"%s", myColumn1</export_format> </xmdf_dimension> </process_each_row> </xmdf_data></xmdf_group>` **Example 2:** The following shows how to write out two columns of a table into two columns of single XMDF dataset. It should be noted that multi-column XMDF datasets may not be text. `<xmdf_group> <title_format>"Test"</title_format> <xmdf_data datatype="double"> <title_format>"Table"</title_format> <process_each_row widget="myColumn1"> <xmdf_dimension dimension="rows"> <xmdf_dimension dimension="columns"> <export>myColumn1</export> </xmdf_dimension> <xmdf_dimension dimension="columns"> <export>myColumn2</export> </xmdf_dimension> </xmdf_dimension> </process_each_row> </xmdf_data></xmdf_group>`
<xmdf_geometry>	
Info:	Writes a geometric object in an SMS friendly format to an XMDF file. Different geometric types will be written differently from each other. If no title_format is used, then the name of the geometry will be used.
Version(s):	3
Attributes:	none
Children:	title_format , xmdf_dataset
Used by:	file_def, xmdf_group
Example:	
<xmdf_group>	
Info:	Writes an XMDF group to the xmdf file. A title_format must be used. The group_type attribute must be used.

Version(s):	3
Attributes:	group_type
Children:	xmdf_group, xmdf_geometry , xmdf_dataset , xmdf_data , title_format , and anything beginning with "process_each".
Used by:	file_def, xmdf_group, xmdf_geometry , xmdf_dataset , and anything beginning with "process_each".
Example:	
<y_column>	
Info:	Used to define the attributes in the y column such as text (heading).
Version(s):	2
Attributes:	text
Children:	none
Used by:	control_curve
Example:	See example in x_column

Attributes

Attributes A-C

<alignH>	
Info:	Align text horizontally in a text box.
Values:	bottom, top, center
Version(s):	1
Used by:	text_box
<alignV>	
Info:	Align text vertically in a text box.
Values:	bottom, top, center
Version(s):	1
Used by:	text_box
<amount>	
Info:	
Values:	integer
Version(s):	3
Used by:	execute_progress_update
<behavior>	
Info:	When a user changes a role, you can define behaviors that automatically update the roles.
Values:	*None* : no enforcement of rules in the dialog

	Swap : all assigned up front. When some object is being assigned a role and is there are already a max number of objects assigned to the role, then any objects that had that role will be switched to the old role of the object being asgined until we surpass the maximum amount of that role, then unassigned. *Max_hide* : Don't allow the role to show up in the combo box to be assigned to a "object" if the max has been reached for that role. *Swap_max_default* : (not implemented yet) Enforce max with default is like swap, but instead of the old option, it changes it to the default or unassigned (if no default provided) value. Chooses the first or last "object" of the role it encounters to change.
Version(s):	3
Used by:	roles
<bind_to>	
Info:	Binds a coverage to a mesh, or grid
Values:	MESH2D, CGRID
Version(s):	3
Used by:	declare_coverage
<blue>	
Info:	A color. Ranges from 0 to 255. Added in SMS 11.2. See also green and red.
Values:	integer (0 to 255)
Version(s):	1
Used by:	color
<bold>	
Info:	Sets the text to be bolded when this attribute is used. Added in SMS 11.2.
Values:	none
Version(s):	1
Used by:	text_style
<columns>	
Info:	Represents a column in the H5 file that we are mapping to
Values:	#timestep, #point, #arc, #polygon, #coverage, #material
Version(s):	3
Used by:	xmdf_data
<corner_skip>	
Info:	Skip corners when exporting neighbors
Values:	true, false
Version(s):	3
Used by:	process_each_neighbor
<create_file>	

Info:	If attribute is specified, then create_file is true
Values:	none
Version(s):	3
Used by:	control_file_selector
<dataset_type>	
Info:	Specify the dataset type as scalar or vector. Default is scalar.
Values:	scalar, vector
Version(s):	1
Used by:	control_dataset
<dataset_widget>	
Info:	When exporting points from a dataset, the attribute identifies the dataset widget being used (contains the selected dataset).
Values:	Name of the dataset_widget
Version(s):	3
Used by:	process_each_point
<datatype>	
Info:	
Values:	float, double, integer, text
Version(s):	3
Used by:	xmdf_data
<default>	
Info:	Sets the initial default value of a widget. Can also be used to state the default combo-box option.
Values:	*edit_box* or any element that begins with *custom_control* : double *check_box* : checked, unchecked *combo_box* : no values or empty string *export_optional* : true, false
Version(s):	1
Used by:	edit_box , check_box , option , any element that begins with "control" ("custom_control" in version 1), combo_box , export_optional
<default_executable_name32>	
Info:	Sets the initial default file name for the 32-bit executable.
Values:	text
Version(s):	3
Used by:	executable
<default_executable_name64>	
Info:	Sets the initial default file name for the 64-bit executable.

Values:	text
Version(s):	3
Used by:	executable
<description_text>	
Info:	A detailed description of why a model check failed. This text is displayed to the user.
Values:	string
Version(s):	3
Used by:	model_check
<dim>	
Info:	Specifies that when the dependency is false, the widget should be dimmed, not hidden.
Values:	none
Version(s):	1
Used by:	dependency
<dimension>	
Info:	Specifies values for rows, columns and layers.
Values:	rows, columns, layers
Version(s):	3
Used by:	xmdf_dimension
<display>	
Info:	This is used to describe how the dialog should display. *flex* is the default. In this mode, there is a tree on the left hand side of the dialog. Clicking on an item in the tree will display that portion of the tree along with any child items below the clicked item. Other items will be hidden on the right hand side. *full* will show a tree on the left hand side of the dialog. Clicking on an item in the tree will move the scroll bars on the right hand side to make the item visible. All items that are not disabled due to dependencies are shown on the right hand side of the dialog in this mode. *no_nav* will have no left hand side tree. Instead, all items that are not disabled due to dependencies are shown in this mode. Added in SMS 11.2.
Values:	flex, full, no_nav
Version(s):	1
Used by:	declare_page
<display_options_hide>	
Info:	Don’t show a specific combo-box option in the display option dialog of SMS. This only applies to <combo_box> ‘s that have included the <display_options> element. Default value is false.
Values:	true, false
Version(s):	3
Used by:	option

<double_click>	
Info:	The double_click attribute (if true) indicates that this is the menu item to be launched on a double click event.
Values:	true, false
Version(s):	2
Used by:	menu_item

Attributes E-L

<executable_order>	
Info:	Defines the order in which this executable should run relative to other executables in the same simulation.
Values:	integer ≥ 1
Version(s):	2
Used by:	executable
<feature>	
Info:	Determine whether to select points, arcs or polygons (poly).
Values:	point, arc, poly
Version(s):	2
Used by:	control_feature_selector , process_each_neighbor
<fix_text>	
Info:	A brief summary of what steps to take to fix a model check failure. This text is displayed to the user.
Values:	string
Version(s):	3
Used by:	model_check
<file_type>	
Info:	Specifies a file type of <declare_file_def> defines the file type.
Values:	CARD_ASCII, SEQUENTIAL_ASCII, SEQUENTIAL_BINARY, XMDF
Version(s):	2
Used by:	declare_file_def, section
<filter>	
Info:	When selecting a file, used to filter the available files by the extension type. The filter tag contains two parts. The first part is the text describing the filter that gets displayed to the user. The second part is a pair of parentheses that contains the actual filter. For example: Cmcards file (.cmcards)
Values:	string (string)
Version(s):	3

Used by:	control_file_selector
<flags>	
Info:	Flags are optional, and are used to modify the behavior/appearance of the control_curve. Only 1 flag is currently defined: XY_USEDATE. When this flag is set, the x column becomes a date/time calendar.
Values:	XY_USEDATE
Version(s):	2
Used by:	control_curve
<footer>	
Info:	Exported text inside a table that is placed at the end.
Values:	string
Version(s):	2
Used by:	export_table , export_each_row , export_column
<geometry>	
Info:	The default behavior is to only allow geometric objects that are contained under a simulation to be a candidate for selection. Otherwise "all" geometries are candidates.
Values:	all
Version(s):	2
Used by:	control_dataset
<green>	
Info:	A color. Ranges from 0 to 255. Added in SMS 11.2. See also blue and red.
Values:	integer (0 to 255)
Version(s):	1
Used by:	color
<header>	
Info:	Exported text inside a table that is placed at the beginning.
Values:	string
Version(s):	1
Used by:	export_table , export_each_row , export_column
<help_button_url>	
Info:	Provides online help when the help button is clicked. Loads the specified url in a web browser. Example: help_button_url="www.aquaveo.com"
Values:	string
Version(s):	2
Used by:	declare_dialog
<help_button_wiki>	

Info:	Provides online help when the help button is clicked. Loads the specified url in a web browser. Example: help_button_wiki="DynSrhModelControl"
Values:	string
Version(s):	2
Used by:	declare_dialog
<increment>	
Info:	An increment values. Used Increment from the min time to the max time, using the increment.
Values:	Integer ≥ 1
Version(s):	2
Used by:	control_set
<interior_polygon>	
Info:	Only used for polygons when finding neighbors. Specify the polygon preference.
Values:	interior_first, exterior_first, interior_only, exterior_only
Version(s):	3
Used by:	process_each_neighbor
<italic>	
Info:	Sets the text to be italicized when the element is specified. Added in SMS 11.2.
Values:	none
Version(s):	1
Used by:	text_style
<i_order>	
Info:	Start at min i and go to max. Only used by cartesian grids and process_each_neighbor on quad trees.
Values:	ascending, descending
Version(s):	3
Used by:	process_each_arc , process_each_coverage , process_each_material , process_each_point , process_each_polygon , process_each_row
<j_order>	
Info:	Start at min j and go to max. Only used by cartesian grids and process_each_neighbor on quad trees.
Values:	ascending, descending
Version(s):	3
Used by:	process_each_arc , process_each_coverage , process_each_material , process_each_point , process_each_polygon , process_each_row
<layers>	
Info:	Represents a layer in the H5 file to which we are mapping.

Values:	#timestep, #point, #arc, #polygon, #coverage, #material
Version(s):	3
Used by:	xmdf_data
<location>	
Info:	The location from which we will be getting dataset values.
Values:	TBD
Version(s):	3
Used by:	xmdf_dataset

Attributes M-S

<max>	
Info:	A max value. The default is 2.147 billion.
Values:	Integer ≥ 1
Version(s):	2
Used by:	control_set , execute_progress_update
<max_row_count>	
Info:	Used to set the maximum number of rows allowed in a table or curve.
Values:	integer
Version(s):	3
Used by:	table , control_curve
<min>	
Info:	A minimum value. The default is 0 (zero).
Values:	Integer ≥ 1
Version(s):	2
Used by:	control_set
<model>	
Info:	Specify the model name of the executable
Values:	string
Version(s):	2
Used by:	control_executable
<name>	
Info:	Specify a name. When used by dialog this is required.
Values:	string
Version(s):	3
Used by:	declare_file_def, declare_dialog , model , control_dataset , declare_coverage

<neighbor_per_edge>	
Info:	Minimum number of neighbors in any direction. The default value is 0 (zero).
Values:	integer
Version(s):	3
Used by:	process_each_neighbor
<null_id>	
Info:	Integer for an id of a neighbor not found
Values:	integer
Version(s):	3
Used by:	process_each_neighbor
<null_value>	
Info:	The null_value for the dataset.
Values:	any integer, any double
Version(s):	3
Used by:	xmdf_dataset
<optional>	
Info:	Placed inside widgets to suppress a warning message from being displayed if the widgets data is empty. By default, data associated with a widget is required. If the <optional> tag is included and the widget's data is empty, a warning message won't be displayed.
Values:	none
Version(s):	2
Used by:	text_box , combo_box , edit_box , table , any element that starts with "custom_control", check_box , column , row
<order>	
Info:	The way spatial entity objects are to be sorted before iterating. Currently only available for Cartesian grids and quadtrees (process_each_neighbor).
Values:	clockwise, counter_clockwise, ij, ji
Version(s):	3
Used by:	process_each_arc , process_each_material , process_each_point , process_each_polygon , process_each_neighbor
<problem_text>	
Info:	A brief summary of why a model check failed. This text is displayed to the user.
Values:	string
Version(s):	3
Used by:	model_check
<red>	

Info:	A color. Ranges from 0 to 255. Added in SMS 11.2. See also blue and green.
Values:	integer (0 to 255)
Version(s):	1
Used by:	color
<required>	
Info:	Specifies that at least one option in the group must be present.
Values:	true
Version(s):	3
Used by:	export_group
<rows>	
Info:	Represents a row in the .h5 file to map to
Values:	#timestep, #point, #arc, #polygon, #coverage, #material
Version(s):	3
Used by:	xmdf_data
<select_time>	
Info:	
Values:	single, #range, #all
Version(s):	2
Used by:	control_dataset
<source>	
Info:	When exporting things such as point or arc locations, the source identifies the desired location such as on the coverage or on the geometry (grid, mesh)
Values:	coverage (use locations from coverage), snapped (use the snapped location on a geometry from a coverage), geometry (use locations from geometry)
Version(s):	3
Used by:	process_each_polygon , process_each_arc , process_each_point , process_each_material , process_each_coverage , process_each_neighbor
<strike_through>	
Info:	Sets the text to have a line through the middle when the element is specified. Added in SMS 11.2.
Values:	none
Version(s):	1
Used by:	text_style

Attributes T-Z

<text>	
Info:	Text that is displayed in the SMS user interface.

Values:	any text string
Version(s):	1
Used by:	group , item , text_box , check_box , option , x_column , y_column , column , row , display_options , menu_item , declare_page , executable
<type>	
Info:	For <display_options> defines point, or arc
Values:	*display_options* : arc, point *edit_box* : text, integer, double
Version(s):	2
Used by:	declare_coverage , display_options , edit_box
<time_type>	
Info:	Type of time either transient, steady state, or all. Default is all.
Values:	transient, steady state, all
Version(s):	2
Used by:	control_dataset
<underline>	
Info:	Sets the text to be underlined when this attribute is specified. Added in SMS 11.2.
Values:	none
Version(s):	1
Used by:	text_style
<unique_name>	
Info:	A unique name given to an widget, which determines how to reference the widget. This is needed if the widget is being used as a dependency (parent), or if the widget value is being exported. When a unique_name is being referenced it should be by the file (nothing if current file), then unique_name. No spaces or punctuation (except _) are allowed in the name. The name must contain at least 1 non-numeric letter. The name must be unique. Names are not case sensitive, hence "aaa" is the same as "AAA".
Values:	string (unique)
Version(s):	1
Used by:	card , text_box , combo_box , edit_box , table , all elements starting with "custom_control", check_box , text_box , control_curve
<unit_keyword>	
Info:	The units of a widget that begins with "control_" (where applicable). This provides a mapping for SMS to know the unit type.
Values:	Length: • #LEN_KM (kilometers) • #LEN_M (meters) • #LEN_FT (feet) • #LEN_CM (cm)

	• #LEN_MM (mm) • #LEN_INCH (inch) • #LEN_YD (yd) • #LEN_MILE (mile) • #LEN_UM (um) Time: • #TIME_SECONDS • #TIME_HOURS • #TIME_MINUTES • #TIME_DAYS • #TIME_WEEKS Volume: • #VFLOW_CU_FT_PER_SEC • #VFLOW_CU_M_PER_SEC
Version(s):	1
Used by:	option
<use_dialog>	
Info:	Determines which dialog definition we are linking to
Values:	string
Version(s):	2
Used by:	material_att , menu_item
<use_file>	
Info:	Complex command arguments can be built by referencing a <declare_file>.
Values:	string = a <declare_filename >
Version(s):	3
Used by:	command_args
<use_icon>	
Info:	Allows for a custom icon to be imported into sms for coverages and simulations. When a coverage or simulation is create the custom icon will be shown instead of the default one in the tree structure. The icon must be given to Aquaveo in advance. The icon dimensions are 16x16 pixels.
Values:	string = (icon_name.bmp)
Version(s):	3
Used by:	simulation , declare_coverage
<version>	
Info:	Sets the version number
Values:	integer
Version(s):	1

Used by:	model , dynamic_model
<z_is_elev>	
Info:	If the z value is elevation, set this to true. Default is false.
Values:	true, false
Version(s):	2
Used by:	declare_coverage

Converting to the Dynamic Model Interface

The Dynamic Model Interface offers more flexibility to model developers than the Generic Model Interface which was used before. Generic Model Interface files can be converted into files for the Dynamic Model Interface. Contact Aquaveo for more information.

Tutorial Links

For more information on using the Dynamic Model Interface, download the XML file tutorial:

- DMI XML File Tutorial
- DMI Tutorial Data
- DMI Sample Pre-Processing Executable
- DMI Sample Executable

7.2. File Support

XMDF

Description

XMDF is a C and Fortran language library providing a standard format for the geometry data storage of river cross-sections, 2D/3D structured grids, 2D/3D unstructured meshes, geometric paths through space, and associated time data. XMDF uses HDF5 for cross-platform data storage and compression.

Version 2.1

- Online Documentation
- XMDF library code (5.9 MB) No XMDF Unix add-on available - Last Revised July 30, 2012
- Change Log

Previous Versions

2.0

- Online Documentation
- XMDF library code (14.6 MB) No XMDF Unix add-on available - Last Revised April 30, 2012

1.9

- Online Documentation
- XMDF library code (6.68 MB) No XMDF Unix add-on available - Last Revised July 12, 2011

1.8

- Online Documentation
- XMDF library code (14.9 MB) No XMDF Unix add-on available - Last Revised April 6, 2011

1.7

- Online Documentation
- XMDF library code (6.52 MB) No XMDF Unix add-on available - Last Revised Jan 19, 2010

1.6

- Online Documentation
- XMDF library code (5.5 MB) No XMDF Unix add-on available - Last Revised Mar 13, 2009

1.4

- Online Documentation
- XMDF library code (5.2 MB) No XMDF Unix add-on available - Last Revised Feb 19, 2008

1.3

- Online Documentation
- XMDF library code (7.8 MB) XMDF Unix add-on (8.4 MB) - Last Revised Oct 4, 2006

1.2

- Online Documentation
- XMDF library code (5.5 MB)

1.1

- Online Documentation
- XMDF library code (2.5 MB)

1.0

- XMDF library code (2.7 MB)

HDF5 File Browsers and Editors

- HDF Explorer
- NCSA HDFView

Back to XMS

7.2.a. File Formats

File Formats

Files used in SMS can be grouped into the following categories:

- **Native SMS Files** – Non-model specific files SMS can read and write
- **Non-native SMS Files** – Non-model specific files SMS can read, but must convert and save to a different format

- **Model Input and Output Files** – Files created by numerical models SMS can read.
  - ADCIRC
  - ADH
  - BOUSS-2D
  - BOUSS Runup/Overtopping
  - CGWAVE
  - CMS-Flow
  - CMS-Wave
  - FESWMS
  - GenCade
  - Generic Model
  - PTM
  - RMA2
  - RMA4
  - SRH-2D
  - TUFLOW

When importing files, SMS will attempt to recognize the file type and import it correctly.

Related Topics

- File Extensions
- Importing Non-native SMS Files

2D Mesh Files *.2dm

A finite element mesh can be saved in a generic format defined by SMS, called the 2dm format. In addition, the Generic Model interface in the Mesh module uses this format to save a template definition, in addition to model parameter, material property, and boundary condition assignments for a specific simulation. When a *.2dm file is opened, the current numerical model changes to the Generic Model interface. To save a numerical model definition as a template file, first set up the template and save a *.2dm file before any nodes are created.

Mesh Cards

Card Type	MESH2D
Description	Identifies the file as a 2d mesh file. Must be the first line of the file.
Required	YES

Card Type	NUM_MATERIALS_PER_ELEM		
Description	Defines Number of Materials per Element		
Required	YES		
Format	MATERIALS Quantity		
Sample	NUM_MATERIALS_PER_ELEM 2		
Field	Variable	Value	Description
1	Quantity	+ integer	Number of Materials

Nodes

Card Type	ND		
Description	Defines the ID and location for each node of the mesh.		
Required	NO		
Format	ND id x y z		
Sample	ND 1 7.75e+005 1.10e+005 5.00e-001		
Field	Variable	Value	Description
1	id	+ integer	The ID of the node.
2-4	x,y,z	± real number	The x, y, and z coordinates of the point.
Note: The xyz positions are positive/negative one digit with eight decimal places followed by “e” (times ten to the…) positive/negative power real numbers.			

Linear Elements

Card Type	E2L		
Description	Identifies a 2-noded linear element.		
Required	NO		
Format	E2L id n_1 n_2 matid		
Sample	E2L 1 1 2 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 3	n_1 - n_2	+ integer	The ID's of nodes in the element.
4	matid	+ integer	The ID of the material assigned to the element.
Card Type	E3L		
Description	Identifies a 3-noded linear element.		
Required	NO		
Format	E3L id n_1 n_2 n_3 matid		
Sample	E3L 1 1 2 3 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 4	n_1 - n_3	+ integer	The ID's of nodes in the element.
5	matid	+ integer	The ID of the material assigned to the element.

Triangular Elements

Card Type	E3T		
Description	Identifies a 3-noded triangular element.		
Required	NO		
Format	E3T id n_1 n_2 n_3 matid		
Sample	E3T 1 1 2 3 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 4	n_1 - n_3	+ integer	The ID's of nodes in the element.
5	matid	+ integer	The ID of the material assigned to the element.
Card Type	E6T		
Description	Identifies a 6-noded triangular element.		
Required	NO		
Format	E6T id n_1 n_2 n_3 n_4 n_5 n_6 matid		
Sample	E6T 1 1 2 3 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 7	n_1 - n_6	+ integer	The ID's of nodes in the element.
8	matid	+ integer	The ID of the material assigned to the element.

- *Note:* The E3T, E4Q, E6T, E8Q and E9Q cards may be intermixed in the written order. The mesh is written by starting with one element and listing neighboring elements. The node IDs for these cards are in counterclockwise order around the element (ending with the centroid node for EQ9s). Each card will be written only if an element of the type is present in the mesh.

Quadrilateral Elements

Card Type	E4Q		
Description	Identifies a 4-noded quadrilateral element.		
Required	NO		
Format	E4Q id n_1 n_2 n_3 n_4 matid		
Sample	E4Q 1 1 2 3 4 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 5	n_1 - n_4	+ integer	The ID's of nodes in the element.

6	matid	+ integer	The ID of the material assigned to the element.
Card Type	E8Q		
Description	Identifies an 8-noded quadrilateral element.		
Required	NO		
Format	E8Q id n_1 n_2 n_3 n_4 n_5 n_6 n_7 n_8 matid		
Sample	E8Q 1 1 2 3 4 5 6 7 8 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 9	n_1 - n_8	+ integer	The ID's of nodes in the element.
10	matid	+ integer	The ID of the material assigned to the element.
Card Type	E9Q		
Description	Identifies an 9-noded quadrilateral element.		
Required	NO		
Format	E9Q id n_1 n_2 n_3 n_4 n_5 n_6 n_7 n_8 n_9 matid		
Sample	E9Q 1 1 2 3 4 5 6 7 8 9 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 10	n_1 - n_9	+ integer	The ID's of nodes in the element.
11	matid	+ integer	The ID of the material assigned to the element.

- *Note:* The E3T, E4Q, E6T, E8Q and E9Q cards may be intermixed in the written order. The mesh is written by starting with one element and listing neighboring elements. The node IDs for these cards are in counterclockwise order around the element (ending with the centroid node for EQ9s). Each card will be written only if an element of the type is present in the mesh.

Nodestrings

Card Type	NS		
Description	Identifies a nodestring.		
Required	NO		
Format	NS n_1 n_2 n_3 ... - n_n (number of nodes in nodestring)		

Sample	`NS 1 3 10 15 6 -2NS 126`
	`127 128 129 173 -194NS 1006 988 987`
	`989 968 948 931 930 929 906NS -`
	`904NS 720 701 699 686 680 664 649`
	`648 647 640NS -621`		

Field	Variable	Value	Description

n_1 - n_n (number of nodes in nodestring)	n_x	+ integer	The ID's of the nodes in the nodestring. The last node id is written as a negative number, thus signaling the nodestring's end. Multiple NS cards can be used on consecutive lines for a single nodestring.

- *Note:* The nodestring tail is denoted by a negative sign in front of the tail node ID. A nodestring may consist of more than ten nodes which constitute a file line, consequently a nodestring may extend multiple NS cards. Each sequential line should be read until the negative tail node ID is found, ending the nodestring definition. If no nodestrings are present in the mesh, this card will not be written.
- *Note:* Node IDs for all cards above are limited to six digits (i.e. 999999 maximum).

Model Parameter Definition Cards

The model control parameters, boundary conditions and material options available for a specific model are defined in the generic model parameter definition section of the 2D mesh file. The generic model parameter definition section is begun by specifying the **BEGPARAMDEF** card and ended with the **ENDPARAMDEF** card.

Starting with SMS version 11.0, all of the parameter cards follow a similar pattern and the available options for several of the types of parameters were increased. The pattern is similar for Global Parameters, **GP** , Bondary Conditions **BC** , and Materials **MAT** . More information can be found in each individual card.

Global Parameters	Boundary Conditions	Materials	Description
GP	BC	MAT	Name,Id
GP_DEF	BC_DEF	MAT_DEF	definitions
GP_OPTS	BC_OPTS	MAT_OPTS	options
GP_VAL	BC_VAL	MAT_VAL	values
(parent, so it has no dependencies)	BC_DEP	MAT_DEP	dependencies

Card Type	**BEGPARAMDEF**
Description	Identifies the beginning of the model parameter definition section of the 2D Mesh File.
Required	Required if model parameters are to be defined.
Card Type	**ENDPARAMDEF**
Description	Identifies the end of the model parameter definition section of the 2D Mesh File.
Required	Required if model parameters are to be defined.

Global Parameters

Card Type	**GM**
Description	Identifies the model name.
Required	NO
Format	GM name
Sample	GM "Gen2DM"

Field	Variable	Value	Description
1	name	string	Model name.
Note: Text is always delimited by quotation marks.			
Card Type	**SI**		
Description	Identifies the model units.		
Required	NO		
Format	SI val		
Sample	SI 0		
Field	Variable	Value	Description
1	val	boolean	• Enter 0 for Meters • Enter 1 for U.S. Survey Feet • Enter 2 for Geographic (Lat/Lon) • Enter 3 for International Feet
Card Type	**DY**		
Description	Identifies whether the model is dynamic or steady state.		
Required	NO		
Format	DY val		
Sample	DY 1		
Field	Variable	Value	Description
1	val	boolean	• Enter 0 for steady state. • Enter 1 for dynamic.
Card Type	**TU**		
Description	Identifies the model time units.		
Required	NO		
Format	TU val		
Sample	TU seconds		
Field	Variable	Value	Description
1	val	string	A string value describing the model units (days, hours, minutes, seconds, etc.).
Card Type	**TD**		
Description	Identifies the model time step and total simulation time.		
Required	NO		

Format	TD time_step total_time		
Sample	TD 20 1000		
Field	Variable	Value	Description
1	time_step	+ real number	Time step (for dynamic simulations).
1	total_time	+ real number	Total simulation run time (for dynamic simulations).
Note: TD data is written as an integer when possible or written as a real number if decimal places are not all zeroes.			
Card Type	**KEY**		
Description	Identifies the key to unlock and edit the model definition inside of the SMS interface.		
Required	NO		
Format	KEY key		
Sample	KEY "sms-gen2dm"		
Field	Variable	Value	Description
1	key	string	Case sensitive key to unlock and edit the model definition.
Card Type	• **DISP_OPTS entity** • **DISP_OPTS inactive** • **DISP_OPTS multiple**		
Description	How a specific display option... colors, line thickness, etc. • **entity** – the main attribute (will either be the node, element, or nodestring display option) • **inactive** – display option for inactive or unassigned • **multiple** – display option for multiple assigned		
Required	NO		
Format	DISP_OPTS EntityId Red Green Blue Display Pattern Width Style		
Sample	DISP_OPTS entity 1 0 0 0 1 0 1 0		
Sample	DISP_OPTS multiple 1 0 0 0 1 0 1 0		
Field	Variable	Value	Description
1	EnitityId	integer (0-2) • 0 = node • 1 = nodestring • 2 = element	id of the group it belongs too.
2	Red	integer (0-255)	red pixels
3	Green	integer (0-255)	green pixels
4	Blue	integer (0-255)	blue pixels
5	Display	boolean (0,1)	turned on = 1, turned off = 0

6	Pattern	Integer	display pattern
7	Width	Integer	width
8	Style	Integer	style pattern

Global Parameter Assignment Cards

Card Type	GP		
Description	Defines a Global Parameter Group		
Required	NO		
Format	GP Id Name Active		
Sample	GP 1 "Hydro" 1		
Field	Variable	Value	Description
1	+ integer	id	
2	name	string	name
3	active	boolean	• 0 = inactive • 1 = active
Card Type	GP_DEF		
Description	Global Parameter Defaults		
Required	NO		
Format	The format will depend up the type (field 4). Fields 5, 6, and 7 will be impacted by the choice of field 4. Note that the min and max information is only applicable to integer and double types. bool, integer, double, text, or options : GP_DEF GroupId ParamId Name Type Default Min Max curve: GP_DEF GroupId ParamId Name Type x_axis_title y_axis_title Float/Curve: GP_DEF GroupId ParamId Name Type Float_Default Float_Min Float_Max DefaultFloatOrCurve x_axis_title y_axis_titleGP param_name val		
Sample	Values depend upon type (see format). bool, integer, double, text or options : GP_DEF 1 1 "manning n" 1 0 0 10 curve: GP_DEF 1 1 "manning n" 5 "x-axis" "y-axis" Float/Curve: GP_DEF 1 1 "manning n" 6 0.2 0.0 1.0 CURVE "x-axis" "y-axis"		
Field	Variable	Value	Description
1	group Id	+ integer	id of the Global Parameter Group that it belongs too
2	param Id	string	its id
3	name	string	its name
4	type	int	0-Bool, 1-Integer, 2-Double, 3-Text, 4-Options, 5-Curve, 6-Float/Curve
5	• default • or x_axis_title (curve)	• type specific () • string	• default value • x-axis title when

			bringing up curve editor
6	• min or • y_axis_title (curve)	• type specific (min) • string	• minimum value • y-axis title when bringing up curve editor
7	max	type specific (max)	max value
8 (only float/curve)	DefaultFloatOrCurve	string either • "FLOAT" • "CURVE"	The default version float or integer that is seen
9 (only float/curve)	x_axis_title	string	x-axis title when bringing up curve editor
10 (only float/curve)	y_axis_title	string	y-axis title when bringing up curve editor
Note: The GP_OPTS card is only written when the previous GP_DEF card has a parameter type of four. The default value of the option parameter should be one of the options list in the GP_OPTS card line. *Note:* Maximum/minimum integer values of 2147483648 and -2147483647 are understood as no bounds. The integer limits are characterized by four bytes. Maximum/minimum real number values of 1.79769e+308 and -1.79769e+308 are understood as no bounds. The real number limits are characterized by eight bytes.			
Card Type	GP_VAL		
Description	Global Parameters values		
Required	NO		
Format	GP_VAL GroupId ParamId (CURVE or FLOAT only if type is float/curve) Value		
Sample	GP_VAL 1 1 30.23		
Sample	GP_VAL 1 2 "Manning"		
Field	Variable	Value	Description
1	Group Id	+ integer	id of the Global Parameter Group that it belongs too
2	Param Id	+ integer	its id
3	Value	varies depending on type	value
Card Type	GP_DEP		
Description	Global Parameters dependencies		
Required	NO		
Format	GP_DEP GroupId ParamId Type Parent ParentActive Opts OptsValue		
Sample	GP_DEP 1 7 "PARENT_SELF" "Friction type" 0 "Manning" 0 "Chezy" 1		
Field	Variable	Value	Description
1	Group Id	+ integer	id of the Global Parameter Group that it belongs too
2	Param Id	+ integer	its id

3	Type	string valid values are: • PARENT_UNASSGINED • PARENT_NONE • PARENT_LOCAL • PARENT_GLOBAL • PARENT_SELF	• parent not assigned • no parent • parent is in the same group id • parent is from global group or GP • parent
4	Parent	+ integer	parent name
5	Parent Active	boolean	• 0 = inactive • 1 = active
6	Opts	string	name of the option
7	OptsValue	boolean	whether this option is turned on/off

Boundary Condition Definition Cards

Card Type	BCPGC		
Description	Defines whether or not to allow boundary condition / parameter group correlation.		
Required	NO		
Format	BCPGC val		
Sample	BCPGC 1		
Field	Variable	Value	Description
1	val	boolean	• 0 = Do not allow boundary condition / parameter group correlation. • 1 = Allow boundary condition / parameter group correlation.
Card Type	BEDISP		
Description	Defines inactive boundary condition display options and boundary condition label options.		
Required	NO		
Format	BEDISP entity_ID font_red font_green font_blue label_on label_vals_on inactive_size inactive_style inactive_red inactive_green inactive_blue inactive_on		
Sample	BEDISP 0 2 0 0 0 1 1 1 255 128 255 1		
Field	Variable	Value	Description
1	entity_ID	integer	• 0 = node • 1 = nodestring • 2 = element

2	font_red	integer	0 - 255, Red component of RGB triplet defining boundary condition font color.
3	font_green	integer	0 - 255, Green component of RGB triplet defining boundary condition font color.
4	font_blue	integer	0 - 255, Blue component of RGB triplet defining boundary condition font color.
5	label_on	boolean	• 0 = Do not display boundary condition labels. • 1 = Display boundary condition labels.
6	label_vals_on	boolean	• 0 = Do not display boundary condition values in boundary condition labels. • 1 = Display boundary condition values in boundary condition labels.
7	inactive_size	integer	• 1 - 63, Default size for inactive boundary condition node symbols. • 1 - 50, Default size for inactive boundary condition element or nodestring symbols.
8	inactive_style	integer	• Default style for inactive boundary condition symbols. • For nodes • 1 = Filled square • 2 = Hollow square • 3 = Filled circle • 4 = Hollow circle • 5 = Filled triangle • 6 = Hollow triangle • 7 = Filled diamond • 8 = Hollow diamond • 9 = Cross

			• 10 = X • 11 = Survey marker • For elements or nodestrings: • 0 = solid line • 1 = dashed line
9	inactive_red	integer	0 - 255, Red component of RGB triplet defining inactive boundary condition font color.
10	inactive_green	integer	0 - 255, Green component of RGB triplet defining inactive boundary condition font color.
11	inactive_blue	integer	0 - 255, Blue component of RGB triplet defining inactive boundary condition font color.
12	inactive_on	boolean	• 0 = Do not display inactive boundary conditions. • 1 = Display inactive boundary conditions.
Card Type	BEFONT	Single integer method.	
Description	Defines boundary condition label font attributes.		
Required	NO		
Format	BEFONT entity_id font_size		
Sample	BEFONT 1 1		
Field	Variable	Value	Description
1	entity_ID	integer	• 0 = node • 1 = nodestring • 2 = element
2	font_size	integer	• 1 = Large font size. • 2 = Small font size.
Note: The font information may be represented by a single integer or represented by multiple integers and a font name. *Note:* The BEDISP and BEFONT cards are written before the entity's boundary condition and values (if any) are written. The next instance of these cards will begin the next entity. The entity type ID equals zero for node, one for nodestring and two for element.			
Card Type	BEFONT	Multiple value method.	
Description	Defines boundary condition label font attributes.		
Required	NO		

Format	BEFONT entity_id height width escapement orientation weight italic underline strikeout charSet precision clipPrecision quality pitchAndFamily faceName		
Sample	BEFONT 1 1		
Field	Variable	Value	Description
1	entity_ID	integer	• 0 = node • 1 = nodestring • 2 = element
2	height	integer	Font height.
3	width	integer	Font width.
4	escapement	integer	Font escapement.
5	orientation	integer	Font orientation.
6	weight	integer	Font weight.
7	italic	integer	Font italic.
8	underline	integer	Font underline.
9	strikeout	integer	Font strikeout.
10	charSet	integer	Font character set.
11	precision	integer	Font precision.
12	clipPrecision	integer	Font clip precision.
13	quality	integer	Font quality.
14	pitchAndFamily	integer	Font pitch and family.
15	faceName	string	Font face name.
Note: The font information may be represented by a single integer or represented by multiple integers and a font name. *Note:* The BEDISP and BEFONT cards are written before the entity's boundary condition and values (if any) are written. The next instance of these cards will begin the next entity. The entity type ID equals zero for node, one for nodestring and two for element.			
Card Type	BC_DISP_OPTS		
Description	Boundary Condition display options (how the boundary conditions are going to be displayed)		
Required	NO		
Format	BC_DISP_OPTS GroupId Red Green Blue Display Pattern Width Style		
Sample	BC_DISP_OPTS 1 0 0 0 0 0 0 0 0		
Field	Variable	Value	Description
1	Group Id	integer	• 0 = node • 1 = nodestring • 2 = element
2	Red	integer (0-255)	red pixels

3	Green	integer (0-255)	green pixels
4	Blue	integer (0-255)	blue pixels
5	Display	boolean (0,1)	turned on = 1, turned off = 0
6	Pattern	integer	displayed pattern
7	Width	integer	width
8	Style	integer	style pattern

Card Type	BEG2DMBC
Description	Identifies the beginning of the boundary condition assignment section of the 2D Mesh File.
Required	Required if boundary conditions are assigned.

Card Type	END2DMBC
Description	Identifies the end of the boundary condition assignment section of the 2D Mesh File.
Required	Required if boundary conditions are assigned.

Boundary Condition Assignment Cards

The following cards are used to assign values to the boundary conditions defined in using **Boundary Condition Definition Cards** :

Card Type	BC		
Description	Defines input values required for a boundary condition.		
Required	NO		
Format	BC EntityId Name Id 0 LegalOnInterior ParamGroupCorrelation		
Sample	• BC 1 "Flow rate (cfs)" 1 0 1 "(none)" • BC 1 "Water temperature (F)" 2 0 "Hydrodynamic" • BC 1 "Flow rate(cfs)" 3 0 1 "(none)"		
Field	Variable	Value	Description
1	entity id	integer	Entity id that the bc belongs to (NODE = 0, NDSTR =1, ELEM = 2
2	name	string	Name of value to be specified.
3	id	integer	The boundary conditions id.
4	filler	0	Always 0
5	legalOnInterior	boolean	• 0 = boundary condition is not legal on the interior of the mesh. • 1 = boundary condition is legal on the interior of the mesh.
6	paramGroupCorrelation	string	Name of the parameter group (defined using the GP card)

			with which the boundary condition is correlated. If the boundary condition is not correlated with a parameter group, "none" should be specified.
Card Type	BC_DEF		
Description	Boundary Condition Parameter Defaults		
Required	NO		
Format	The format will depend up the type (field 4). Fields 5, 6, and 7 will be impacted by the choice of field 4. Note that the min and max information is only applicable to integer and double types. bool, integer, double, text, or options : BC_DEF GroupId ParamId Name Type Default Min Max curve: BC_DEF GroupId ParamId Name Type x_axis_title y_axis_title Float/Curve: BC_DEF GroupId ParamId Name Type Float_Default Float_Min Float_Max DefaultFloatOrCurve x_axis_title y_axis_title		
Sample	Values depend upon type (see format). bool, integer, double, text or options : BC_DEF 1 1 "manning n" 1 0 0 10 curve: BC_DEF 1 1 "manning n" 5 "x-axis" "y-axis" Float/Curve: BC_DEF 1 1 "manning n" 6 0.2 0.0 1.0 CURVE "x-axis" "y-axis"		
Field	Variable	Value	Description
1	group Id	+ integer	id of the boundary condition group that it belongs too
2	param Id	string	its id
3	name	string	its name
4	type	int	0-Bool, 1-Integer, 2-Double, 3-Text, 4-Options, 5-Curve, 6-Float/Curve
5	default or x_axis_title (curve)	type specific string	default value x-axis title when bringing up curve editor
6	min or y_axis_title (curve)	type specific (min) string	minimum value y-axis title when bringing up curve editor
7	max	type specific (max)	max value
8 (only float/curve)	DefaultFloatOrCurve	string either • "FLOAT" • "CURVE"	The default version float or integer that is seen
9 (only float/curve	x_axis_title	string	x-axis title when bringing up curve editor
10 (only float/curve)	y_axis_title	string	y-axis title when bringing up

			curve editor
Card Type	BC_OPTS		
Description	Boundary Condition options		
Required	NO		
Format	BC_OPTS GroupId, ParamId, Values		
Sample	BC_OPTS 1 5 "a" "b" "c"		
Field	Variable	Value	Description
1	group Id	+ integer	id of the boundary condition group that it belongs too
2	param Id	string	its id
3	values	string	
Card Type	BC_VAL		
Description	Boundary Condition values		
Required	NO		
Format	BC_VAL N, E, S (Node, Elem or nodeString) Node or Elem Id, GroupId, ParamId, (CURVE or FLOAT only if type is float/curve) Value		
Sample	BC_VAL S 1 1 2 2		
Sample	BC_VAL S 1 1 2 CURVE 2		
Sample	BC_VAL S 1 1 2 FLOAT 7.675		
Field	Variable	Value	Description
1	N, E or S	+ integer	Node, Elem or Nodestring
2	Node, Elem, or Nodestring Id	+ integer	id of the node/element
3	group Id	+ integer	id of the group it belongs too
4	param Id	+ integer	id of the parameter it belongs too
5	value	varies depending on type	value
Note: These cards are only written if the data that exists is different than the default data.			
Card Type	BC_DEP		
Description	Boundary Condition dependencies		
Required	NO		
Format	BC_DEP GroupId ParamId Type Parent ParentActive Opts OptsValue		
Sample	BC_DEP 1 7 "PARENT_SELF" "Friction type" 0 "Manning" 0 "Chezy" 1		
Field	Variable	Value	Description
1	Group Id	+ integer	id of the Global Parameter Group that it belongs too
2	Param Id	+ integer	its id

3	Type	string valid values are: • PARENT_UNASSGINED • PARENT_NONE • PARENT_LOCAL • PARENT_GLOBAL • PARENT_SELF	• parent not assigned • no parent • parent is in the same group id • parent is from global group or GP • parent
4	Parent	+ integer	parent name
5	Parent Active	boolean	• 0 = inactive • 1 = active
6	Opts	string	name of the option
7	OptsValue	boolean	whether this option is turned on/off

Material Properties Cards

Card Type	MAT_MULTI		
Description	Is Material Assignment Multiple		
Required	NO		
Format	MAT_MULTI Assigned		
Sample	MAT_MULTI 1		
Sample	MAT_MULTI 0		
Field	Variable	Value	Description
1	assigned	boolean	0- Single, 1 - Multiple
Card Type	MAT_PARAMS		
Description	Material Parameters Assigned		
Required	NOT NEEDED IF MAT_MULTI 0		
Format	MAT_PARAMS MaterialId, GroupId (1 or More)		
Sample	MAT_PARAMS 1 2 3 5		
Sample	MAT_PARAMS 1 1		
Field	Variable	Value	Description
1	Material Id	+ integer	id of the material
2	Group id	+ integer	id of the assigned group

Material Properties Assignment Cards

Card Type	MAT
Description	Defines the assigned material properties.
Required	NO

Format	MAT ID "name"		
Sample	MAT 1 "Clay"		
Field	Variable	Value	Description
1	mat_ID	+ integer	Material id (sequentially numbered, starting at 1). The disabled material is always id 0 and does not need to be specified in the 2dm file.
2	name	string	The name of the material. Should be unique.
Card Type	MAT_DEF		
Description	Material Parameters defaults		
Required	NO		
Format	The format will depend up the type (field 4). Fields 5, 6, and 7 will be impacted by the choice of field 4. Note that the min and max information is only applicable to integer and double types. bool, integer, double, text, or options : MAT_DEF GroupId ParamId Name Type Default Min Max curve: MAT_DEF GroupId ParamId Name Type x_axis_title y_axis_title Float/Curve: MAT_DEF GroupId ParamId Name Type Float_Default Float_Min Float_Max DefaultFloatOrCurve x_axis_title y_axis_title		
Sample	Values depend upon type (see format). bool, integer, double, text or options : MAT_DEF 1 1 "manning n" 1 0 0 10 curve: MAT_DEF 1 1 "manning n" 5 "x-axis" "y-axis" Float/Curve: MAT_DEF 1 1 "manning n" 6 0.2 0.0 1.0 CURVE "x-axis" "y-axis"		
Field	Variable	Value	Description
1	group id	+ integer	id of the Material Group that it belongs too.
2	param id	string	its id
3	name	string	its name
4	type	int	0-Bool, 1-Integer, 2-Double, 3-Text, 4-Options, 5-Curve, 6-Float/Curve
5	default or x-axis_title (curve)	type specific string	default value x-axis title when bringing up curve editor
6	min or y-axis_title (curve)	type specific (min) string	minimum value y-axis title when bringing up curve editor
7	max	type specific (max)	max value
8 (only float/curve)	DefaultFloatOrCurve	string either	The default version float or

		• "FLOAT" • "CURVE"	integer that is seen
9 (only float/curve)	x_axis_title	string	x-axis title when bringing up curve editor
10 (only float/curve)	y_axis_title	string	y-axis title when bringing up curve editor
Card Type	MAT_OPTS		
Description	Material options		
Required	NO		
Format	MAT_OPTS GroupId, ParamId, Values		
Sample	MAT_OPTS 1 5 "a" "b" "c"		
Field	Variable	Value	Description
1	group id	+ integer	id of the Boundary Condition Group that it belongs too
2	param id	+ integer	its id
3	values	string	
Card Type	MAT_VALS		
Description	Material values		
Required	NO		
Format	MAT_VAL MaterialId, GroupId, ParamId, (CURVE or FLOAT only if type is float/curve) Value		
Sample	MAT_VAL S 1 1 2 2		
Sample	MAT_VAL 2 1 8 VALUE 8.8888		
Sample	MAT_VAL 2 1 8 CURVE 1		
Field	Variable	Value	Description
1	material id	+ integer	id of the material
2	group id	+ integer	id of the group it belongs too
3	param id	string	id of the parameter it belongs too
4	VALUE or CURVE	string	Only if type is float/curve
5	value	varies depending on type	value
Card Type	MAT_DEP		
Description	Material dependencies		
Required	NO		
Format	MAT_DEP GroupId ParamId Type Parent ParentActive Opts OptsValue		
Sample	MAT_DEP 1 7 "PARENT_SELF" "Friction type" 0 "Manning" 0 "Chezy" 1		
Field	Variable	Value	Description

1	GroupId	+ integer	id of the Global Parameter Group that it belongs too
2	ParamId	+ integer	its id
3	Type	string valid values are: • PARENT_UNASSGINED • PARENT_NONE • PARENT_LOCAL • PARENT_GLOBAL • PARENT_SELF	• parent not assigned • no parent • parent is in the same group id • parent is from global group or GP • parent
4	Parent	+ integer	parent name
5	ParentActive	boolean	• 0 = inactive • 1 = active
6	Opts	string	name of the option
7	OptsValue	boolean	whether this option is turned on/off

Time Series Data Cards

Card Type	BEGCURVE Version: version		
Description	Identifies the beginning of the time series data section of the 2D Mesh File.		
Required	NO		
Format	BEGCURVE		
Sample	BCE 2 2 300		
Field	Variable	Value	Description
1	version	+ integer	Used to version file format for the curve portion of the file.
Note: Version 1 writes the time series data using the XY Series File, XYS Format within the 2D Mesh File. See XY Series Files (*.xys) for a description of the cards used to define the time series data.			
Card Type	ENDCURVE		
Description	Identifies the end of the time series data section of the 2D Mesh File.		
Required	NO		

Obsolete Cards

The following cards are no longer used in SMS:

PG, PD, PO, GG, GP, BD, BV, MD, MV, BCE, BCN, BCS, TIME, BEDISP

Changes from 10.1 to 11.0

New Cards:

- NUM_MATERIALS_PER_ELEM

- GP replaces PG
- GP_DEF replaces PD
- GP_VAL
- GP_DEP
- BC replaces BD
- BC_DEF replaces BV card
- BC_OPTS replaces PO card
- BC_VAL replaces BC card
- BC_DEP
- BC_DISP_OPTS replaces bedisp
- MAT_MULTI
- MAT_PARAMS
- MAT_DEF replaces MV card
- MAT_OPTS
- MAT_VAL
- MAT_DEP
- DISP_OPTS entity
- DISP_OPTS inactive
- DISP_OPTS multiple

Sample File 1

```
BEGCURVE Version: 1
XYS 1 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
```

```
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 3 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 5 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
```

```
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 7 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
```

```
XYS 9 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 11 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
```

```
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 13 8 "new_series"
0.0 237.35
24.0 137.9
48.0 1347.4
72.0 351.05
96.0 1465.25
120.0 1247.1
144.0 847.35
168.0 731.75
XYS 14 8 "Curve"
0.0 1600.7
24.0 700.85
48.0 1353.7
72.0 712.25
96.0 866.6
120.0 1626.35
144.0 567.6
168.0 980.55
XYS 15 8 "Curve"
0.0 1240.7
24.0 772.25
48.0 741.3
72.0 908.45
96.0 599.45
120.0 522.8
144.0 946.8
168.0 170.4
XYS 16 8 "Curve"
0.0 1252.55
24.0 1467.95
48.0 876.25
72.0 250.2
96.0 479.95
120.0 981.65
144.0 1432.4
168.0 1382.8
XYS 17 8 "Curve"
0.0 1507.65
```

```
24.0 202.6
48.0 905.3
72.0 1051.45
96.0 434.95
120.0 267.5
144.0 547.95
168.0 349.2
ENDCURVE
```

Sample File 2

Sample files are available in the SMS tutorials in the Generic Model tutorial under the models section.

```
MESH2D
E3T      1      4      1      3      2
E3T      2      2      5      6      2
E4Q      3      7      8      5      2      2
.
.
.
E4Q   1543    205   1226   1225   1221      2
E4Q   1544    191   1222   1226    189      1
E3T   1545    205    189   1226      2
ND      1 -7.62907961e+001 4.00243909e+001 8.41808447e+001
ND      2 -7.62907174e+001 4.00219296e+001 8.36614138e+001
ND      3 -7.62907700e+001 4.00238340e+001 7.32122342e+001
.
.
.
ND   1222 -7.62811008e+001 4.00272795e+001 7.28898113e+001
ND   1225 -7.62814608e+001 4.00273631e+001 7.29479847e+001
ND   1226 -7.62812859e+001 4.00271526e+001 7.41231480e+001
NS       1      3     10     15      6     -2
NS     126    127    128    129    173   -194
NS    1006    988    987    989    968    948    931    930    929    906
NS    -904
NS     720    701    699    686    680    664    649    648    647    640
NS    -621
BEGPARAMDEF
GM  "Gen2DM"
SI  0
DY  1
TU  "minutes"
TD  20  1000
KEY  "sms-gen2dm"
PG  "Hydrodynamic" 1
PD  "Time interval" 1 20 0 2147483647
PD  "Velocity max (ft/sec)" 2 75 0 100
```

```
PD  "H min" 2 0.25 0 1.79769e+308
PD  "A min" 2 1 1e-015 1.79769e+308
PD  "Check for dry elements" 0 1
PD  "Element style" 3 "quadratic"
PD  "Critical scour velocity" 4 "2.0 ft/sec"
PO  "0.8 ft/sec" "2.0 ft/sec" "2.6 ft/sec"
PG  "Sediment transport" 0
PD  "Time interval" 1 10 0 2147483647
PD  "Source X postion" 2 0 -1.79769e+308 1.79769e+308
PD  "Source Y position" 2 0 -1.79769e+308 1.79769e+308
PD  "Source elevation" 2 0 -1.79769e+308 1.79769e+308
PD  "Parcel mass (slug)" 2 0.5 0.0001 1.79769e+308
PD  "Particle mass (slug)" 2 0.003 0.0001 1.79769e+308
PD  "Particle size (in)" 2 0.05 0 1.79769e+308
PD  "Deviation" 2 0 -1.79769e+308 1.79769e+308
PD  "Average density (slug/ft^3)" 2 3 1.5 6
NUME  3
BCPGC  1
BEDISP  0 2 0 0 0 1 1 1 255 128 255 1
BEFONT  0 1
BD  0 "Water sink/source" 2 3 "Flow rate (cfs)" "Water temperature (F)" "Flow
rate(cfs)" 1 "(none)"
BV  "Flow rate (cfs)" 0 -1.79769e+308 1.79769e+308
BV  "Water temperature (F)" 65 32.5 100
BV  "Flow rate(cfs)" 0 0 1.79769e+308
BCDISP  0 2 10 1 0 255 255 1
BD  0 "Ceiling (pressure flow)" 1 1 "Ceiling (ft above sea level)" 0 "(none)"
BV  "Ceiling (ft above sea level)" 0 -1.79769e+308 1.79769e+308
BCDISP  0 1 3 1 128 128 255 1
BD  0 "Water surface observation gauge" 3 0 1 "(none)"
BCDISP  0 3 3 1 255 128 128 1
BEDISP  1 0 0 255 1 1 1 0 255 128 0 1
BEFONT  1 1
BD  1 "Water surface" 1 3 "Elevation" "Essential/Natural factor" "Vary along nodestring
factor" 0 "(none)"
BV  "Elevation" 0 -1.79769e+308 1.79769e+308
BV  "Essential/Natural factor" 0 0 1
BV  "Vary along nodestring factor" 1 0 10
BCDISP  1 1 5 0 255 0 0 1
BD  1 "Flow" 2 1 "Flow rate (cfs)" 0 "(none)"
BV  "Flow rate (cfs)" 0 0 1.79769e+308
BCDISP  1 2 5 0 128 255 0 1
BD  1 "Supercritical" 3 0 0 "(none)"
BCDISP  1 3 1 0 0 0 0 1
BD  1 "1D weir segment" 4 4 "Discharge coefficient" "Weir width (ft)" "Crest level (m
above sea level)" "Equation (0 = water level / 1 = energy head)" 1 "(none)"
BV  "Discharge coefficient" 1 0 1.79769e+308
BV  "Weir width (ft)" 1 0 1.79769e+308
BV  "Crest level (m above sea level)" 0 -1.79769e+308 1.79769e+308
```

```
BV  "Equation (0 = water level / 1 = energy head)" 0 0 1
BCDISP  1 4 1 0 0 0 0 0
BD  1 "Sediment trap" 5 0 1 "Sediment transport"
BCDISP  1 5 1 0 0 0 0 1
BEDISP  2 2 0 0 0 1 1 0 0 0 0 1
BEFONT  2 1
BD  2 "2D weir" 1 3 "Discharge coefficient" "Crest level (ft above sea level)"
"Equation (0 = water level / 1 = energy head)" 1 "(none)"
BV  "Discharge coefficient" 1 0 1.79769e+308
BV  "Crest level (ft above sea level)" 0 -1.79769e+308 1.79769e+308
BV  "Equation (0 = water level / 1 = energy head)" 0 0 1
BCDISP  2 1 1 0 0 0 0 1
MD  2 "Manning" "Kinematic eddy viscosity"
MV  "Manning" 0.035 0.01 0.18
MV  "Kinematic eddy viscosity" 0 -1.79769e+308 1.79769e+308
ENDPARAMDEF
BEG2DMBC
MAT  1 0.03 20
MAT  2 0.045 20
GG "Hydrodynamic"
GP "Time interval" 20
GP "Velocity max (ft/sec)" 75
GP "H min" 0.25
GP "A min" 1
GP "Check for dry elements" 1
GP "Element style" "quadratic"
GP "Critical scour velocity" "2.0 ft/sec"
GG "Sediment transport"
GP "Time interval" 10
GP "Source X postion" 0
GP "Source Y position" 0
GP "Source elevation" 0
GP "Parcel mass (slug)" 0.5
GP "Particle mass (slug)" 0.003
GP "Particle size (in)" 0.05
GP "Deviation" 0
GP "Average density (slug/ft^3)" 3
BCN  772 3
BCN  774 3
BCN  776 3
.
.
.
BCS  4 5
BCS  1 1 80 0 1
BCE  1293 1 1 0 0
TIME 20
BCS  2 2 380
TIME 40
```

```
BCS  2 2 400
TIME 60
BCS  2 2 380
TIME 80
BCS  2 2 300
TIME 100
BCS  2 2 300
TIME 120
BCS  2 2 300
TIME 140
BCS  2 2 300
TIME 160
BCS  2 2 300
TIME 180
BCS  2 2 300
TIME 200
BCS  2 2 300
TIME 220
BCS  2 2 300
TIME 240
BCS  2 2 300
TIME 260
BCS  2 2 300
TIME 280
BCS  2 2 300
TIME 300
BCS  2 2 300
TIME 320
BCS  2 2 300
TIME 340
BCS  2 2 300
TIME 360
BCS  2 2 300
TIME 380
BCS  2 2 300
TIME 400
BCS  2 2 300
TIME 420
BCS  2 2 300
TIME 440
BCS  2 2 300
TIME 460
BCS  2 2 300
TIME 480
BCS  2 2 300
TIME 500
BCS  2 2 300
TIME 520
BCS  2 2 300
```

```
TIME 540
BCS  2 2 300
TIME 560
BCS  2 2 300
TIME 580
BCS  2 2 300
TIME 600
BCS  2 2 300
TIME 620
BCS  2 2 300
TIME 640
BCS  2 2 300
TIME 660
BCS  2 2 300
TIME 680
BCS  2 2 300
TIME 700
BCS  2 2 300
TIME 720
BCS  2 2 300
TIME 740
BCS  2 2 300
TIME 760
BCS  2 2 300
TIME 780
BCS  2 2 300
TIME 800
BCS  2 2 300
TIME 820
BCS  2 2 300
TIME 840
BCS  2 2 300
TIME 860
BCS  2 2 300
TIME 880
BCS  2 2 300
TIME 900
BCS  2 2 300
TIME 920
BCS  2 2 300
TIME 940
BCS  2 2 300
TIME 960
BCS  2 2 300
TIME 980
BCS  2 2 300
TIME 1000
BCS  2 2 300
```

```
END2DMBC
```

Related Topics

- File Formats
- Generic Model Files
- HYDRO AS-2D
- SRH-2D

2D Scatter Point Files

2D Scatter Point Files (*.xy)

Two-dimensional scatter point sets are stored in 2D scatter point files. The file includes the scatter point locations and requires that functional information be defined in a separate dataset file. However, multiple scatter point sets can be stored in a single file. An XY coordinate pair defines each point in a scatter point set. The format allows time variant datasets to be associated with scattered data points as well as to organize datasets by allowing the user to assign an ID to the scattered dataset. Scatter files are opened through *File* | **Open** and are saved from *File* | **Save Scatter** from the Scatter module. When scatter point files are saved, a super file is saved. The super file saves and references the scatter and dataset files.

File Format

SCAT2D	/* File type identifier */
BEGSET	/* Beginning of cards for scatter point set */
NAME "name"	/* Name of scatter point set */
ID id	/* ID of scatter point set */
DELEV elev1	/* Default elevation */
IXY np	/* Number of points in set, begin point listing */
id1 x1 y1	/* Point id and coordinatess, one per line */
id2 x2 y2	
.	
.	
idnpxnpynp	
ENDSET	/* End of cards for scatter point set */
	/* Repeat point set cards as many times as necessary */

Sample File

```
SCAT2D
BEGSET
NAME "lakes"
ID 8493
DELEV 0.00000000000e+00
```

```
IXY 25
1 1.47000000000e+02 3.90000000000e+02
2 8.82000000000e+02 9.49000000000e+02
.
.
24 1.73000000000e+02 7.01000000000e+02
25 5.39000000000e+02 8.98000000000e+02
ENDSET
```

Cards

Card Type	**BEGSET**
Description	Identifies the beginning of a scatter point set. No fields.
Required	NO

Card Type	**NAME**		
Description	Defines the name for the following scatter point set.		
Required	NO		
Format	NAME "name"		
Sample	NAME "st mary"		
Field	*Variable*	*Value*	*Description*
1	name	str	The name for the following scatter points. Remains as default until new NAME card is encountered.
Card Type	**ID**		
Description	Defines the ID for the scatter point set.		
Required	YES		
Format	ID id		
Sample	ID 43098		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID for the following scatter point set.
Card Type	**IXY**		
Description	Defines a scatter point set.		
Required	YES		
Format			
IXY np			
id1 x1 y1			
id2 x2 y2			

.			
.			
idnp xnp ynp			
Sample			
IXY 4			
1 12.3 34.5			
2 52.2 23.5			
3 63.2 27.4			
4 91.1 29.3			
Field	*Variable*	*Value*	*Description*
1	np	+	The number of scatter points in the scatter point set.
2	id	+	The ids of the points.
3-4	x,y	±	The coordinates of the points.
Repeat fields 2 - 4 np times			
Card Type		**ENDSET**	
Description		Identifies the end of a scatter point set. No fields.	
Required		NO	

Related Topics

File Formats

2D Grid Files

Two-dimensional grids are stored in 2D grid files. The grids can be either cell-centered or mesh-centered. If the grid is mesh-centered, a set of material IDs may be included in the file. The 2D grid file format is shown in Figure 1, and a sample file in Figure 2.

GRID2D	/* File type */
TYPE i	/* Type of grid. Mesh or Cell centered. */
IJ ±idir ±jdir	/* Card for defining rows, columns. */
DIM nx ny	/* # of cell boundaries in each direction. */
x1	/* X coord. of cell boundaries. */
x2	
.	
.	
x_{nx}	
y1	/* Y coord. of cell boundaries. */

y2	
.	
.	
y_{ny}	
DELEV el	/* Default elevation for grid. */

Figure 1. 2D Grid File Format.

GRID2D
ID 5758
TYPE 1
DELEV 0.000000000000000e+00
IJ -y +x
DIM 4 4
0.000000000000000e+00
3.333333333333334e+01
6.666666666666667e+01
1.000000000000000e+02
0.000000000000000e+00
3.333333333333334e+01
6.666666666666667e+01
1.000000000000000e+02

Figure 2. Sample 2D Grid File.

2D Grid Files Card Types

The card types used in the 2D grid file format are as follows:

Card Type	GRID2D
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	TYPE		
Description	Defines the type of grid as either cell- or mesh-centered.		
Required	YES		
Format	TYPE i		
Sample	TYPE 0		
Field	Variable	Value	Description
1	i	0,1	The type code: • i = 0 for mesh-centered

			• i = 1 for cell-centered
Card Type	IJ		
Description	Defines the orientation of the i,j indices.		
Required	YES		
Format	IJ ±idir ±jdir		
Sample	IJ +x -y		
Field	Variable	Value	Description
1	±idir	±x,±y	The direction corresponding to an increasing i index.
2	±jdir	±x,±y	The direction corresponding to an increasing j index.
Card Type	DIM		
Description	Defines the dimensions of the grid.		
Required	YES		
Format	DIM nx ny x1 x2 . . xnx y1 y2 . . yny		
Sample	DIM 4 6 0.0 1.0 2.0 4.0 10.0 12.0 14.0 16.0 18.0 20.0		
Field	Variable	Value	Description
1	nx	+	The number of cell boundaries in the x direction.

2	ny	+	The number of cell boundaries in the y direction.
3 to (nx+2)	x1-xnx	±	The coordinates of the x boundaries.
(nx+3) to (nx+ny+2)	y1-yny	±	The coordinates of the y boundaries.
Card Type	DELEV		
Description	Defines a default elevation for the grid.		
Required	NO		
Format	DELEV el		
Sample	DELEV 100.0		
Field	Variable	Value	Description
1	el	±	The default elevation.

Back to XMS

ARC/INFO® ASCII Grid Files *.arc

SMS can import ARC/INFO® ASCII grid. Since it is a simple file format, other digital elevation data can be formatted in the same way and then imported into SMS using the **Open** command in the *File* menu. This will bring up the *Importing ArcInfo Grid* dialog.

The CASC2D model (now GSSHA) may also import and use ARC/INFO® grid files when defining map parameters. The file format is shown below in Figure 1 and an example file in Figure 2.

ncols ncol	/* Number of columns in the grid */
nrows nrow	/* Number of rows in the grid */
xllcorner x	/* Lower left x coordinate of grid */
yllcorner y	/* Lower left x coordinate of grid */
cellsize size	/* Grid cell size */
NODATRA_value NODATA	/* value of an empty grid cell */
z_{11} z_{12} z_{13} ... z_{1ncols}	/* values of row 1 */
z_{21} z_{22} z_{23} ... z_{2ncols}	/* values of row 2 */
.	
.	
.	
z_{nrows1} z_{nrows2} z_{nrows3} ... $z_{nrowsncols}$	/* values of last row*/

Figure 1. ARC/INFO® ASCII Grid File Format.

ncols 128
nrows 136
xllcorner 422415

yllcorner 4515405
cellsize 30
NODATA_value -9999
1287 1286 1286 1288 ...
1288 1288 -9999 1289 ...
.
.
1282 -9999 1283 1284 ...

Figure 2. Sample ARC/INFO® ASCII Grid File.

The card types used in the ARC/INFO® grid file format are self explanatory.

ASCII Dataset Files *.dat

Datasets can be stored in either ASCII or binary files . Multiple datasets can be stored in a single file and both scalar and vector datasets can be saved to the same file. For scalar dataset files, one value is listed per vertex, cell, node, or scatter point. For vector dataset files, one set of XY vector components is listed per vertex, cell, node, or scatter point. If necessary, a set of status flags can be included in the file. If the status flag is false (0), the corresponding item (node, cell, etc.) is inactive. If status flags are not included in the file, it is assumed that all items are active. Dataset files are opened through *File* | **Open** and are saved when other files are saved such as 2D Scatter Point Files or through the Export Dataset Dialog .

File Format

```
DATASET                  /* File type identifier */
OBJTYPE type             /* Type of object data set is associated with */
BEGSCL                   /* Beginning of scalar data set */
OBJID id                 /* Object id */
ND numdata               /* Number of data values */
NC numcells              /* Number of cells or elements */
NAME "name"              /* Data set name */
RT_JULIAN                /* The reference time as a Julian number. */
TIMEUNITS                /* The time units. */
TS istat time            /* Time step of the following data. */
stat1                    /* Status or activity flags */
stat2
.
.
statnumcells
val1                     /* Scalar data values */
val2
.
.
valnumdata
/* Repeat TS card for each time step */
ENDDS                    /* End of data set */
BEGVEC                   /* Beginning of vector dataset */
VECTYPE type             /* Vector at node/gridnode or element/cell */
```

```
OBJID id              /* Object id */
ND numdata            /* Number of data values */
NC numcells           /* Number of cells or elements */
NAME "name"           /* Data set name */
TS istat time         /* Time step of the following data. */
stat1                 /* Status or activity flags */
stat2
.
.
statnumcells
vx1 vy1
vx2 vy2
.
.
vnumdata vnumdata vnumdata
/* Repeat TS card for each time step */
ENDDS                 /* End of data set */
/* Repeat BEGSCL and BEGVEC sequences for each data set */
```

Sample File

Note: This sample file is using an activity array, so there are 16 values per TS. The first 8 values are activity flags for each of the 8 nodes. Values 9-16 are the scalar values for the 8 nodes.

```
DATASET
OBJTYPE "grid2d"
BEGSCL
ND 8
NC 8
NAME "sediment transport"
RT_JULIAN 2453867.068720
TIMEUNITS seconds
TS 1 1.00000000e+00
0
0
0
1
1
1
1
0
0.00000000e+00
0.00000000e+00
0.00000000e+00
3.24000000e+00
4.39000000e+00
2.96000000e+00
7.48000000e+00
0.00000000e+00
```

```
ENDDS
BEGVEC
VECTYPE 0
ND 8
NC 8
NAME "velocity"
TS 1 5.00000000e+00
0
0
0
1
1
1
1
0
1.60000000e+01 1.60000000e+01
6.40000000e+01 6.40000000e+01
1.44000000e+02 1.44000000e+02
1.96000000e+02 1.96000000e+02
2.25000000e+02 2.25000000e+02
9.21600000e+03 9.21600000e+03
9.60400000e+03 9.60400000e+03
9.80100000e+03 9.80100000e+03
ENDDS
```

Cards

Card Type	**DATASET**		
Description	File type identifier. Must be on first line of file. No fields.		
Required	YES		
Card Type	**OBJTYPE**		
Description	Identifies the type of objects that the datasets in the file are associated with.		
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.		
Format	OBJTYPE type		
Sample	OBJTYPE tin		
Field	*Variable*	*Value*	*Description*
1	type	"mesh2d"	2D mesh
		"scat2d"	2D scatterpoints
		"cgrid2d"	2D cartesian grid
		"specgrid2d"	Spectral energy grid
Card Type	**OBJID**		

Description	Identifies the object that the data sets in the file are associated with.		
Required	NO. Card is only used if the OBJTYPE is scat2d.		
Format	OBJID id		
Sample	OBJID 1254		
Field	*Variable*	*Value*	*Description*
1	id	Integer > 0	The ID of the object.

Card Type	**RT_JULIAN**			
Description	The reference time as a Julian number.			
Required	NO			
Field	*Variable*	*Size*	*Value*	*Description*
1	reference time	8 byte float	+/-	Continuous count of days and fractions of days since noon Universal Time on January 1, 4713 BCE (on the Julian calendar).

Card Type	**TIMEUNITS**			
Description	The time units.			
Required	NO, but recommended			
Field	*Variable*	*Size*	*Value*	*Description*
1	time units	4 byte float		
			0	Hours
			1	Minutes
			2	Seconds

Card Type	**BEGSCL**
Description	Scalar data set file identifier. Marks beginning of scalar data set. No fields.
Required	YES
Card Type	**BEGVEC**
Description	Vector data set file identifier. Marks beginning of vector data set. No fields.
Required	YES

Card Type	**VECTYPE**			
Card ID	150			
Description	Identifies the type of vector data that will be read and where to apply it.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The data sets would then be assigned to the objects corresponding to the active module.			
Field	*Variable*	*Size*	*Value*	*Description*

1	type	4 byte int	0 1	The vectors are applied to the nodes/grid nodes. The vectors are applied to the elements/cells.

Card Type	**ND**		
Description	The number of data values that will be listed per time step. This number should correspond to the total number of vertices, nodes, cells centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.		
Required	YES.		
Format	ND numdata		
Sample	ND 10098		
Field	*Variable*	*Value*	*Description*
1	numdata	+	The number of items. At each time step, numdata values are printed.

Card Type	**NC**		
Description	This number should correspond to the maximum element id (meshes) or the number of cells (grids).		
Required	YES.		
Format	NC numcells		
Sample	NC 3982		
Field	*Variable*	*Value*	*Description*
1	numcells	+	The number of elements or cells

Card Type	**NAME**		
Description	The name of the data set.		
Required	YES.		
Format	NAME "name"		
Sample	NAME "Total head"		
Field	*Variable*	*Value*	*Description*
1	"name"	str	The name of the dataset in double quotes.

Card Type	**TS**	
Description	Marks the beginning of a new time step, indicates if stat flags are given, and defines the time step value, status flags, and scalar data values for each item.	
Required	YES.	
Format	TS istat time	

	stat1 stat2 . . stat numcells val1 val2 . . valnumdata			
Field	*Variable*	*Value*	*Description*	
1	active array flag	0 1	Activity or status values not specified Activity or status values specified - A value of 0 (inactive) or 1 (active) must be specified for each node following the TS card, before the scalar or vector values are specified.	
2	Time step	double	Time step value for data	

Related Topics

- File Formats
- Binary Dataset Files (*.dat)
- Export Dataset Dialog

Binary Dataset Files *.dat

Datasets can be stored to either ASCII or binary files. Compared to ASCII files, binary files require less memory and can be imported to SMS more quickly. The disadvantages of binary files are that they are not as portable and they cannot be viewed with a text editor. The binary format is patterned after the ASCII format in that the data are grouped into "cards". However, the cards are identified by a number rather than a card title.

Dataset files are opened through *File* | **Open** and are saved when other files are saved such as 2D Scatter Point Files .

File Format

Card	**Item**	**Size**	**Description**
	version	4 byte integer	The SMS binary dataset file format version. value = 3000
100	objecttype	4 byte integer	Indentifies the type of objects that the datasets in the file are associated with. Options are as follows:

			1 TINs 3 2D meshes 5 2D scatter points	
110	SFLT	4 byte integer	The number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).	
120	SFLG	4 byte integer	The number of bytes that will be used in the remainder of the file for status flags.	
110	SFLT	4 byte integer	The number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).	
130 or 140	BEGSCL or BEGVEC		The number of bytes that will be used in the remainder of the file for status flags.	
150	VECTYPE	4 byte integer	(0 or 1) In the case of vector dataset files, indicates whether the vectors will be applied at the nodes/gridnodes or the elements/cells.	
160	OBJID	4 byte integer	The id of the associated object. Value is ignored for grids and meshes.	
170	NUMDATA	4 byte integer	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid) or scatter points.	
180	NUMCELLS	4 byte integer	This number should correspond to the number of elements (meshes) or the number of cells (mesh-centered grids). Value is ignored for other object types.	
190	NAME	40 bytes	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.	
200	TS		Marks the beginning of a time step.	
	ISTAT	SFLG integer	(0 or 1) Indicates whether or not status flags will be included in the file.	
	TIME	SFLT real	Time corresponding to the time step.	
	statflag1	SFLG integer	Status flag (0 or 1) for node 1	
	statflag2	SFLG integer	Status flag (0 or 1) for node 2	
	...			
Repeat card 200 for each time step in the dataset.				
210	ENDDS		Signal the end of a set of cards defining a dataset.	
240	RT_JULIAN	8 byte float	The reference time as a Julian number.	
250	TIMEUNITS	4 byte integer	The time units as follows: 0 - hours 1 - minutes	

			2 - seconds 4 - days	

Cards

Card Type	**VERSION**	
Card ID	3000	
Description	File type identifier. No fields.	
Required	YES	

Card Type	**OBJTYPE**			
Card ID	100			
Description	Identifies the type of objects that the datasets in the file are associated with.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.			
Format	OBJTYPE type			
Sample	OBJTYPE tin			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	tin mesh2d scat2d	Tins 2D mesh 2D scatterpoints
Card Type	**SFLT**			
Card ID	110			
Description	Identifies the number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizefloat	4 byte int	4 8 or 16	Number of bytes
Card Type	**SFLG**			
Card ID	120			
Description	Identifies the number of bytes that will be used in the remainder of the file for status flags (1, 2, or 4).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizeflag	4 byte int	Number of bytes	

		1 2 or 4		

Card Type	**BEGSCL**	
Card ID	130	
Description	Marks the beginning of a set of cards defining a scalar dataset.	
Required	YES	
Card Type	**BEGVEC**	
Card ID	140	
Description	Marks the beginning of a set of cards defining a vector dataset.	
Required	YES	

Card Type	**VECTYPE**			
Card ID	150			
Description	Identifies the type of vector data that will be read and where to apply it.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	0 1	The vectors are applied to the nodes/grid nodes. The vectors are applied to the elements/cells.
Card Type	**OBJID**			
Card ID	160			
Description	The id of the associated object.			
Required	This card is required in the case of TINs, 2D scatter points, and 3D scatter points. With each of these objects, multiple objects may be defined at once. Hence the id is necessary to relate the dataset to the proper object.			
Field	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	+	The id of the object.
Card Type	**NUMDATA**			
Card ID	170			
Description	The id of the associated object.			
Required	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered			

	grid), maximum node id (meshes) or scatter points.			
Field	*Variable*	*Size*	*Value*	*Description*
1	numdata	4 byte int	+	The number of items. At each time step, numdata are listed.
Card Type	**NUMCELLS**			
Card ID	180			
Description	This number should correspond to the element id (meshes) or the number of cells (grids).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	numcells	4 byte int	+	The number of elements or cells.
Card Type	**NAME**			
Card ID	190			
Description	The name of the dataset.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	name	40 bytes	str	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.
Card Type	**TS**			
Card ID	200			
Description	Defines the set of scalar values associated with a time step. Should be repeated for each time step.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	istat	SFLG int	0 1	Use status flags from previous time step. For the first time step, this value indicates that all cells are active. Status flags will be listed.
2	time	SFLT int	+	The time step value. This number is ignored if there is only one time step
	stat	SFLG int	0	Inactive

			1	Active One status flag should be listed for each cell or element. These flags are included only when istat = 1.
	val	SFLT real	+/-	The scalar values.
Card Type		**ENDDS**		
Card ID		210		
Description		Signals the end of a set of cards defining a dataset		
Required		YES		
Card Type	**RT_JULIAN**			
Card ID	240			
Description	The reference time as a Julian number.			
Required	NO			
Field	*Variable*	*Size*	*Value*	*Description*
1	reference time	8 byte float	+/-	Continuous count of days and fractions of days since noon Universal Time on January 1, 4713 BCE (on the Julian calendar).
Card Type	**TIMEUNITS**			
Card ID	250			
Description	The time units.			
Required	NO, but recommended			
Field	*Variable*	*Size*	*Value*	*Description*
1	time units	4 byte float	0 1 2	Hours Minutes Seconds

Related Topics

- File Formats
- ASCII Dataset Files (*.dat)

Boundary ID Files

Boundary ID files (*.bid) are used to store the ids of the nodes on all of the nodestrings in a mesh. The idea of this file is to export only the boundary ids. To do this, make sure nodestrings only exist along the boundary.

Boundary ID files are not opened in SMS but are saved using the menu command *File* | **Save As...** in the Mesh module . To save a Boundary ID File (*.bid), the current mesh module model should be set to ADCIRC . Use the menu command *Data* | **Switch Current Model...** to change the current mesh module model.

Sample File

```
BOUNDARY
3 Number of nodestrings
3 Number of nodes in nodestring
24
18
12
5 Number of nodes in nodestring
149
49
43
37
31
8 Number of nodes in nodestring
11
112
120
127
134
141
148
155
```

Related Topics

- Boundary XY Files
- File Formats

Boundary XY Files

Boundary XY files (*.bxy) are used to store the x,y location of the nodes on all the nodestrings in a mesh. If wanting to export only the boundary node locations, make sure nodestrings only exist on the boundary of the mesh. If nodestrings exist on the interior of the mesh, their locations will also be included in the Boundary XY file.

Boundary ID files are not opened in SMS but are saved using the menu command *File* | **Save As...** in the Mesh module . To save a Boundary XY File (*.bxy), the current mesh module model should be set to ADCIRC . Use the menu command *Data* | **Switch Current Model...** to change the current mesh module model.

Sample File

```
BOUNDARY
2 Number of nodestrings
3 Number of nodes in nodestring
500000.000000 0.000000
497592.400000 49008.600000
490392.600000 97545.200000
```

```
5 Number of nodes in nodestring
518643.200000 65498.400000
654128.400000 54688.400000
845478.800000 53491.100000
987681.700000 65385.800000
551972.600000 48687.200000
```

Related Topics

- Boundary ID Files
- File Formats

Coastline Files *.cst

Coastline files include lists of two-dimensional polylines that may be closed or open. The open polylines are converted to *Feature Arcs* and are interpreted as open sections of coastline. Closed polylines are converted to arcs and are assigned the attributes of islands. Coastline files contain x, y, z location information for arcs defining coastlines. The arcs defining a coastline can be created from ADCIRC and CGWAVE type coverages . When a Coastline file is read into SMS, feature arcs are created. If a coastline closes, the final point is not repeated. If no z-value is specified, SMS defaults the node z-value to 0.0.

Coastline files are opened through the menu *File* | **Open** and saved from *File* | **Save As...** from the Map module .

Sample File

COAST	/* File type identifier */
2	/* Number of coastlines */
1309 0	/* Number of segments in coastline and if coastline closes (not closes = 0, closes = 1) */
-7794.9054 3396.0346 0.0	/* Node X, Y, Z Locations - Z is optional (defaults to 0.0) */
-7822.6129 3391.8341 0.0	
-7852.6508 3386.68 0.0	
.	
.	
.	
151 1	/* Start next coastline */
.	
.	
.	
/* EOF */	

Related Topics

- Create Coastline
- File Formats
- ADCIRC

- CGWAVE

Color Palette Files *.pal

Color Palette (*.pal) files hold user-defined palette information for use with display contour data. Palettes must be construct using an RGB color model. SMS can both generate and import color palette files.

The *New Palette* dialog and the **Save Palettes** command in the *Color Options* dialog are used to create color palette files in SMS. The process for creating a color palette is described in the User Defined Color Palettes article.

Importing color palette files is done through the *Color Options* dialog using the **Load Palettes** command. The *Color Options* dialog will then display the color palettes saved in the file.

Each color palette file can hold information for multiple color palettes. SMS will display all available palettes in the *Color Options* dialog.

A sample palette file is shown below:

```
PALETTE                   File type identifier
2                         Number of Palettes
Palette1  3               Name of Palette; Number of colors in palette
0 0.0  1.0                Percentage (0) or Value (1); Minimum Range; Maximum Range
0.111111 0 0  0           1st color - Percentage; Red (0-255); Green; Blue
0.444444 45  136 45       2nd color - Percentage; Red (0-255); Green; Blue
1.000000 255  255 255     3rd color - Percentage; Red (0-255); Green; Blue
Palette2  5               Next Palette
...                       etc.
```

Related Topics

- File Formats
- Color Options Dialog
- Native SMS Files

Drogue Files *.pth

Drogue files contain particle/path data. Drogue plots are generated by the ADCIRC model.

Drogue files are opened through *File* | **Open** but are not saved from SMS.

Sample File

ACE/vis drogue path file			/* Title */
5			/* Number of Time Steps */
7200.0000	199		/* Current Time Step and Number of Particles */
-0.766993322E+02	0.346589454E+02	1	/* xy values and id */
-0.766986001E+02	0.346616775E+02	2	
.			
.			
8000.0000	199		/* Next Time Step and Number of Particles */
.			

```
.
/* EOF */
```

The number of particles must be the same for each time step.

Related Topics

- File Formats

File Extensions

Model specific files such as those used by FESWMS , RMA2 , and other models are documented in their respective reference documentation, which is available from the model developers, not the developers of the SMS software.

Open/Save Files

- ASCII Data (*.dat)
- AutoCAD (*.dxf)
- Binary Data (*.dat)
- Bitmap Image Files (*.bmp)
- Coastline (*.cst)
- Image (*.img)
- INI / Settings (*.ini)
- JPEG Image Files (*.jpg, *.jpeg)
- Map (*.map)
- Material (*.mat)
- Meta Data (*.met)
- Observation Table (*.obt)
- Palette File (*.pal)
- Project (*.spr; *.prj)
- Scatter Point (*.xy)
- TIN (*.tin)
- 2D Mesh (*.2dm)
- XY Series (*.xys)
- XYZ (*.xyz)

Open Only

- Arc Info (*.arc)
- MIKE 21 (*.mesh)
- Shape File (*.shp)

ADCIRC

- ADCIRC Control (*.ctl; *.15)
- ADCIRC Harmonic Solution (*.53; *.54)
- ADCIRC Simulation (*.grd; *.14)

- ADCIRC Unit 63 (*.63; *.sol)
- ADCIRC Unit 64 (*.64; *.sol)
- ADCIRC Unit 71 (*.71; *.sol)
- ADCIRC Unit 72 (*.72; *.sol)
- ADCIRC Unit 73 (*.73; *.sol)
- ADCIRC Unit 74 (*.74; *.sol)
- ADCIRC Unit 75 (*.75; *.sol)
- ADCIRC Unit 83 (*.83; *.sol)
- ADCIRC Unit 84 (*.84; *.sol)
- ADCIRC Unit 85 (*.85, *.sol)
- ADCIRC Unit 91 (*.91, *.sol)
- ADCIRC Unit 93 (*.93, *.sol)
- Drogue (*.pth)

CGWAVE

- CGWAVE Simulation (*.cgi)
- CGWAVE Solution (*.cgo; *.out)
- CGWAVE 1-D (*.cg1)

CH3D

- CH3D Simulation (*.ch3)

CMS-Wave

- CMS-Wave Simulation (*.sim)
- CMS-Wave Current File (*.cur)
- CMS-Wave Depth File (*.dep)

DAMBRK

- DAMBRK Simulation (*.dat)

FESWMS

- FESWMS Simulation (*.fil)
- FESWMS Solution (*.flo; *.out)

GENESIS

- GENESIS Solution (*.sol)

STWAVE

- STWAVE Current (*.cur)
- STWAVE Depth (*.dep)
- STWAVE Observation (*.obs)
- STWAVE Simulation (*.sim)

- STWAVE Spectral Energy (*.eng)
- STWAVE Wavefield (*.wav)

TABS

- RMA2 BC (*.bc)
- RMA2 Geometry (*.geo)
- RMA2 Simulation (*.sim)
- RMA2 Solution (*.sol)
- RMA4 Simulation (*.trn)
- RMA10 Simulation (*.bc)

WSPRO

- WSPRO Simulation (*.dat)
- WSPRO Solution (*.out)

Related Topics

- File Formats

Fleet Wind Files

Datasets can be read in from fleet wind files (*.fleetwind). SMS supports two different formats for fleet wind files, which are equivalent to NWS = 3 and NWS = 6 for PBL models. More information regarding the format for these files can be found here .

Opening the Files

When opening a fleet wind file, SMS will bring up a dialog to prompt the user for information regarding file type, grid parameters, time steps, and projection.

Format: Option to select between "Direction, speed (NWS 3)" and "X, Y, pressure (NWS 6)"

Existing Grid: If the grid specified in the fleet wind file already exists in SMS, a user can select to create the datasets on the existing grid

Grid Parameters: If the grid specified in the fleet wind file does not already exist, a user must specify the origin, cell size, and number of cells to create the grid.

Time (NWS 6 only): Specify the size and units of the time increment, as well as a reference time (if desired).

Projection: Specify the projection of the grid. If the projection is not specified, the grid will be created using the current projection.

Datasets

When SMS reads a fleetwind file, it will create the following datasets on the grid:

- Wind (vector)
- Magnitude (scalar)
- Pressure (scalar) – NWS 6 only

Related Topics

- File Formats

Importing Non-Native SMS Files

SMS can import many files generated by other software in their native format. The files that can be imported to SMS are shown in the tables below. Each file type is identified by the file extension. The file filter corresponding to the desired extension should be selected in the *Open File* dialog.

In addition to the file types listed below, several other types of data can be imported via the Import Wizard . Refer to the Import Wizard article for more information.

Import File Types

File Type	File Ext	Description
Model Simulation Files	*.sim, *.fpr, *.cmcards, *.par	Numerous numerical engines use simulation files to help organize model files. These simulation files usually reference a number of other model specific geometry, parameter or data files. Opening the simulation file will import the entire model and all associated model files. This will NOT open any SMS specific data, such as Map data or TIN data etc. The models associated with each super file extension are as follows: *.sim = TABS or STWAVE or CMS-Wave, *.fpr = FST2DH, *.cmcards = CMS-Flow, *.par = BOUSS2D
Model Geometry Files	*.net, *.geo, *.dep, *.cgi, fort.14, *.grd	The numerical engines each support their own format for storing geometry (mesh or grid). Opening the geometry file will import create a geometric object of the appropriate type (mesh or grid) and set the current model to be the associated type. Some models use standard file formats such as *.2dm or *.3dm. Themodels associated with each geometry file extension are as follows: *.net = FST2DH, *.geo = RMA2 (TABS), *.dep =CMS-Wave , *.cgi = CGWAVE, fort.14 = ADCIRC (also uses *.grd), *.grd = BOUSS2D
DXF/DWG	*.dxf, *.dwg	Vector drawing data used for background display or for conversion to feature objects.
JPEG – TIFF	*.jpg, *.tif, *.tiff	Raster image files used for background display or for texture mapping to a surface.
Shapefiles	*.shp	ArcView shapefiles.
DEM / Grid	*.asc, *.ddf, *.ggd, *.dem	ASCII 2D grid exported from Arc/Info or ArcView, ASCII 2D grid exported from GRASS
ARC/INFO® ASCII Grid Files	*.arc	Cartesian grid and associated datasets.
Drogue Files	*.pth	Particle / path data from ADCIRC.
MIKE 21 *.mesh	*.mesh	Grid file in DHI grid format.
ARC/INFO® Shapefiles	*.shp	Geographical shapes and associated attributes.
MapInfor® MIF/MID Pairs	*.mid/*.mid	Geographical shapes and associated attributes.
NOAA HURDAT , ATCF	*.hurdat, *atcf	Tropical cyclones database.
NOAA HURDAT 2	*.hurdat2	Tropical cyclones database in unprocessed format.
LIDAR	*.las, *.laz	Light detection and ranging.

Related Topics

- [File Import Wizard]
- [Native File Formats]
- [File Formats]

KMZ Files

XMS software can export KMZ files. KMZ files can be imported into Google Earth .
KMZ files can be imported into GMS , SMS , and WMS .

Raster vs. Vector

The KMZ file format supports both vector data (lines, points, polygons, triangles etc.) and raster data (images). When exporting raster data, the image shown in the XMS main graphics window is saved as a PNG image file with georeferencing data. The data is clipped to match the window bounds of the current view. When exporting vector data, all data displayed, as specified by the display options and project explorer, is exported to a raster KMZ file. The following versions of XMS software support vector export:

- GMS – 7.1
- SMS – 10.1
- WMS – 8.1

How To Export – Screen Capture

- The project must be in plan view before exporting a KMZ file.
- Export a KMZ file by using the standard *File* | **Save As** dialog and selecting either the *Google Earth© Raster KMZ File (*.kmz)* or *Google Earth© Vector KMZ File (*.kmz)* option in the *Save as type* field.
- To specify a resolution higher than the screen resolution:
  - GMS – Select the **Options** button in the *Save As* dialog.
  - SMS – Change the copy to clipboard scale factor in the Preferences dialog

The background color is made transparent in the exported KMZ file so the Earth can be seen through the overlayed image in Google Earth©.

How To Export – Transient Data Animation

Export a KMZ transient data animation using the Film Loop Wizard . This will export a series of raster images which can be animated in Google Earth©. The background color can be specified in the Film Loop Wizard . It is recommended that the option for no background be used so the Earth can be seen through the overlayed image in Google Earth©. The project must be in Plan View to export a KMZ transient data animation.

By default, Coordinated Universal Time (UTC) is assumed when exporting KMZ files. An offset from UTC can be specified. A list of time zone offsets from UTC is given here .

See "Viewing a Timeline" on page 90 of the Google Earth© user Guide or here for an explanation of how to change the time zone used by Google Earth©.

Coordinate System

KMZ files contain latitude and longitude information to define the location of the image. If the current coordinate system type is a projection, like UTM for example, and not a geographic system (which uses latitude and longitude), XMS will attempt to determine the latitude and longitude using coordinate conversion. It's possible that the coordinate conversion may fail, or that the distortion from converting from the current coordinate system to a geographic system is too high. In either case, XMS will issue a warning.

Transparency

The background color is made transparent in the exported PNG image which is in the KMZ file. This makes it so that the Earth can be seen through the overlayed image.

See Also

- Official Google Earth website
- KML documentation

Back to XMS

LandXML Files

SMS supports importing TINs from a LandXML file.

LandXML Files (*.xml)

A LandXML file is a non-proprietary file format that stores civil/survey data such as points, faces, etc. making it easier to share surfaces between different programs. Several CAD and other packages support exporting data into the LandXML format. This makes LandXML a good choice for getting bathymetry/topographic data into SMS as the connectivity will be maintained.

Locating Data in a LandXML file

SMS only supports a subset of the LandXML definition specific to TINs. The specific data SMS looks for point location, and triangular connectivity information. To locate this data within the LandXML file, start by looking under the identifier "Definition surfType". Make sure that this is defined as "TIN". Then, look for the 'Pnts' identifier to read point locations by id. Next, look for the 'Faces' identifier to read the connectivity information. Note: In the LandXML file, the points are identified by their id number as well as by their coordinates (y,x,z) respectively.

LandXML Identifiers

A complete list of LandXML identifiers with their respective definitions can be found at www.landxml.org/ on the right side of the page under LandXML-1.2 Schema. Click on the LandXML-1.2 Data Diagram.

Related Topics

- Importing Non-native SMS Files

Lidar Support

The term LIDAR refers to a technology known as LIght Detection And Ranging. This technology is used to gather a large amount of survey data from fixed wing aircraft or helicopters as they fly over the survey region. The technique has the ability to penetrate clear water up to a limited depth (maybe up to 20 meters but varies based on conditions).

A LIDAR file (normally it has an extension of ".las" or ".laz") is a survey file that includes data points gathered from a LIDAR system and processed.

Importing a LIDAR File

When a lidar is read in, SMS uses the Global Mapper library. The *LIDAR Load Options* dialog appears to allow the user to specify importing options. Principally, the user specifies whether the data should be loaded as a raster or scatter. Then create a raster or scatter set . This dialog is actually a component of the Global Mapper library. Therefore, the options in the dialog are described as a point cloud (scatter set) or a grid (raster).

This dialog also lists all the data sources/types of data included in the file and allows the user to select which data types should be imported.

Reading as a point cloud (scatter set)

If the user selects point cloud, SMS creates both a scatter set and an image. Note that because this is not a uniformly arranged set of data and thus not a raster, all of the raster editing options will be unavailable for this data. There are several options for the appearance of the image. These include:

- Use Colors if Present (Elevations Otherwise) – The points in the LIDAR set are given a color. This appears like a contour plot image.
- Color Elevations Using Current Shader – currently SMS does not allow the user to select shader options, so this option also appears as a contour plot image.
- Color by intensity – this results in a gray scale image.
- Color by classification – the points in the LIDAR set are colored based on their type. This can be useful to see what types of data (layers) exist in the file. If all the points in a file are the same type, the image is a single color.
- Color by random number – colored points – no documentation is available for this feature

The scatter points created when reading a LIDAR file are not triangulated. They must be triangulated to extract observation profiles or use them as source of bathymetry.

Reading as an elevation grid (raster)

If the user selects grid, SMS creates a raster and an image. SMS adds the raster as a VTK raster and the image as an image. The image appears as a hillshade.

This option is much slower than the point cloud option.

Options include:

- *Elevation Grid creation Options* – Tight to Loose
    - Loose (default) – for a test case it created 63752617 points/cells
    - Medium – for test case it created 63714287 points/cells with some holes
    - Tight – for test case it created 15058938 points that are mostly disjoint
- If the LIDAR data is not a box, SMS can fill either the entire box or just the convex hull with gridded data.

Holes left in the grid are also reflected as holes in the image. The tighter the option, the slower the process appears to be.

LIDAR Images

Reading a LIDAR file always results in an image in SMS. For some images, SMS allows the conversion to a RASTER. Since that option is available when reading the LIDAR file, this capability is not available for images created as a side effect of reading a LIDAR file.

General Options

In addition to filtering out point classifications, SMS supports the following options:

- Preview mode – load 1 in N points (user specifies N)
- Delete samples over N standard deviations from the mean (user specifies N)
- Swap elevations (multiply by -1)

Related Topics

- Rasters
- Scatter Module

Map Files

When SMS saves a project, all the data related to coverages in the Map Module is written to a single ASCII file named *proj_name.map* .

Although this file is ASCII and can therefore be read in a text editor, it's recommend that a user not edit these files. It's also not recommend that a user develop utilities based the current format of these files. The format has changed between versions of SMS, and although SMS maintains backward compatibility to read older file formats, there is no guarantee that older formats will be written by future versions of SMS. In fact, the ASCII file may not be maintained at all.

Related Topics

- File Formats
- Native SMS Files

MapInfo MID/MIF

MapInfo Interchange Format is a map and database exporting file format of MapInfo software product. The MIF-file filename usually ends with .mif-suffix. Some MIF-files also have a related MID-file.

Export

To export maps in MIF/MID format:

1) Load or create an existing map
2) Select *File* | **Save As**
3) Select *Save as type:* and choose "Catalog file (*.xml)"

An XML file is created which outlines the directories of the MIF/MID files. A points.mif/mid, polygons.mif/mid and polylines.mif/mid (if exists) is also created.

Import

Using the *File* | **Open** menu command, open the saved XML file to import the mesh from a MIF/MID file.

After selecting a MIF/MID file, the *MIF/MID Import* dialog may appear. Users have the option to import the MIF/MID file as one of the following options:

- GIS Layer
- TUFLOW 2D/2D Linkage
- TUFLOW 2D Z Lines/Polygons (simple)
- TUFLOW Area Property
- TUFLOW Boundary Condition (flow over area)
- TUFLOW Boundary Condition (rainfall)
- TUFLOW Codes
- TUFLOW Flow Constriction (cell based)
- TUFLOW From Loss Coefficient
- TUFLOW Initial Water Level
- TUFLOW Storage Reduction Factor
- TUFLOW Water Level Lines
- TUFLOW Water Level Points
- TUFLOW Weir Factor

MIF-MID files can also be imported using the **Add MIF/MID File Data** command in the *Data* menu of the GIS module.

Related Topics

- Importing Non-native SMS Files

External Links

- MapInfo MIF/MID Format at MapInfo.com

Material Files *.mat

Each element of a 2D mesh has an assigned material ID. Specific material properties are related to the analysis models, and are stored in the analysis files. However, general material properties, such as color, are not stored in these files. Therefore, they are stored in the material file. A material ID represents an index to a global list of materials. The material file associates general attributes such as a name, color, and pattern with each of the materials.

Material files are opened through *File* | **Open** and are saved from *File* | **Save As...** from the Mesh module.

File Format

The file format for material files is as follows:

```
MAT                      /* File type identifier */
MN id name               /*Material name */
MC id red green blue     /* Material color */
MS id stippleid          /* Material stipple (fill pattern) */
```

Cards

Card Type	**MAT**		
Description	File type identifier. Must be on first line of file. No fields.		
Required	YES		
Card Type	**MN**		
Description	Identifies a name to be associated with the material.		
Required	NO		
Format	MN id name		
Sample	MN 5 bedrock		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID of the material.
2	name	str	The name of the material.
Card Type	**MC**		
Description	Identifies a color to be associated with the material.		
Required	NO		
Format	MN id red green blue		
Sample	MN 5 124 67 245		
Field	*Variable*	*Value*	*Description*

1	id	+	The ID of the material.
2	red	0-255	The value of the red component of the color.
3	green	0-255	The value of the green component of the color.
4	blue	0-255	The value of the blue component of the color.
Card Type	**MS**		
Description	Identifies a stipple (fill pattern) to be associated with the material. This stipple is used whenever an object is being drawn using color filled polygons.		
Required	NO		
Format	MN id stippleid		
Sample	MN 5 13		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID of the material.
2	stippleid	+	The ID of the stipple.

Related Topics

- File Formats
- Native SMS Files

MIKE 21 *.mesh

A MIKE 21 mesh file is a text/plain file and has the *.mesh extension. The text file defines a simple unstructured grid and is similar to a *.2dm file. It lists the node and a simplified element table in text format using spaces as separators. The MIKE 21 program is produced and distributed by the DHI group.

If a grid/mesh is available in this format, it can be loaded into SMS and used for further model construction/management. SMS reads the grid, stores it as a two-dimensional unstructured mesh and allows the user to associate it with any of the numerical engines supported in SMS by using the *Data* | **Switch Current Model** command.

Loading a MIKE 21 *.mesh file into SMS

To load the file:

1) *File* | **Open...** or *File* | **Open As...**
2) In the *Open file* dialog change the *Files of type* to "Mike 21 files (*.mesh)"
3) Select the file and press **Open**

MIKE 21 File Format

A MIKE 21 mesh file uses the following format:

- **Header line** – The first line. This line contains two integers expressing the type and unit of the bathymetry data. This followed by the number of nodes then by a string defining the projection.
- **Node lines** – Every line after the header line for as many lines as stated in the header line. Each line contains information for one node in the format as follows:
    - The node point Id

  - The x coordinate (Easting or Longitude)
  - The y coordinate (Northing or Latitude)
  - The z value (bathymetry)
  - The node type (0 for water, 1 for land and above 1 for all other boundaries)
- **Element header line** – One line containing three numbers. The numbers represent the following:
  - The number of elements
  - The maximum number of nodes per element
  - Internal code for mesh type (21 for purely triangular elements, 25 if quadrangular elements exists)
- **Element lines** – Every line after the element header line for as many lines as stated in the element header line. Each line contains information for one element in the format as follows:
  - The element Id
  - Id for node_1
  - Id for node_2
  - Id for node_3
  - (Id for node_4)

  Each line contains at most as many nodes as stated in the element header line. If an element has fewer nodes than the maximum number of nodes per element stated in the element header line then the remaining node Id value is 0.

For additional information, see the DHI User Guide [200] .

Related Topics

- File Formats
- File Import Wizard
- Importing Non-native SMS Files

Native SMS Files

Native SMS Files are files SMS can read and write.

SMS Interface Files

SMS interface files contain data used by SMS to save display options, user preferences, etc.

SMS Interface Files include:

File Type	File Extension	Purpose
Color Palette Files	*.pal	User defined contour palette information
Settings (INI) Files	*.ini	SMS preferences and default settings

Generic Data Files

A generic file is defined as any file that was not formatted for a specific numeric model. Model specific files such as those used by FESWMS , RMA2 , and other models are documented in their respective reference documentation, which is available from the model developers, not the developers of the SMS software.

Most of the generic files used by SMS use a modified form of the HEC style card type format. With this format, the different components of the file are grouped into logical groups called "cards." The first component of each card is a short name that serves as the card identifier. The remaining fields on the line contain the information associated with the card. In some cases, such as lists, a card can use multiple lines.

While card style input makes the file slightly more verbose, there are many advantages associated with the card type approach to formatting files. Some of the advantages are:

- Card identifiers make the file easier to read. Each input line has a label, which helps to identify the data on the line.
- The card names are useful as text strings for searching in a large file. All input lines of a particular type can be located quickly in a large input file.
- In many cases, Cards allow the data to be input in any order (i.e., the order that the cards appear in the file is usually not important).
- Cards make it easy to modify a file format. New card types can be added without invalidating older files. New files have additional data in the new cards. The new card must be optional (which is typically the case for new cards) for old files to remain compatible. If an old card type is no longer used, the card can simply be ignored without causing input errors.

The generic file formats supported by SMS include:

File Type	File Extension	Purpose
2D Mesh Files	*.2dm	Finite element mesh and generic model interface template definition
2D Scatter Point Files	*.xy	Scatter point x, y location
ASCII Dataset Files	*.dat	Mesh, grid, or scatter set datasets
Binary Dataset Files	*.dat	Mesh, grid, or scatter set datasets
Boundary ID Files	*.bid	Export node ids of nodestring nodes
Boundary XY Files	*.bxy	Export node x, y locations of nodestring nodes
Coastline Files	*.cst	Export node and vertice x, y locations of coastline arcs
Map Files	*.map	File that contains information for feature data
Material Files	*.mat, *.materials	Mesh material options for SMS (display options, name, etc.)
Quad4 Files	*.qd4	QUAD-4 format finite element mesh file
Shapefiles	*.shp	GIS vector data file
SMS Super Files	*.sup	File containing references to geometry and dataset files
Tabular Data Files – SHOALS	*.pts	Mesh node, or scatter point locations and values
TIN Files	*.tin	Triangulated irregular network data
XY Series Files	*.xys	X, Y series data curves
XYZ Files	*.xyz	Mesh node, or scatter point locations and values

Related Topics

- Import Wizard
- Non-native SMS Files
- File Formats

NOAA HURDAT

The North Atlantic hurricane database , or HURDAT, is the database for all tropical cyclones in the Atlantic Ocean, Gulf of Mexico and Caribbean Sea. Starting with version 11.0, SMS has been able to open an *easy read* HURDAT file with the *File* | **Open** command or using the drag/drop technique. SMS uses the file extension of **.hurdat* to identify this format. If the database has been downloaded and another extension put on the file (such as *.dat*) SMS will not automatically recognize the file as an easyread HURDAT file and will ask the user to specify the file type. In this case, the user should select the HURDAT easy read format. The *easy read* format is a process format that is usually available by accessing the URL with an extension for this format (i.e. */hurdat/easyread-2009.html* , */hurdat/easyread-2011.html* , or */hurdat/easyread-2012.html*). As the data is reprocessed it can be accessed in this form usually around 2 years after the events occur. The format is slightly easier for humans to read.

Starting with version 12.0 of SMS, support was expanded to support the unprocessed HURDAT format (HURDAT2). SMS recognizes this format using the file extension of *.hurdat2* . This data is accessible from NOAA for both the North Atlantic and Eastern Pacific basins.

Also beginning with version 12.0, SMS added a feature to allow the user to filter available storms from the database file based on the following optional and user specified criterion:

- Date range – only storms whose data overlaps specified range will be displayed.
- Lat/Lon range – only storms that pass through the specified rectangle will be displayed.
- Speed range – only storms which include at least one data point of storm speed (knots) inside the range will be displayed.
- Wind speed range – only storms which include at least one data point with a maximum wind speed (mph) inside the range will be displayed. This would typically be used to filter out storms which never reached a minimum strength.
- Pressure range – only storms which include at least one data point with a minimum central pressure (mb) inside the range will be displayed.

The user may select as many filtering criterion as desired and press the **Run** button. Storms which pass the specified filter limits will be displayed in the list. The user may then select as many of these storms as are desired and click the **OK** button to read the storm parameters into SMS as Wind model coverages.

File Format

HEADER

```
92620  08/16/1992  M=13   2  SNBR=  899  ANDREW       XING=1  SSS=4
Card#  MM/DD/Year  Days  S#  Total#...  Name........US  Hit.Hi  US  category
```

DAILY DATA

```
92580  04/22S2450610   30  1003S2490615   45  1002S2520620   45  1002S2550624   45  1003*
Card#  MM/DD&LatLongWindPress&LatLongWindPress&LatLongWindPress&LatLongWindPress
```

TRAILER

```
92760  HRCFL4BFL3  LA3
Card#  TpHit.Hit.Hit.
```

Header

- **Card#** – Sequential card number starting at 00005 in 1851
- **MM/DD/Year** – Month, Day, and Year of storm

- **Days** – Number of days in which positions are available (note that this also means number of lines to follow of Daily Data and then the one line of the *Trailer)
- **S#** – Storm number for that particular year (including subtropical storms)
- **Total#** – Storm number since the beginning of the record (since 1851)
- **Name** – Storms only given official names since 1950
- **US Hit** –
  - '1' – Made landfall (i.e., the center of the cyclone crossed the coast) on the continental United States as a tropical storm or hurricane,
  - '0' – did not make a U.S. landfall
- **Hi US category** –
  - '0' – Used to indicate U.S. tropical storm landfall, but this has not been utilized in recent years
  - '1' to '5' – Highest Saffir-Simpson Hurricane Scale impact in the United States based upon extimated maximum sustained surface winds produced at the coast. See scale below.

Daily Data

- **Card#** – As above.
- **MM/DD** – Month and Day
- Positions and intensities are at 00Z, 06Z, 12Z, 18Z
  - **&** –
    - '*' (tropical cyclone stage),
    - 'S' (Subtropical stage)
    - 'E' (extratropical stage)
    - 'W' (wave stage – rarely used)
    - 'L' (remanent Low stage – rarely used)
- **Lat** – Latitude of storm: 24.5N
- **Long** – Longitude of storm: 61.0W
- **Wind** – Maximum sustained (1 minute) surface (10m) windspeed in knots (these are to the nearest 10 knots for 1851 to 1885 and to the nearest 5 kt for 1886 onward).
- **Press** – Central surface pressure of storm in mb (if available). Since 1979, central pressures are given everytime even if a satellite estimation is needed.

Trailer

- **Card#** – As above.
- **Tp** – Maximum intensity of storm
  - 'HR' – hurricane
  - 'TS' – tropical storm
  - 'SS' – subtropical storm

- **Hit** – The impact of the hurricane on individual U.S. states ('LA' = Louisiana, etc.) based upon the Saffir-Simpson Scale category (through the estimate of the maximum sustained surface winds for each state). See scale below. Occasionally, a hurricane will cause a hurricane impact (estimated maximum sustained surface winds) in an inland state. To differentiate these cases versus coastal hurricane impacts, these inland hurricane strikes are denoted with an "I" prefix before the state abbreviation. States that have been so impacted at least once during this time period include Alabama (IAL), Georgia (IGA), North Carolina (INC), Virginia (IVA), and Pennsylvania (IPA). The Florida peninsula, by the nature of its relatively landmass, is all considered as coastal in this database.

  Note that Florida and Texas are split into smaller regions:

  - 'AFL' – Northwest Florida
  - 'BFL' – Southwest Florida
  - 'CFL' – Southeast Florida
  - 'DFL' – Northeast Florida
  - 'ATX' – South Texas
  - 'BTX' – Central Texas
  - 'CTX' – North Texas

Saffir-Simpson Scale

Saffir-Simpson Category	Maximum sustained wind speed		
	mph	kts m/s	kts
1	74-95	33-42	64-82
2	96-110	43-49	83-95
3	111-130	50-58	96-113
4	131-155	59-69	114-135
5	156+	70+	136+

Related Topics

- Non-Native Formats

Quad4 Files

QUAD4 files are exported from SMS for use with a proprietary finite element program called QUAD-4. QUAD-4 uses an equivalent-liner soil model. Basic input to QUAD4M includes the two-dimensional soil profile, equivalent-linear soil properties, and the time history of horizontal ground motion. The file format is a specific format listed in the QUAD-4 documentation.

Related Topics

- File Formats

External Links

- Idriss, I.M., Lysmer, John, Hwang, Richard N., and Seed, H. Bolton, "QUAD-4, A Computer Program for Evaluating the Seismic Response of Soils Structures by Variable Damping Finite Elements, "Earthquake Engineering Research Center, Report No. EERC 73-16, University of California, Berkeley, June 1973.
- The Earthquake Engineering Online Archive QUAD-4 and QUAD4M Software and Manuals

Settings Files *.ini

Settings, or INI, files store all of the user defined settings that made inside of SMS. For example, the user defined display options, the coordinate system, mesh, grid, and scatter options, etc. are stored in the file. These files should not be confused with the old .ini files that were previously used as FESWMS Initial Condition files.

SMS.INI

The main settings file, called sms.ini, is created when SMS is started for the first time. The file is then opened every time SMS starts. When making a change, such as setting mesh nodes to draw in blue, the change is permanently saved by invoking the *File* | **Save Settings** command, which updates the sms7.ini file. If the sms7.ini file is deleted, the next time that SMS begins, the settings will be reset to the factory defaults.

Other Settings Files

Settings files are also saved with a project file. The next time a project is loaded into SMS, the project settings are restored and the SMS environment returns to its saved state.

File Format

Settings files are ASCII text files. Each SMS setting is stored with a keyword and a value. For example:

```
Show  Welcome  Dialog=0
```

signifies that SMS will not show the *Welcome* dialog on startup.

Related Topics

- File Formats
- Native SMS Files

Shapefiles

One common method for creating feature objects is to import a shapefile. The concept of a shapefile was established by Environmental Systems Research Institute (ESRI) in their ArcView® program and it has become the defacto standard for sharing GIS vector data (points, lines, and polygons).

Shapefiles contain data exported from ARC/INFO® or ArcView® in binary format. When they are imported into SMS, the data is converted to feature objects, points, arcs, or polygons. Shapefiles are opened through *File* | **Open** . Only map data can be saved out from SMS as a shapefile.

A shapefile is actually comprised of three or more files. The primary file is the SHP and it contains the geometric information (coordinates and if necessary connectivity of the points, lines, polygons). The DBF file is a standard database file and stores the attributes of the feature objects. Finally there will be a SHX file which is an indexing file. There may be a few other files that accompany the shapefile and so always move them around together if copying or moving them to a new directory.

Only one "theme" or type of feature can exist in a shapefile. For example it's not currently possible to store points and polygons in the shapefile, or streams and basin boundaries and so it may be required to import multiple files to make up the drainage coverage in SMS.

When a shapefile is opened, the *Import Shapefile Data* dialog appears. It is necessary to know if the file is a Point, Arc, or Polygon shapefile. The options in the dialog are:

- **Coverage Options** – Bring up the *Coverage Options* dialog to set the coverage in which the shapefile data is created.
- **Points / Arcs / Polygons** – Under each of these sections, the file browser button allows selecting a point, arc, and polygon file. One file of each type may be selected.

- **Attribute Mapping** – This button brings up the *Map Shapefile Attributes* dialog. In this dialog, select an attribute from each of the Database (from the shapefile) and Coverage attributes (SMS supported attributes) fields. If an attribute is selected from each, the **Map** button maps the attributes and the **Unmap** button unmaps the attributes. If an attribute is mapped, the attributes will be assigned in SMS when the file is opened.

SMS includes all of the tools necessary to import shapefiles and convert the geometric and attribute information into feature objects. This can be done by directly opening the shapefile and converting to feature objects in the active coverage or by loading the shapefile in the GIS module.

Export Map Data in Shape Format

Data created in SMS can be exported as a shapefile. To export feature objects in a map coverage to a shapefile (*.shp):

1) Load or create feature objects in a map module coverage.
2) Select *File* | **Save As**
3) Select *Save as type:* and choose "Shapefiles (*.shp)"
4) In the *Export Shapefile* dialog, select the feature object types to be saved.
5) Click **Ok**

Shapefiles are created for the selected feature object types.

Open the saved XML file to import the data from the shapefile.

Related Topics

- Data Acquisition
- Importing Shapefiles
- GIS Module
- Native SMS Files
- Feature Objects

SMS Super Files *.sup

SMS Super files were used in previous versions of SMS to save most of the working data in SMS. Super files have been replaced by Project files. Old super files are still opened in SMS. Super files are only saved from the *File* | **Save Scatter Super File** command. These contain a 2D Scatter Point file and the corresponding ASCII data file.

A super file contains a list of other files. Each of the files in the list must be one of the basic SMS file types (2D meshes, 2D scatter points, materials, TINs). If a super file is selected using the *File* | **Open** command, each of the files listed in the super file are opened and imported. This makes it possible to quickly read in several files without having to identify each file individually in the file browser.

The file format for a super file is shown below. The first line in the file is the SUPER card, which identifies the file as a super file. Each of the other cards shown are optional. Each of the file cards has a card identifier representing the type of file. The identifier is followed by a file name. The file name should be a complete path if the file is not in the same directory as the super file. Any suffix may be used for the file name.

File Format

SUPER	/* File type identifier */
MAT filename	/* Material File */
SCAT2D filename	/* 2D scatter point file */
MAP filename	/* Map file */
MESH2D filename	/* 2D Mesh file */

DATA filename	/* Dataset File */
STNGS filename	/* Settings (*.ini) File */
IMAGE filename	/* Image file */

Sample File

```
SUPER
MAT     c:\SMS\DATA\SITE1\site1.mat
SCAT2D  c:\SMS\DATA\SITE1\site1.xyf
```

Related Topics

- File Formats
- Native SMS Files

Tabular Data Files - SHOALS *.pts

SMS includes the ability to import and export tabular data files. These files can include any number of columns of data. Select which columns are to be imported and how they are to be interpreted. The columns of data follow an optional header and may be delimited by any character (such as *TAB* , *SPACE* , *COMMA* , etc.). Files with the extensions of *.xyz and *.pts are defaulted to be of this type.

History

This capability was originally developed to support files generated by the SHOALS group of the US Army Corps of Engineers . SHOALS files generally have an optional file header describing the data in each column of the file.

Importing Tabular Data

Tabular data files are opened using the *File* | **Open** menu command. Select the tabular data file to open. The *Text Import Wizard* appears which allows selecting how the data should be interpreted (as scatter points, mesh nodes, or map nodes) and which columns should be imported.

Exporting Tabular Data

Tabular data files are saved using the *File* | **Save As...** menu command and selecting *Tabular Data Files (*.txt)* or *Shoal Files (*.pts)* from the save as type filter combo box.

When in the Scatter module this saves scattered data vertices to the tabular file. When in the Mesh module this save the mesh nodes into the tabular data file. In either case the Export Tabular File dialog appears to support this operation.

Related Topics

- File Formats
- Native SMS Files

TIN Files

TIN files are used for storing Triangulated Irregular Networks. The TIN file format is shown below and a sample file is shown after. The TIN file format can be used to import a simple set of xyz coordinates since the triangle information (beginning with the TRI card) does not need to be present. If there is a file of XYZ coordinates, only add the TIN, BEGT, and VERT nv cards to the top of the file and the ENDT card at the end.

```
TIN                /* File type identifier */
```

```
BEGT                /* Beginning of TIN group */
TNAM name           /* Name of TIN */
TCOL id             /* TIN material id */
VERT nv             /* Beg. of vertices */
x1 y1 z1 lf1        /* Vertex coords. */
x2 y2 z2 lf2
.
.
.
xnv ynv znv lfnv
TRI nt              /* Beg. of triangles */
v11 v12 v13         /* Triangle vertices */
v21 v22 v23
.
.
.
vnt1 vnt2 vnt3
ENDT                /* End of TIN group */
```

Sample TIN File:

```
TIN
BEGT
TNAM Aspen
TCOL 255 255 255
VERT 408
0.0 3.1 7.8 0
5.3 8.7 4.0 1
.
.
2.4 4.4 9.0 1
TRI 408
5 1 4
4 1 2
.
.
4 2 3
ENDT
```

Cards used in the TIN file

Card Type	**TIN**
Card ID	3000
Description	File type identifier. Must be on first line of file. No fields.
Required	YES
Card Type	**BEGT**

Card ID		3000	
Description		Marks the beginning of a group of cards describing a TIN. There should be a corresponding ENDT card at a latter point in the file. No fields.	
Required		YES	
Card Type	**TNAM**		
Description	Provides a name to be associated with the TIN.		
Required	NO		
Format	TNAM name		
Sample	`TNAM aspen`		
Field	Variable	Value	Description
1	name	str	The name of the TIN.
Card Type	**TCOL**		
Description	Defines a default color for the triangles of the TIN		
Required	NO		
Format	TCOL color_red color_green color_blue		
Sample	`TCOL 255 255 255`		
Field	Variable	Value	Description
1	color_red	0–255	The red color component of TIN triangles.
2	color_green	0–255	The green color component of TIN triangles.
3	color_blue	0–255	The blue color component of TIN triangles.
Card Type	**MAT**		
Description	Associates a material id with the TIN. This is typically the id of the material which is below the TIN.		
Required	NO		
Format	MAT id		
Sample	MAT 3		
Field	Variable	Value	Description
1	id	+	The material ID.
Card Type	**VERT**		
Description	Lists the vertices in the TIN		
Required	YES		
Format	VERT nv x_1 y_1 z_1 lf_1 x_2 y_2 z_2 lf_2		

	. . $x_{nv}\ y_{nv}\ z_{nv}\ lf_{nv}$		

Sample	`VERT 4`
	`0.0 3.1 7.8 0`
	`5.3 8.7 4.0 1`
	`2.4 4.4 9.0 1`
	`3.9 1.2 3.6 0`		

Field	Variable	Value	Description
1	nv	+	The number of vertices in the TIN
2–4	x,y,z	±	Coords. of vertex
5	lf	0,1	Locked / unlocked flag for vertex (optional). 0=unlocked, 1=locked. Repeat fields 2-5 nv times.
Card Type	**TRI**		
Description	Lists the triangles in the TIN		
Required	NO (a set of triangles can be generated from the vertices)		
Format	TRI nt $v_{11}\ v_{12}\ v_{13}$ $v_{21}\ v_{23}\ v_{23}$ . . $v_{nt1}\ v_{nt2}\ v_{nt3}$		

Sample	`TRI 4`
	`5 1 4`
	`4 1 2`
	`4 2 3`
	`5 4 3`		

Field	Variable	Value	Description
1	nt	+	The number of triangles in the TIN.
2–4	v1,v2,v3	+	Vertices of triangle listed in a counter-clockwise order. Repeat nt times.
Card Type		**ENDT**	
Card ID		3000	
Description		Marks the end of a group of cards describing a TIN. There should be a corresponding BEGT card at a previous point in the file. No fields.	
Required		YES	

XY Series Files

The *XY Series Editor* is used to define x, y series data curves. These data curves can then be used to define such things as:

- Time dependent boundary conditions
- Rating curves

XY Series Files (*.xys)

XY Series files are imported and exported through the *XY Series Editor* .

XYS Format

```
Line 1  – XYS file identifier, Curve ID, Number of Points, and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XYS 1 5 Head

0.0	0.0
1.0	2.0
2.0	7.0
3.0	8.0
4.0	9.5

XY1 Format – Discontinued Format

Both the x and y values are listed for each point on the curve. There is no limit to the spacing or interval used between subsequent x values.

```
Line 1  – XY1 file identifier, Curve ID, Number of Points, Delta X (not used), Delta
Y (not used), Repeat (not used), Begin X Cycle (not used), and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XY1 1 5 0 0 0 0 Head

0.0	0.0
1.0	2.0
2.0	7.0
3.0	8.0
4.0	9.5

XY2 Format – Discontinued Format

Identical to the XY1 card except that the number of points and the x values are assumed to be static and cannot be altered by the user.

```
Line 1  – XY2 file identifier, Curve ID, Number of Points, Delta X (not used),
Delta Y (not used), Repeat (not used), Begin X Cycle (not used), and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XY2 1 5 0 0 0 0 Head

0.0	0.0
1.0	2.0
2.0	7.0
3.0	8.0
4.0	9.5

XY3 Format – Discontinued Format

The x values are defined by a beginning x value, an increment in x, and a percent change in x per increment (applied after adding the increment). Only the y values are explicitly listed. The x-values are calculated by starting with the Initial X value given and recursively adding the X Increment and then multiplying by the X Percent Change.

```
Line 1 – XY3 file identifier, Curve ID, Number of Points, Initial X, X Increment, X Percent Change, Delta X (not used), Delta Y (not used), Repeat (not used), Begin X Cycle (not used), and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XY3 1 5 0.0 2 10.0 0 0 0 0 Head

0.0

2.0

7.0

8.0

9.5

Related Topics

File Formats

XY Series Editor

The *XY Series Editor* is a special dialog that is used to generate and edit curves defined by a list of x and y coordinates. The curve can be created and edited by directly editing the xy coordinates using a spreadsheet list of the coordinates. An entire list of curves can be generated and edited with the Editor and curves can be imported from and exported to text files for future use. It is also possible to paste xy data directly to the spreadsheet.

The *XY Series Editor* is used in GMS, SMS, and WMS. It was designed to be general in nature so that it could be used anywhere that a curve or function needs to be defined. In some cases, the x values of the curve must correspond to a pre-defined set of values. For example, the x values may correspond to a set of time steps whose interval is established in a separate dialog. In such cases, the x fields cannot be edited but the y values associated with the pre-defined x values can be edited. In other cases, there is no limit on the number of x values or on the x spacing and both the x and y values can be edited.

The XY Edit Fields

The two vertical columns of edit fields on the left side of the dialog are for direct editing of the xy series values. A pair of application specific titles appears at the top of the columns.

The buttons below the xy edit fields are used to manipulate the values in the edit fields. The buttons are as follows:

Use dates/times

For selected situations such as entering time series data in the Map module, it is useful to enter the data in date/time format. Checking this toggle allows the x values in the curve to be entered in date/time format.

Import/Export Buttons

The **Import** and **Export** buttons allow reading in or saving an xy series file.

The XY Series Plot

The window in the upper right hand corner of the *XY Series Editor* is used to plot the curve corresponding to the xy values in the edit fields. As each value in the edit fields is edited, the corresponding point on the curve is adjusted instantaneously. Plot options are accessed by right-clicking on the plot.

Related Pages

- GMS main page
- SMS main page
- WMS main page

XYZ Files

XYZ files are opened through the *File Import Wizard* . Any ASCII tabular file can be opened through the wizard by renaming the file to have a *.xyz file extension. XYZ files may have an optional file header describing the data in each column of the file. The columns of data follow the header and may be delimited by any character (such as *TAB* , *SPACE* , *COMMA* , etc.). XYZ files are opened using the *File* | **Open** menu command.

The *File Import Wizard* is used to import delimited or fixed width files. See the File Import Wizard article for more information.

Sample File

Below is an example of an XYZ file for ten points using a space to separate each column.

```
XYZ
2.19786556e+004 9.60065430e+003 3.35000000e+002
2.20193127e+004 9.56005650e+003 3.10144900e+002
2.20619056e+004 9.52148270e+003 3.07512200e+002
2.21044428e+004 9.48274190e+003 3.05989400e+002
2.21468239e+004 9.44378440e+003 3.07568900e+002
2.21899932e+004 9.40571910e+003 3.08509600e+002
2.22319898e+004 9.36635290e+003 3.17684400e+002
2.22753169e+004 9.32845010e+003 3.22013100e+002
```

```
2.23182063e+004 9.29005790e+003 3.23000000e+002
2.23521385e+004 9.26219550e+003 3.23000000e+002
```

Related Topics

- File Import Wizard
- File Formats

Generic Vector/Raster Files

There are several kinds of vector files and many kinds of raster files that are automatically identified and read into SMS.

These kinds of vector files include:

- DXF/DWG
- ESRI Shapefiles – read into GIS module
- MapInfo MIF/MID files – read into GIS module

Raster data is read in as images or rasters (for example DEMs). SMS will recognize most image and DEM formats.

The generic vector/raster files can be used to read in vector data not natively supported by SMS or not recognized as a raster/image format. Data unrecognized by SMS should try to load the data as generic vector/raster data. If this fails, Use the **Open As** command and specify for the type "Generic vector/raster data."

Generic vector data is converted into one of the natively supported types as selected when importing. After the conversion has been made, SMS will link to the converted data and not the original file. The conversion is performed by "GlobalMapper from Blue Marble Geographics." All of the formats supported by this package are supported. The list of formats can be found here: http://www.globalmapper.com/product/formats.htm .

Related Topics

- Raster Module

7.2.b. GSDA

GSDA

GeoSpatial Data Acquisition Home

DEM

Bathymetry

TIN

Streams

Rivers, Lakes, and Seas

DRG Images

Aerial and Satellite Photos

Nautical Charts

Meteorologic Data

Coastline

Tidal

Current

Wave

Land Use

Soil Type

GSDA:Digital Elevation

Topography and Terrain

Obtain DEMs directly from WMS using the Get Data tools

WMS offers options to download various types of digital elevation data for anywhere in the world using the Get Data Toolbar. Use either the Get Data From Map button to select an area and download data for the selected area or the Get Data tool to get data for a selected area in the WMS window. This is the recommended method to get DEM data into WMS unless you have other DEM sources that may be more recent or of better quality.

Find US DEMs and bathymetry download sites

The United States Interagency Elevation Inventory is a great source of high-resolution elevation and bathymetry information. If you need higher resolution elevation data, LIDAR data, or bathymetry data that cannot be downloaded directly from WMS, use this web site to locate your area of interest and find data for that area.

Obtain Data from USGS National Map Viewer Application

The USGS provides DEM, imagery, land use, hydrography, and other datasets as part of the National Map. The types of data provided as part of this system include:

Shuttle Radar Topography Mission

30 Meter – Data available for contiguous US, Hawaii and southern Alaska
90 Meter *ndash; Extensive coverage including most areas of Earth

National Elevation Dataset (NED)

1/9 Arc Second – 3 meter resolution limited to select areas of the United States
1/3 Arc Second – 10 meter res. available covering most of the contiguous US
1 Arc Second – 30 meter res. for all of the United States
2 Arc Second – Dataset specific for the state of Alaska

Details:

SRTM and NED data is offered in Gridfloat, ArcGRID, and TIFF formats.
You can define a custom, seamless area to download.
The interface allows you to display various GIS layers to aid in the selection process.
Downloads are free (no cost).

Download various data (including elevation) from the USGS

Details:

Download DEM data in various formats.
Download various land cover and vegetation data.
Download USGS imagery.
Downloads are free, but an account must be created to download.

Download various data collected by NASA

Details:

Download DEM data from various sources and in various formats from around the world.
Downloads are free, but an account must be created to download.

DEM Data from webGIS

Details:

Terrain data offered

1. USGS 7.5 minutes - 1:24K scale (~30 m res) <Covers all of contiguous United States>
2. USGS 1 degree - 1:250K scale (~90 m res) <Hawaii and Alaska only>

Data only available in *.dem format

Graphical US map makes locating an area easy

Downloads are free (no cost).

Canadian DEM Data

Worldwide DEM data in SRTM format-90 meter resolution

SDTS DEM data from MapMart

Details:

Terrain data offered

10 Meter SDTS (Spatial Data Transfer Standard) <Limited coverage of US>
30 Meter SDTS <Covers all of contiguous United States and Hawaii>

Limit of 10 quads per download
Graphical US map makes locating an area easy
Downloads are free (no cost).
The following types of formats are also available for a fee

USGS DEM
XVY ASCII Grid
Arc ASCII Grid
DXF Mesh

SDTS DEM Data from GIS Data Depot

Details:

Digital Elevation Model (DEM) data:

10 Meter Resolution
30 Meter Resolution

SDTS Format
Extensive coverage of the U.S.
Free downloads.
Data offered free may also be available in other formats for a fee.

Tips:

The USGS GNIS and Google Maps are useful tools to determine the *name* of the DEM you need.

DEM Data from Land Info

Details:

Digital Elevation Model (DEM) data:

Numerous formats

- 1:250K Resolution
- 1:24K Resolution

Downloads must be purchased

- *.dem
- ESRI GRID
- ESRI BIL w/HDR
- *.dted

Bathymetric

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA_NGDC.png

Obtain bathymetry data using GEODAS Design-a-Grid

The National Oceanic and Atmospheric Administration (NOAA) has produced the National Geophysical Data Center (NGDC) website to allow for US data access. Oceanic bathymetry can be found using GEODAS (GEOphysical DAta System) Grid Translator Design-a-Grid.

Details:

- Must indicate latitude and longitude degree boundaries
- Three format choices all available with headers:
  1) Binary Raster
  2) ASCII Raster
  3) XYZ Delimited
- Downloads are free (no cost).

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA_NOS.png

Obtain estuarine bathymetry data from NOAA's NOS

The National Oceanic and Atmospheric Association (NOAA) has produced the National Ocean Service (NOS) Estuarine Bathymetry website to allow for US estuarine bathymetry access. Details:

- Estuaries divided into six regions:
  1) North Atlantic
  2) Middle Atlantic
  3) South Atlantic
  4) East Coast Gulf of Mexico
  5) West Coast Gulf of Mexico
  6) Pacific Coast
- 4 downloadable options of individual estuaries in *.dem format:
  1) 1 Arc Second (30 meter resolution) – Split into 7.5" quads
  2) 1 Arc Second – One big file
  3) 3 Arc Seconds (90 meter resolution) – Split into 1 degree quads
  4) 3 Arc Seconds – One big file
- Downloads are free (no cost).

Obtain topographic and bathymetric data from the Topographic and Bathymetric Data Inventory

The National Oceanic and Atmospheric Association (NOAA) offers topographic and bathymetric data from the NOAA coastal Services Center and the Federal emergency Management Agency through the Topographic and Bathymetric Data Inventory. This resource provides both topographic and bathymetric data for the coastal areas of the United States. More information on this resource can be found here .

DEM Overview

There are many uses and applications of DEMs. GIS (Geographic Information System) software such as ArcView and ARC/INFO, as well as modeling software such as WMS (Watershed Modeling System) can use DEMs for many engineering and scientific applications. WMS uses DEM data to produce watersheds which are then used to model storm events, create hydrographs, route floods down rivers and through reservoirs, etc. This information can be used to design culverts, dams, detention basins and other hydraulic structures. DEM data are commonly used to create another type of digital terrain model called a TIN (Triangulated Irregular Network). Public domain software is available for tasks such as simply viewing a DEM—one example is dlgv32 Pro , a tool provided by the USGS.

With powerful modeling software such as WMS, using DEM data to run a model is not difficult, and can be accomplished in four general steps:

1) Find and/or download the needed DEM data.
2) Import the DEM(s) into WMS

3) Delineate the watershed by inserting stream networks, one or more outlets, and reservoirs. If desired, a TIN can be created as well. Additional hydrologic data such as land use and soil type can also be used in WMS.
4) Run the model and view the results. WMS supports several models such as HEC-1, NFF, Rational, TR-55, TR-20, HEC-HMS, and GSSHA.

A DEM (Digital Elevation Model) is simply a digital map of elevation data. These maps, a type of DTM (Digital Terrain Model), are raster data meaning that they are made up of equally sized gridded cells each with a unique elevation.

DEMs come in different scales and resolutions. For example, 1:24,000 scale DEM is simply a USGS (United States Geological Survey) 7.5' quadrangle that has been digitized and each cell in the DEM represents a block of terrain 30 meters x 30 meters. The 1:250,000 scale DEM (also known as a 1-degree or a 3 arc-second DEM) has a resolution of 90 meters x 90 meters. DEMs with better resolution are available, but require large amounts of computer memory and disk space and are often impractical to use for large areas of land. If an individual DEM does not cover the entire area of interest, then multiple DEMs can be tiled together to make one large DEM.

The projection and datum for a DEM varies. A common projection for DEMs is UTM (Universal Transverse Mercator) coordinates (meters) and have a specific datum associated with them. Elevations are usually in meters, but sometimes are in feet for areas of low relief, and are referenced to mean sea level.

DEM Tips

- How do I obtain a DEM from USGS?
- How do I obtain a DEM from GIS Data Depot?
- How do I use the USGS Map Locator?
- How do I use the USGS GNIS (Geographic Names Information System)?
- How do I decompress data files?
- How do I import a DEM (native *.dem or SDTS format) into WMS?
- How do I import an NED Gridfloat file into WMS?
- How do I use WMS to convert data to a different coordinate system?

Bathymetry Tips

- How do I obtain bathymetry data from GEODAS Design-a-Grid?
- How do I obtain estuarine bathymetry data from NOAA's NOS?
- How do I import bathymetry data into SMS?
- How do I import estuarine bathymetry data into SMS?

GSDA:Hydrography Data

Stream Depths, Flow Rates, and Forecasts

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain stream stage data from the USGS WaterWatch

Water Watch offers real-time stage and discharge data for about 3,000 stream gages. Plus, it is linked to NWIS-Web, which furnishes surface and ground water data, as well as water quality data.

Advantages:

Easy-to-use graphical map allows accessing data for a gaging station in a couple of clicks. A list of all stations within a state can be quickly accessed.

- Help - Obtaining stream-stage data from WaterWatch

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain stream stage data from the USGS NWISWeb

NWISWeb offers an extensive amount of data, including surface and ground water data, as well as water quality data.

Advantages:

Users can search for a gaging station by its name, ID number, State, Drainage Basin, and more. Use this site if you need advanced search capabilities.

- Help - Obtaining stream-stage data from NWISWeb

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png

Obtain stream forecast data from the USDA/NRCS

The USDA/NRCS offers stream forecast data as well as current water supply maps.

- Help - Obtaining stream forecast data from the USDA/NRCS

Rivers, Lakes, and Seas

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_webGIS.png

Obtain hydrographic data from WebGIS

WebGIS provides data in shapefile and standard DLG formats

- Help - getting hydrographic data from WebGIS

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_ESRI.png

Obtain hydrographic data from ESRI Census 2000 TIGER

ESRI and the U.S. Census Bureau offer hydrographic data in shapefile format for ArcView or ARC/INFO

- Help - Obtaining hydrographic data from ESRI Census TIGER 2000

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain hydrographic data from the USGS/EPA

The USGS/EPA offers hydrographic data for each CU (cataloging unit) in the US in either ESRI/ARC or USGS/SDTS format.

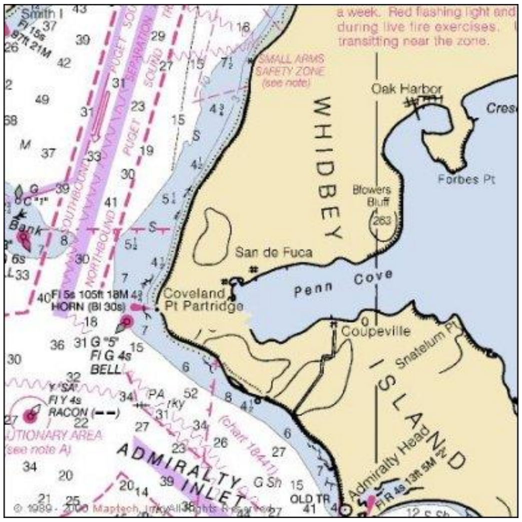
- Help - Obtaining hydrographic data from the USGS/EPA

Stream Data Overview

Stream stage simply refers to the water depth in a stream or river. The water depth is related to the flowrate and rating curves (plots of stage vs. flowrate) have been created for many streams and rivers. Available data sources can give real-time (i.e., current to the moment) stream-stage data whereas other sources give historical figures along with statistical data such as median flowrates etc. Some examples are given below for the Provo River in Utah. Another important flow data type are flow-duration curves (also called reliability curves) which give engineers an idea of what flowrates can be expected a certain percentage of the time. An annual hydrograph is required to create a flow-duration curve which is shown in the third figure. Streamflow data used for forecasting is also available for specific streams or for regions in general.

Hydrographic Waterbody Overview

By definition, hydrography is the study and survey of rivers, streams, creeks, springs, wells, ponds, lakes, reservoirs, oceans, seas, bays and estuaries with respect to their tides, flow characteristics, and navigability. Much of the available hydrographic information is in a format compatible with a GIS such as ArcView. Locating streams and other water bodies may be essential in creating a good model.



Stream Data Tips

- Help – Obtain stream stage data from USGS NWISWeb?
- Help – Obtain real-time stream stage data from USGS WaterWatch?
- Help – Obtain stream forecast data from the USDA/NRCS?

Hydrographic Waterbody Tips

- How do I obtain hydrographic data from WebGIS?
- How do I obtain hydrographic data from the USGS/EPA?
- How do I obtain hydrographic data from ESRI Census TIGER 1995?
- How do I decompress data files?
- How do I import hydrographic data into WMS?
- How do I import a standard DLG file into WMS?

GSDA:Imagery

DRG Image Data

Obtain images directly from WMS using the Get Online Maps tool

WMS offers an option to download various types of high-quality image data for anywhere in the world using the Get Online Maps tool. This is the recommended method to get image data into WMS unless there are other image sources that may be more recent or better quality.

USGS
science for a changing world

Obtain Data from USGS National Map Viewer Application

The national map viewer has several sources for downloading images, including GeoPDF's and high-resolution aerial photographs.

Obtain a DRG from MSR Maps

MSR Maps offers USGS Topos in several different resolutions. WMS uses this web service to download imagery for the United States.

Obtain a DRG from other public sites

Sometimes, the best way to locate GIS data from a public source, such as a state or county agency, is to consult an internet search engine. Go to a search engine website, such as www.google.com and enter the search criteria, such as "DRG Georgia."

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_GeoCommunity.png

Obtain a DRG from GIS Data Depot

- The USGS GNIS and Google Maps are useful tools for locating the correct DRG.
- Need additional help using USGS Map Locator?
- Need additional help using the USGS Geographic Names Information System (GNIS)?
- Need additional help obtaining a DRG from GIS Data Depot?

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_LandInfo.png

Obtain a DRG from Land Info

Land Info is a commercial site offering a wide variety of spatial data for purchase.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain a DRG from the USGS

You may purchase DRGs from the USGS.

DOQ Image Data

Obtain images directly from WMS using the Get Online Maps tool

WMS offers an option to download various types of high-quality image data for anywhere in the world using the Get Online Maps tool. This is the recommended method to get image data into WMS unless you have other image sources that may be more recent or better quality.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain a High Resolution DOQ from The National Map

This USGS site offers 1/3 meter resolution Orthos for various metropolitan areas.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_GeoCommunity.png

Obtain a DOQ from GIS Data Depot

- The USGS GNIS can be a useful tool to determine the *name* of the correct DOQ (GIS Data Depot also requires that you know the county).
- Additional help obtaining a DOQ from GIS Data Depot
- Additional help using the USGS GNIS (Geographic Names Information System)

Obtain a DOQ from other public sites

This list maintained by the USGS offers DOQ data for a selection of states.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_TerraServer.png

Obtain a DOQ from Terraserver

- Additional help obtaining a DOQ from Terraserver
- Additional help uncompressing data files

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain a DOQ from the USGS

You may purchase DOQs from the USGS.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_LandInfo.png

Obtain a DOQ from Land Info

Land Info is a commercial site offering a wide variety of spatial data for purchase.

Satellite Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain satellite images from the USGS SRTM

The SRTM offers 30 meter elevation data in GeoTIFF and ArcGRID formats

- Additional help obtaining satellite images from the USGS SRTM database

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_TerraServerUSA.png

Obtain satellite images from Terraserver

Among other image types, Terraserver offers OrbView and SPIN-2 satellite images for purchase.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_LandInfo.png

Obtain satellite images from Land Info

Land Info is a commercial site offering a wide variety of spatial data for purchase.

Nautical Charts

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png

- Obtain nautical charts from NOAA.
- Obtain historical nautical charts from NOAA.
- Obtain additional information from NOAA.

Image Overview

The use of maps and photographs within WMS/GIS serves many purposes. These images can be used as a backdrop to create a conceptual hydrologic model. In other words, the engineer can create feature objects (i.e., streams, outlets) on top of the map image to ensure that they are in the proper locations. There are three types of images generally used with WMS: TIFF, DRG, and DOQ. DRG images are the most commonly used.

In order for images to be useful they must be georeferenced. This means that the map image itself has been fit to actual coordinates on the earth's surface to some coordinate system such as UTM (Universal Transverse Mercator). If a map or image is not available for the area of interest, then any map or image can be scanned using a typical scanner. This image can then be georeferenced within WMS.

- Learn about TIFF images and DRGs.
- Learn about DOQ images.
- Learn about satellite images.

A common type of image format is a TIFF (tagged image file format) image. If a TIFF image is not already available from some source, they can be easily created by scanning a map using a typical scanner.

A special type of TIFF images that is already georeferenced is called a GEOTIFF, and does not need to be georeferenced with any additional steps since the georeferencing information is contained within the image itself. Some software will read GEOTIFF images and some won't. Other TIFF images come with a separate file called a TIFF World File which contains the georeferencing information. Finally, if a TIFF image is not georeferenced for some reason, it can be done in WMS or GIS through a process called registration (see WMS tutorial for more information).

DRGs are a special type of TIFF image. The USGS has scanned their standard series topographic maps and created georeferenced TIFF images from them - images like these are known as a DRG (Digital Raster Graphic). They are usually available in 1:24,000 (7.5' quads), 1:100,000, and 1:250,000 scale. The scale required for a project in WMS is very important since DRGs are not normally tiled together as are DEMs, but the proper size should be selected for the project (i.e., a very large watershed may require a 1:250,000 scale DRG). DRG images are often large with respect to disk space, often exceeding 10 megabytes each. Most DRGs are GEOTIFFs.

WMS and GIS will both read a DRG, and most graphics programs will at least display one on the computer screen. The USGS offers free software that will also read a DRG image (visit http://mcmcweb.er.usgs.gov/viewers/dlg_view.html for more information).

DOQ images are high-resolution (often 1 square meter per pixel) computer generated images of aerial photographs shot at about 20,000 ft altitude. They are usually very large in file size (often exceeding 40 megabytes for black & white) so typically 4 DOQ images are taken for each 7.5' quadrangle as a NW, NE, SW, and SE quarter-quad. The "ortho" refers to the fact that the image displacements caused by camera tilt and terrain topography have been removed from the aerial photo (this process is called orthorectification). This means that the image is a photographic map which can be used to accurately measure distances directly from the photograph and other cartographic (map) information can be directly overlaid onto the DOQ. DOQ images are sometimes georeferenced, and sometimes they are not. DOQs come in various file formats. Some images are in .jpg format and have a file associated with them similar to a TIFF world file (as is the case with DRGs). Another common format is .bil which stands for "band interleaved by line" multiband image. ArcView and ARC/INFO require the JPEG extension to view .jpg images but will automatically read the .bil format as an image data source (versus a feature data source such as a shapefile). WMS cannot read .bil format directly. DOQ images should be available for the conterminous United States by 2004, and then will be updated every 5 or 10 years depending on how rapid the land use change is in a certain area.

- FAQ about DOQs.
- Learn more about DOQ images at the USGS.

Other image types such as SPIN-2 and OrbView satellite photographs can be found, viewed, and purchased from several sources. As mentioned in the overview, satellite images are sometimes clearer and more resolute than aerial photographs.

Image Tips

- How do I obtain image data from Terraserver?
- How do I obtain a DRG from GIS Data Depot?

- How do I obtain a DOQ from USGS?
- How do I obtain a DOQ from GIS Data Depot?
- How do I obtain a DOQ from Terraserver?
- How do I obtain satellite images from USGS?
- How do I use the USGS Map Locator?
- How do I use the USGS GNIS (Geographic Names Information System)?
- How do I decompress data files?
- How do I register an image for use in WMS?
- How do I import a DRG into WMS?
- How do I use WMS to convert DRG data into a different coordinate system?

GSDA:Meteorologic Data

Maps, Tables, and Charts of Snow and Rainfall

Obtain NEXRAD Radar Data from NCDC

- Help – Obtaining NEXRAD Radar Data from NCDC

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA_NESDIS.png

Obtain historical precipitation data from the NNDC Online site

Fifteen minute and hourly historical data in ascii format at no cost.

- Help – Obtaining precipitation data from the NNDC

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png

Obtain digital rainfall grids from NOAA Atlas 2

Precipitation-Frequency maps for the Western US at a 15 ' resolution (~400m)

- Grid in Geographic NAD 83 coordinates
- Grid values in inches * 100,000
- Help – Obtaining a digital rainfall grid from NOAA Atlas 2

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png

Obtain precipitation data from NOAA

A massive collection of climate-related data, tables and maps for gages, states and more.

- Help – Obtaining precipitation data from NOAA

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA_RCC.png

Obtain precipitation data from the RCC

A variety of precipitation data from NOAA Atlas 2 precipitation/duration/frequency maps to snow data.

- Help – Obtaining precipitation data from the RCC

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_UCC.png

Obtain precipitation data from the Utah Climate Center

Gage data (i.e., precipitation, snowfall, temperature) for all 50 US states and around the world.

- Help – Obtaining precipitation data from the UCC

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png

Obtain precipitation data from NRCS PRISM

Thirty-year average (normal) precipitation maps for each state as well as GIS precipitation data.

- Help – Obtaining precipitation data from NRCS PRISM http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png

Obtain precipitation data from NRCS NWCC

A wide variety of precipitation and snow data in graph, chart and table format.

- Help – Obtaining precipitation data from the NRCS

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_OneRain.png

Obtain precipitation data from OneRain

Radar and rain gage data combined to create accurate rainfall estimates. This data must be purchased.

Precipitation Overview

Climatological data such as precipitation, evapo-transpiration, wind and temperature are usually found in university libraries and are published by the U.S. Department of Commerce National Oceanic and Atmospheric Administration (NOAA). Precipitation data includes depth, intensity, duration, recurrence intervals, snowfall etc. Methods of calculating precipitation for a watershed from gage data include the Thiessen polygon method, the isohyetal method, and a simple arithmetic mean. Other forms of precipitation such as snow are also important. Snow data can be collected by various means such as SNOTEL and snow course.

Precipitation Tips

- How do I obtain precipitation data from NNDC Climate Data Online
- How do I obtain precipitation data from NOAA
- How do I obtain precipitation data from RCC
- How do I obtain precipitation data from the UCC
- How do I obtain precipitation data from NRCS PRISM
- How do I obtain precipitation data from NRCS NWCC
- How do I decompress data files

GSDA:Oceanic Data

Coastline Data

Obtain coastline data from NOAA NGDC

The National Oceanic and Atmospheric Association (NOAA) has produced the National Geophysical Data Center (NGDC) website to allow for US data access. It is possible to download coastline data from an interactive global map.

- Need help obtaining coastline data from NOAA NGDC, click here .

Tidal Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png

NOAA's National Ocean Service (NOS) [201] is home to the Center for Operational Oceanographic Products and Services (CO-OPS). This center gathers oceanographic data along the coast of the United States. Data available from this site includes historical and real-time observations and predictions of water levels and currents.

Obtain tidal data from NOAA's NOS

Obtain historical tidal data from NOAA's NOS

Obtain tidal precictions from NOAA's NOS

Current Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png

NOAA's National Ocean Service (NOS) is home to the Center for Operational Oceanographic Products and Services (CO-OPS). This center gathers oceanographic data along the coast of the United States. Data available from this site includes historical and real-time observations and predictions of water levels and currents.

Obtain current data from NOAA's NOS

Help with NOAA's NOS

About NOAA's NOS

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_DNR.png

The Texas Coastal Ocean Observation Network (TCOON) monitors several buoys off the coast of Texas. Real-time and historical data can be obtained from this sight. A variety of data types are available for each buoy. More help and information can be found on the web page.

Obtain current data from TCOON

Wave Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_ERDC.png

The Wave Information Studies (WIS) is part of the U.S. Army Corps of Engineers Coastal and Hydraulics Laboratory. Bulk wave parameters (significant wave height, period, direction) and wind speed and direction can be downloaded and viewed on the WIS site.

Obtain wave data from WIS

Help with WIS

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png

NOAA's National Data Buoy Center (NDBC) supplies data for moored buoys. The buoys measure a variety of data including: barometric pressure; wind direction, speed, and gust; air and sea temperature; wave energy spectra; and direction of wave propagation. Significant wave height, dominant wave period, and average wave period are derived from the wave energy spectra. The data available depends on the buoy type and data provider.

Obtain wave data from NDBC

Help with NOAA's NDBC

About NOAA's NDBC

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_CDIP.png

The Coastal Data Information Program (CDIP) measures coastal environment data. A variety of data types and formats are available on their website.

Obtain wave data from CDIP

Help with CDIP

About CDIP

Oceanic Data Overview

Oceanic Tips

Coastline Tips

- How do I obtain coastline data from NOAA NGDC?
- How do I import coastline data into SMS?

Tidal Tips

- Help with NOAA's NOS
- About NOAA's NOS

GSDA:Surface Characteristics

Land Use

Obtain a land use grid directly from WMS using the Get Data tool

WMS offers an option to download NLCD land use data for anywhere in the United States and CORINE data for anywhere in Europe using the Get Data or the Get Data from Map tools. This is the recommended method to get land use data into WMS unless you have other land use sources that may be more recent or better quality.

Obtain Data from USGS National Map Viewer Application

The national map viewer has several land use data sources (impervious areas, land use types, vegetation types, etc.) from various dates. All land use sources are in GeoTIFF format and can be read into WMS as raster GIS data and then converted to a land use grid. The land use grid can the be used to compute composite runoff parameters such as Curve Number and to assign parameters to 2D grids for distributed hydrology.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_webGIS.png

Land use data from WebGIS

WebGIS offers land use/cover shapefiles in Geographic and UTM coordinates at no cost.

- Additional help obtaining land use data from WebGIS
- View a text file containing the Land Use / Land Cover codes and their definitions
- Help with creating a land use table for Curve Number computation within WMS

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_EPA.png

Obtain the HUC number for your watershed

The EPA provides the "Locate Your Watershed" site to help users determine their region's HUC.

Obtain the land use data from the EPA HUC index

The EPA offers land use shapefiles.

- Additional help obtaining land use data from the EPA
- BASINS Metadata website

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png

Obtain land use data from the USGS

The USGS offers land use data in GIRAS format for ARC/INFO.

- Additional help obtaining land use data from the USGS?
- More information about USGS land use data.

Soil Type Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_EPA.png

Obtain the HUC number for your watershed

Obtain land use and soil type data from the EPA HUC index

The EPA offers land use and soil type shapefiles.

- Additional help obtaining soil type data from the EPA?
- BASINS Metadata website.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png

Obtain STATSGO soil type data from the NRCS

The United States Department of Agriculture (USDA) supports the Natural Resources Conservation Service (NRCS). A website they have developed is the National Cartography and Geospatial Center (NCGC). Supplied on this website are numerous links and descriptions of geospatial datasets. Through these links it is possible to acquire soil type data maps from the two main soils databases; STATSGO, and SSURGO.

State Soil Geographic (STATSGO) Database

Details:

- Database product files in three formats:
  1) USGS Digital Line Graphs (DLG-3)
  2) ArcInfo 7.0 Coverage
  3) GRASS 4.13 Vector
- Coordinate systems are:
  4) Albers Conical Equal - Area: Continental U.S. and Alaska

5) UTM Zone 4: Hawaii
6) UTM Zone 19: Puerto Rico

- Resolution at 1:250K scale, except Alaska (1:1M).
- Complete coverage of U.S. and includes Puerto Rico.
- Data files are downloadable in zip file compression.
- Downloads are free (no cost).

- Additional help obtaining soil type data (STATSGO format) from the NRCS

It is possible to obtain SSURGO data from the NRCS by two methods. The first is the Web Soil Survey. The other is through the Geospatial Data Gateway. Both are accessible on the NCGC website.

Obtain SSURGO soil type data from Web Soil Survey

Web Soil Survey is a data request site supported by the United Sates Department of Agriculture (USDA) through the Natural Resources Conservation Service (NRCS). From Web Soil Survey you can place orders requesting certain soil survey maps. It specifically caters to only soil data needs.

Details:

- Database products available in three forms:
  1) Tabular
  2) Spatial
  3) Template Database
- Spatial products downloads provided in the following format:
  1) ESRI Shapefile
- Data are provided in the following coordinate system:
  1) Geographic WGS84
- Typical scale resolution between 1:16K and 1:64K.
- Soil downloads can be confined to a user defined area of interest
- Files can be downloaded only one survey at a time.
- Data files are acquired in zip file compression.
- Downloads are free (no cost).

- Additional help obtaining soil type data (SSURGO format) from Web Soil Survey

Obtain SSURGO soil type data from the Geospatial Data Gateway

The Geospatial Data Gateway is a site sponsored by the United States Department of Agriculture (USDA) through the Natural Resources Conservation Service (NRCS). Numerous databases are available by means of the Geospatial Data Gateway including; transportation, census, land use / land cover, soil type, ortho imagery, elevation, and topographic maps. Description below applies to SSURGO soil type data only.

Details:

- Database product files in three formats:
  1) ESRI Shape File
  2) ESRI Coverage
  3) ESRI ASCII Export
- Coordinate systems are available in a variety of projections:
  1) Geographic
  2) Universal Transverse Mercator (UTM)
  3) State Plane
- Typical scale resolution between 1:16K and 1:64K.

  - Extensive coverage of U.S.
  - Data files are downloadable in zip file compression.
  - Downloads are free (no cost).
- Additional help obtaining soil type data (SSURGO format) from the Geospatial Data Gateway

Land Use Overview

Of the many methods to estimate infiltration, the NRCS Curve Number (CN) method is one that is commonly used. For watersheds with multiple soil types and land uses, a composite CN (CCN) must be calculated to estimate the infiltration losses. The USDA/NRCS supplies tables so that a CCN can be determined from soil type, land use, moisture condition, and hydrologic condition. In hydrology, land use (also known as cover crop or land cover) refers to the way in which the land is being used and/or its condition. Land uses can be urban or agricultural/rural. Examples of these include streets, industrial areas, commercial areas, row crops, meadows, pasture/range and woods.

Soil Type Overview

For watersheds with multiple soil types and land uses, a composite CN (CCN) must be calculated to estimate the infiltration losses. The NRCS (part of the USDA) supplies tables so that a CCN can be determined from soil type, land use, moisture condition, and hydrologic condition. In hydrology, soil type can be classified in many ways. The USDA classifies them according to their infiltration rate and are referred to as either A, B, C, or D soils. Soil type A has a high infiltration rate whereas soil type D usually consists of clays that are nearly impermeable (low infiltration) and produce higher volumes of runoff.

Land Use Tips

- How do I obtain land use data from the EPA?
- How do I obtain land use data from the USGS?
- How do I format land use data (coordinate system, clipping) for use in WMS?
- How do I decompress data files?
- How do I import EPA land use data into WMS?
- How do I create a land use table for composite CN calculations?

Soil Type Tips

- How do I obtain soil type data from the EPA?
- How do I obtain STATSGO soil type data from the NRCS?
- How do I obtain SSURGO soil type data from Soil Data Mart?
- How do I obtain SSURGO soil type data from the Geospatial Data Gateway?
- How do I format soil type data (coordinate system, soil attribute, clipping) for use in WMS?
- How do I decompress data files?
- How do I import EPA soil type data into WMS?
- How do I create a land use table for composite CN calculations?

- How do I create a soil type runoff coefficient table for composite RC calculations?
- How do I obtain soil type data from the NRCS - STATSGO?
- How do I obtain soil type data from the NRCS - SSURGO?
- How do I use soil type data with GSSHA?

7.3. Archives

Archive Features

The follow features are obsolete in the current release version of SMS.

Functionalities

- ADH Velocity Series Editor
- Create Datasets
- CMS-Flow Transport Control
- CMS-Flow Tidal Constituents
- CMS-Wave Nesting Options Dialog
- CMS-Wave Spectral Coverage
- Size Dataset Command

Modules

1. 1D River Module

1D River Module

1D River Hydraulics Data Browser

1D River Hydraulics Profile Plots

1D River Module Menus

Interpolate Cross Sections

2. 3D Cartesian Grid Module

3D Cartesian Grid Module

3D Cartesian Grid Display Options

Related Topics

- Archived Models
- Tutorial Archives

Archived Models

The following models are either obsolete or have not yet been released to the public.

1D River Conceptual

- 1D River Conceptual Model

BASEMENT

- BASEMENT

Cascade

- Cascade

CSHORE

- CSHORE

CSTORM

- CSTORM-MS

Dredge Source Model

- Dredge Source Model
- Dredging Material Properties_Stickiness
- Dredging Scheduling
- Dredging Sediment Characteristics
- Dredging Simulations
- Importing Dredge Tracks from ASCII Files
- Dredging Coverage \

EFDC

- EFDC Coverage

ELCIRC

- ELCIRC

FATE

- FATE
- FATE Menu
- FATE Model Control Current

Generic Grid Model

- Generic Grid Model

GENESIS

- GENESIS
- GENESIS Arc Attributes Dialog
- GENESIS Graphical Interface
- GENESIS Model Control Dialog
- GENESIS Observation Stations
- Saving GENESIS

- GENESIS Structures
- GENESIS Menu

HEC-RAS

- HEC-RAS
- HEC-RAS Material Properties
- Q&A HEC-RAS

Holland\PBL

- Holland\PBL
- Hurricane Path Pertubations Dialog
- Holland Symmetrical/Asymmetrical
- PBL

LTFATE

- LTFATE
- LTFATE Menu
- LTFate Coverage

M2D

- M2D

RiverFlow2D

- RiverFlow2D

STFate

- STFate
- STFate Cloud File
- STFATE Clouds to PTM Sources
- STFate Compliance Reports