GMS User Manual v9.2, vol. 1

Introduction, Set Up, and General Tools

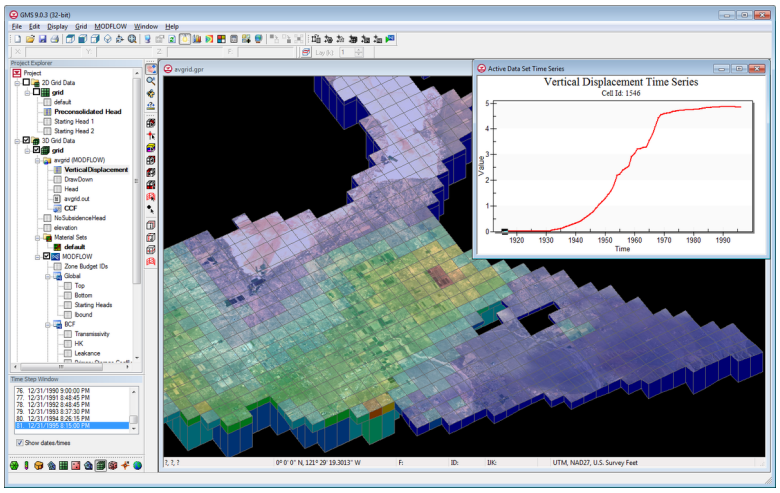
Contents

Articles

1. Learning GMS

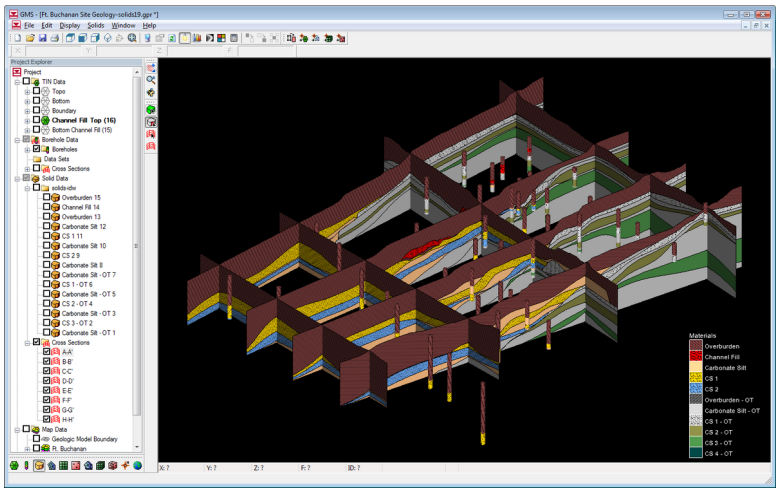
What is GMS?

The Groundwater Modeling System (GMS) is a comprehensive graphical user environment for performing groundwater simulations. The entire GMS system consists of a graphical user interface (the GMS program) and a number of analysis codes (MODFLOW, MT3DMS, etc.). The GMS interface is developed by Aquaveo, LLC in Provo, Utah.



SUB package vertical displacement plot.

GMS was designed as a comprehensive modeling environment. Several types of models are supported and facilities are provided to share information between different models and data types. Tools are provided for site characterization, model conceptualization, mesh and grid generation, geostatistics, and post-processing.



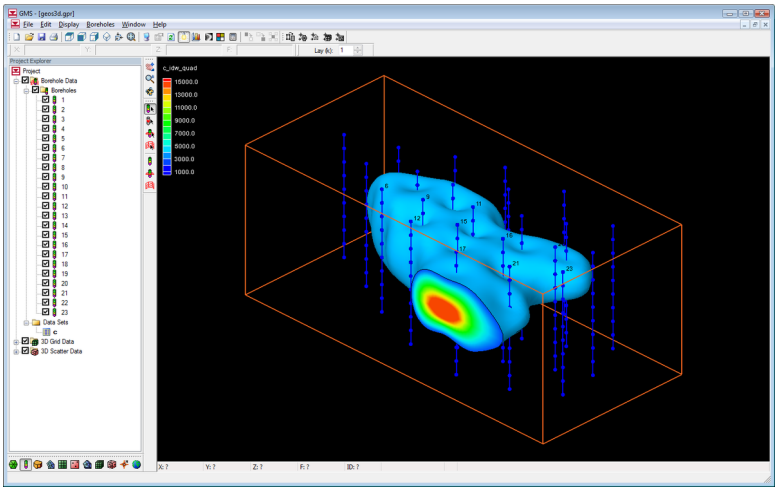
Solid cross sections and boreholes.

Modules

The interface for GMS is divided into eleven modules. A module is provided for each of the basic data types supported by GMS. As you switch from one module to another module, the **Dynamic Tool Palette** and the **Menus** change. This allows you to focus only on the tools and commands related to the data type you wish to use in the modeling process. Switching from one module to another can be done instantaneously to facilitate the simultaneous use of several data types when necessary. The following modules are supported in GMS:

TIN Module

- Borehole Module
- Solid Module
- 2D Mesh Module
- 2D Grid Module
- 2D Scatter Point Module
- 3D Mesh Module
- 3D Grid Module
- 3D Scatter Point Module
- Map Module



Isosurface on a 3D grid and borehole sample data.

- GIS Module

Numerical Models

Numerical models are programs that are separate from GMS that are used to run an analysis on a model. The models can be built in GMS, and then run through the numerical model program. GMS can then read in and display the results of the analysis.

With most numerical models, GMS has the option of using a Model Wrapper to run the model and display real-time results during the model simulation.

The following numerical models are currently supported in GMS.

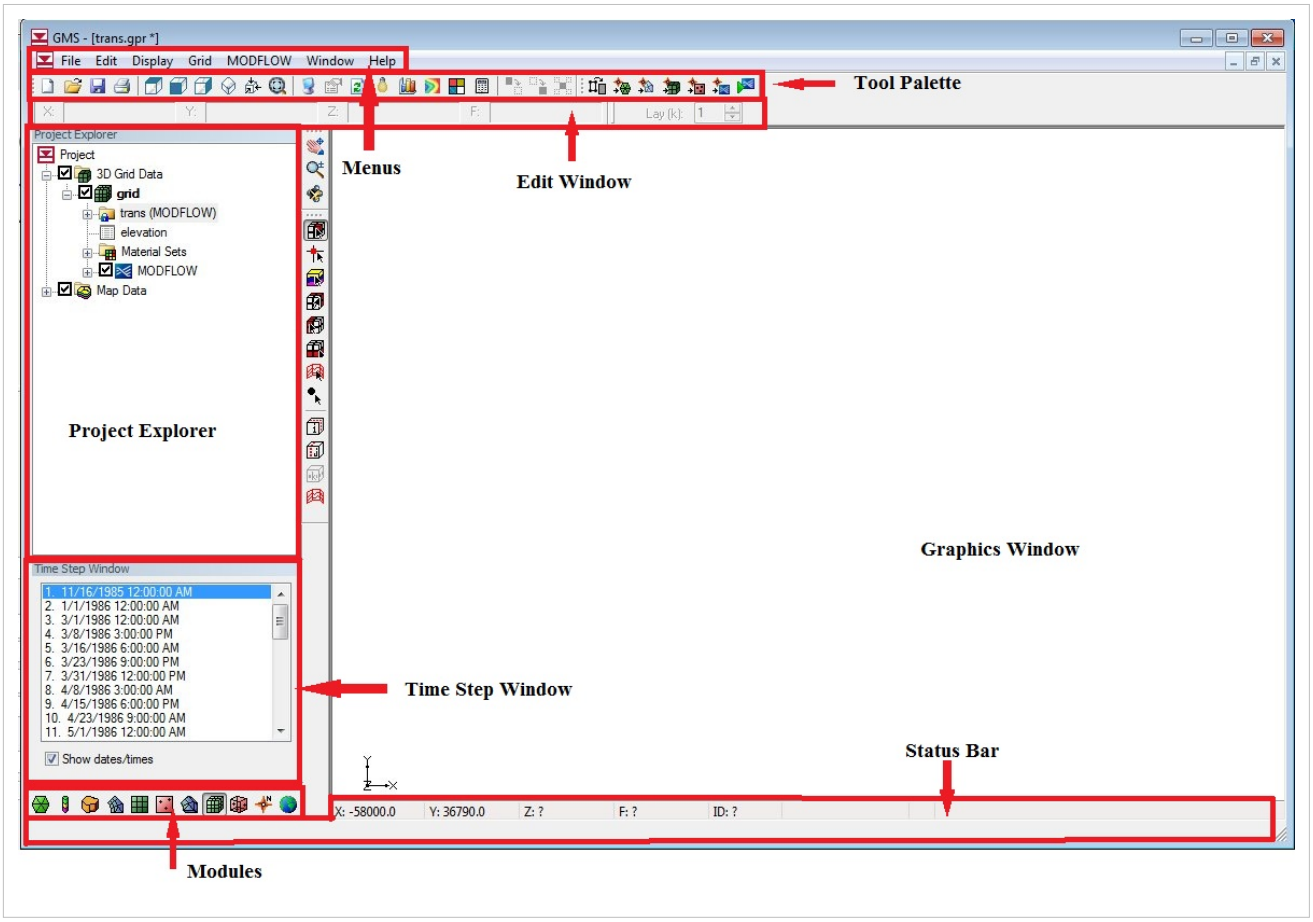
Model Name	GMS Module
• MODFLOW • MODPATH • MT3DMS • RT3D • ART3D • SEAM3D • UTCHEM • PEST • SEAWAT (starting v8.0)	3D Grid
• MODAEM • UTEXAS	Map Module
• SEEP2D	2D Mesh
• FEMWATER	3D Mesh
• T-PROGS	Boreholes

Source code for most models is available upon request. Contact technical support [1] to request source code.

References

[1] http://www.aquaveo.com/technical-support/

The GMS Screen



The GMS screen is divided into seven main sections:

Menu Bar

Each module has its own set of menus. The first three menus (File, Edit, Display) are the same for every module. The remaining menus are dependent on the selected module.

Edit Window

These fields are used to edit the coordinates of selected items (vertices, nodes, scatter points, etc.). The coordinates are changed by typing in new values and hitting the *ENTER* or *TAB* key. The scalar data value (F:) associated with the selected object can also be edited.

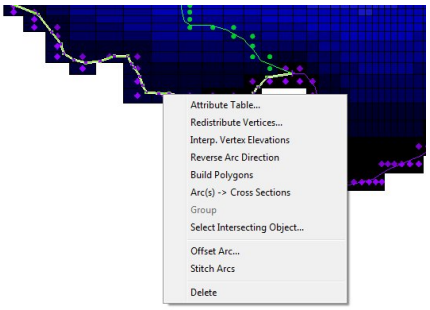
Tool Palette

Several Tool Palettes can be displayed in the GMS interface. Default tool palettes include the *Macros Palette*, the *Static Tool Palette*, the *Dynamic Tool Palette*, the *Mini-Grid Plot Palette*, and the *Module Palette*. (For more information on each of these palettes see Tool Palettes)

Graphics Window

The primary graphical input and output for GMS takes place in the *Graphics Window*. The action taken when you interact with the *Graphics Window* depends on which tool is selected.

Every object that can be selected in the *Graphics Window* now has a corresponding right-click menu available. These menus contain commands that allow the user to manipulate the data or display of the selected items.



Example of a right-click menu on a arc in the map module.

Project Explorer

The *Project Explorer* is located at the left side of the GMS window by default, but it can be moved to anywhere on the window since it is a dockable toolbar. The [*Project Explorer* contains a hierarchical representation of the data associated with a modeling project. The *Project Explorer* can also be resized simply by clicking on the window borders and dragging them to a new location.

Many commands on the data in GMS can be executed by right-clicks in the *Project Explorer*. For general commands or to create new data objects you can right-click on the empty space in the *Project Explorer* and the following pop-up menu is available. The visibility of items in the *Graphics Window* can also be controlled by selecting the toggle next to each item in the *Project Explorer*.

Project Explorer View

Time Step Window

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* only appears when a transient dataset is selected in the Project Explorer. Time steps can be viewed in date/time format or relative time format by selecting the toggle below the *Time Step Window*. The *Time Step Window* can be resized simply by clicking on the window borders and dragging them to a new location.

Status Bar

The *Status Bar* is used to display the cursor coordinates, the IJK indices of the cell beneath the cursor, the dataset value beneath the cursor, data associated to selected items, and other information.

Tool Palettes

There are several Tool Palettes that can be displayed in the GMS interface. Below are the tool palettes that are on by default.

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.

Static Tool Palette

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

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

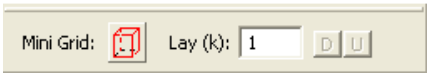
	Measure	The **Measure** tool provides a quick way to measure distances. • The tool is only available in plan view. • Measuring is done by selecting the tool and clicking on the Graphics Window with the mouse. A single line or a polyline can be created with a double-click used to end the line. The backspace key removes the last point clicked. • The instantaneous and total lengths are given at the bottom of the GMS window. The length units correspond to those selected in the Units dialog. A units conversion takes place if the current horizontal Projection units are different than those in the Units dialog.

Dynamic Tool Palette

When the active module is changed by selecting an object in the Project Explorer, 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.

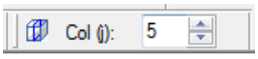
Mini-Grid Plot

The Mini-Grid Plot appears when a 3D grid is in memory and is activated when the orthogonal viewing mode is active. In the orthogonal mode, the viewing angle is always parallel to one of the three grid axes (I, J, or K) and only one of the rows, columns, or layers is displayed at one time. The Mini-Grid Plot shows an idealized representation of the 3D grid and shows which of the rows, columns, or layers is currently being displayed. The current row, column, or layer can be changed using the arrows just below the Mini-Grid Plot.



Mini-Grid Toolbar

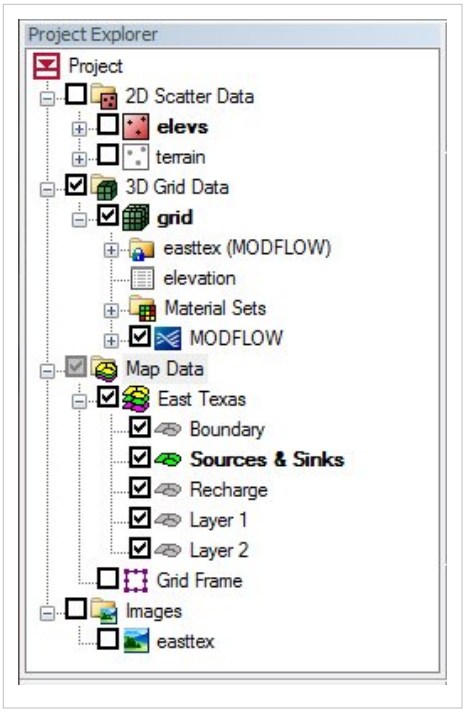
The Mini-Grid Toolbar appears when a 3D grid is in memory and is activated when the orthogonal viewing mode is active. In the orthogonal mode, the viewing angle is always parallel to one of the three grid axes (I, J, or K) and only one of the rows, columns, or layers is displayed at one time. The Mini-Grid Toolbar shows an idealized representation of the 3D grid and shows which of the rows, columns, or layers is currently being displayed. The current row, column, or layer can be changed using the arrows just below the Mini-Grid Toolbar.



Module Palette

The Module Palette is used to switch between modules. Only one module is active at any given time. However, the data associated with a module (ex. a 3D finite element mesh) is preserved when the user switches to a different module. Activating a module simply changes the set of available tools and menu commands.

Project Explorer



The Project Explorer is located at the left side of the GMS window by default. It can be moved to anywhere on the window since it is a "dockable" toolbar. The Project Explorer contains a hierarchical representation of the data associated with a modeling project. Previously only the data form the active module was displayed in the Project Explorer, but now all of the data in GMS is always displayed independent of the active module. The new Project Explorer can also be resized simply by clicking on the window borders and dragging them to a new location.

All of the modules have root items coresponding to them. Each root item may be right clicked on to accomplish certain actions. Data for each module is grouped into folders. The modules and items can also be expanded and colapsed to show sub folders. Each item may also be turned on or off by clicking the check box next to the item. Many items in the project explorer can be dragged to different locations to be used with different modules. Items may also be duplicated.

Many commands on the data in GMS can be executed by right-clicks in the Project Explorer. For general commands or to create new data objects you can right-click on the empty space in the Project Explorer and the following pop-up menu is available. Right-click commands can also be used to export or transform items. The visibility of items in the Graphics Window can also be controlled by selecting the toggle next to each item in the Project Explorer.

Tutorials

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

The tutorials are in PDF format. They are installed in the "docs" or "tutfiles" directory in the folder where GMS is installed.

The tutorials are listed below by subject and are **not** necessarily listed in the suggested order of completion. Some tutorials assume a basic knowledge of GMS and some build on other tutorials. When this is the case the tutorial itself will state which tutorials you should complete beforehand.

Many of the tutorials have files that are needed to run the tutorial. These files can be found in the "tutfiles" folder where GMS was installed.

[1].

Previous Versions

For previous versions of the GMS tutorials, click on one of the following links. For the current version, see the tables below.

- GMS 6.5 Tutorials
- GMS 7.0 Tutorials
- GMS 7.1 Tutorials
- GMS 8.0 Tutorials
- GMS:Tutorial Tables 8.1 [2]
- GMS:Tutorial Tables 8.2 [3]
- GMS:Tutorial Tables 8.3 [4]

References

[1] http://www.aquaveo.com/gms-learning
[2] http://www.aquaveo.com/gms-81tutorials
[3] http://www.aquaveo.com/gms-82tutorials
[4] http://www.aquaveo.com/gms-83tutorials

2. Set Up

64 bit

Starting at GMS 8.1, GMS is available in a 64 bit version. This means:

- GMS can access more RAM so larger models can be created
- A 64 bit version of MODFLOW is included and can be used by changing an option in the Preferences dialog
- ArcObjects is not available because ESRI [1] has not created a 64 bit version

References

[1] http://www.esri.com

License Agreement

AQUAVEO SOFTWARE LICENSE AGREEMENT

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

License agreements for software used by Aquaveo software

FTGL LICENSE AGREEMENT

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

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

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

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

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

WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Registering GMS

When you first install GMS, it will be running in Demo Mode. All GMS functions will be enabled with the exception of printing and saving. Anyone can run GMS in Demo Mode on any computer and it can be freely distributed. To enable the print and save functions, you need either a password or a hardware lock.

The components (modules, interfaces) can be licensed individually depending on the needs and interests of the user. The components of GMS are licensed using a password system. The ***Register*** command is used to enter a password that enables the licensed components. This command can be used to enable the program after initially installing GMS, or for adding additional modules to the program at a later time. The ***Register*** command must be used before any files can be saved or printed. Before registration, GMS will run in Demo Mode.

When the ***Register*** command is selected, the *Register* dialog appears. The first item shown in the dialog is the security string. This string is keyed to the hard drive of the computer where GMS is installed and uniquely identifies the computer. When first registering GMS, this security string should be reported to the distributor or reseller where GMS was purchased. The reseller then provides a password which should be entered in the edit field at the top of the dialog. Once the password is entered, the **Register** button is selected. If the security string was reported correctly and the password was entered correctly, the text next to each of the licensed components changes from "DISABLED" to "ENABLED".

The **Details** button brings up a dialog listing the phonetic code for the security string. When reporting the security string over the phone to get a password, using the phonetic code can be helpful in avoiding errors.

Once GMS has been registered, a file called gmspass.txt is created. Since GMS is licensed on a per/seat basis, arrangement must be made to get an additional password if GMS is to be moved to another computer.

Also, GMS can also be enabled using a hardware lock, rather that a password. Contact your GMS reseller for details.

Password

From the *File* menu, select the ***Register*** command. This brings up the Register dialog, which has a "security string" listed at the top. Send this security string along with your name, company, phone number, and e-mail address to your vendor.

After verifying that you are a licensed user, a password will be sent to you. When you receive the password, enter it in the **Password** field in the *Register* dialog and click the **Register** button. The modules you purchased will become enabled and GMS will run in Normal mode.

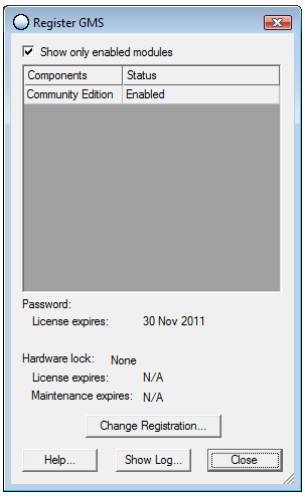
Hardware Lock

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

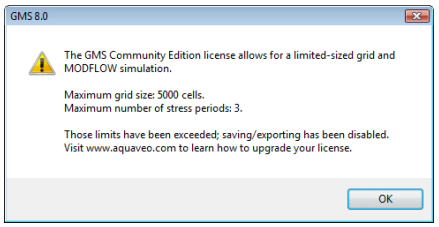
Community Edition

Starting at version 8.0 there is a free version of GMS called "Community Edition". It is limited to include only the 3D grid module and the MODFLOW model interface. It is also restricted in the size of the grid and the number of MODFLOW stress periods. Any size model can be imported, but if the grid exceeds 5000 cells or the number of stress periods is more than 3 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).

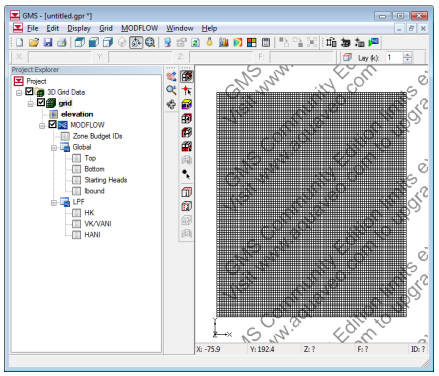
You can check if you are running in Community Edition mode by going to the Registration dialog. The size limits are displayed in the Help|About dialog.



GMS registration dialog showing that GMS is running in Community Edition mode.



GMS Community Edition warning dialog.



GMS showing watermark in Community Edition when size limits are exceeded.

Graphics Card Troubleshooting

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

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

Remote Desktop

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

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

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

Parallels Desktop for Mac

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

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

OpenGL Graphics Dialogs

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

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

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

Determining Graphics Card Manufacturer

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

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

You can also:

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

Updating Laptop Graphics Card Drivers

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

Updating Desktop Graphics Card Drivers

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

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

Updating Windows Operating System

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

Updating XMS Software

Many problems are resolved by installing the latest version of XMS. Bugfixes and updates are released frequently. The updates can be downloaded at the Aquaveo Download Center [19].

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 [20] 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 [1].

External Links

- Aquaveo Technical Support [1]

References

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

Reporting Bugs

The *Help | Report A Bug* menu command can be used to send a bug report to GMS tech support. Reporting bugs helps to improve the quality of GMS. An internet connection is required to use this feature.

3. General Tools

3.1. The File Menu

File Menu

The *File* menu is one of the standard menus and is available in all of the modules. The commands in the *File* menu are used for file input and output for the basic GMS file types, for printing, and to exit the program. The following commands are contained in the *File* menu:

- **New**

  Deletes all data associated with all data types and all modules. It resets the status of the program to the default state that is set when the program is first launched. This command should be selected when an entirely new modeling problem is started.

- **Open**

  This command is most often used to read in project files or other GMS native files. However, the **Open** command is also used to import data or other files generated outside of GMS. Projects can either be opened into the current window which replaces the existing project, or the project can be opened into a new instance of GMS. This setting is selected in the *Preferences* dialog that is accessed by the *Edit* | **Preferences** command.

- **Import From Database**

  Allows users to access data stored in a database and import it.

- **Import From Web**

  Allows users to import an image from the world-wide-web.

- **Save**

  Used to save GMS projects. A project contains all of the files associated with a modeling project. When a GMS project is saved, all files associated with the data currently in memory are saved. This includes any model simulations which are open. By default the model simulation will be saved to the same location as the project. However, in the *Save* dialog the path for the model simulation can be specified.

- **Save As**

  Used to designate the path for saving a GMS project. It can also be used to Export data.

- **Edit File**

  Prompts for the name of a file and opens the file in a program of your choice. This command is used to edit model input files or to view output files. Output files that are part of a Solution can also be viewed by double-clicking on the text file in the Project Explorer.

- **Page Setup**

  Launches the *Page Setup* dialog. The *Page Setup* dialog contains 3 tabs: Margins, Options, and Paper Size. The *Options* tab allows the user to specify the printing scale. The *Paper Size* tab allows the user to select the paper size and source. Also, the orientation, portrait or landscape, can be selected. The *Margin* tab allows the user to change the Margins. On the right side of each tab is a print preview.

- **Print**

  Printed copies of the current GMS image are generated with this command. This launches the standard Windows *Printing* dialog.

  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. To return to normal operating mode, select the **Normal Mode** command. (Also See Registering GMS)

- **Register**

  Brings up the *Register* dialog which allows users to register the software.

- **Recent File List**

  Near the bottom of the file menu is a list of recently opened projects. You can have as many as 5 different files in the recent file list.

- **Exit**

  Terminates the program.

3.2. The Edit Menu

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. The *Edit* menu contains the following commands:

- **Delete**

  Delete the object currently selected in the Graphics Window.
- **Select All**

  Selects all items associated with the current selection tool.
- **Unselect All**

  Unselects all items associated with the current selection tool.
- **Invert Selection**

  Selects the items that were not initially selected and are associated with the current selection tool.
- **Select With Poly**

  Used to enter a polygon enclosing the items to be selected (one of the selection tools must be active). This option is useful when selecting a large irregularly shaped group of objects. To enter the polygon, click on the polygon's starting point and each intermediate point defining the polygon and double-click on the ending point. All items within the polygon will be selected.
- **Select With Polyline**

  Selects the items that intersect a polyline associated with the current selection tool.
- **Select From List**

  Some of the objects in GMS (TINs, solids, cross sections, and scatter point sets) are selected by selecting an icon that appears on the object when the selection tool for the object is active. With a large number of objects, the display of the icons can become complicated and it may be difficult to select the desired object. In such cases, this command can be used as an alternative method for selecting such objects. It brings up a list of the objects currently in memory. An object is selected by checking the box next to it in the list and then hitting the **OK** button.
- **Locate Selections**

  Causes a rectangle to be drawn on screen and move to surround the location of the selected items, helping to see where they are.
- **Selection Window**

  When objects are selected, various information about the objects can be displayed or saved. The values are displayed by default in the Status Bar at the bottom of the GMS main window. However, since space along the bottom of the window is limited, the user has the option of displaying the information to a separate window and echoing the information into a file. The Selection Information dialog allows you to turn on and off the echo of the information to a file and to a separate, dockable window.
- **Dataset Calculator**

  The Data Calculator can be used to perform mathematical operations with datasets to create new datasets.
- **Properties**

This command brings up the *Properties* dialog for the currently selected item. This dialog varies depending on the selected item.

- **Materials**

  Brings up the *Material Properties* dialog.
- **Model Interfaces**

  Bring up a dialog containing all of the models, allowing the user to define which model menus are to be static in the menu bar.
- **Units**

  Brings *Units* dialog which allows the user to adjust the units for the model.
- **Projection...**

  The projection currently associated with the project can be specified via this command. Changing the projection does not alter the XYZ coordinates of the project data.
- **Reproject...**

  Reprojecting means to convert data from one coordinate system to another.
- **Single Point Reprojection...**

  This option can be used to view the coordinates of a point to be transformed.
- **Transform...**

  This command is used to scale, rotate and translate the entire project. Clicking on this command will bring up the *Transform* dialog. Individual objects can be transformed by using the same command found in right-click context menus for the item in the Project Explorer.
- **Preferences**

  Brings up the *Preferences* dialog where the user can adjust the general preferences for GMS.
- **Screen Capture...**

  Images in the Graphics Window can be copied to the Windows clipboard by selecting the **Copy** command from the *Edit* menu. Once on the clipboard, the image can be copied into other applications for report generation. If the user wants to copy an image that is larger or smaller than the graphics window this can be done by editing the copy scale factor in the Preferences dialog.
- **Paste Text**

  The **Paste** command in the *Edit* window allows the user to paste tabular data directly into GMS. When this command is executed the File Import Wizard is launched.

These were taken out of the menu

- **Coordinate System**

  Brings up the *Coordinate System* dialog which allows the user to adjust the current horizontal and vertical coordinate systems.
- **Coordinate Transformation**

  Brings up the *Coordinate Transformation Wizard* which allows the user to perform geographic system transformations, translations, rotations, and scaling.

Units

When building a ground water model, it is important to ensure that consistent units are used when entering model parameters. To simplify the management of model units, the user can define the units for length, time, mass, force and concentration in the **Units** dialog. A units label is placed next to each of the input fields in all the model dialogs in GMS where the units are known. For example, the units for hydraulic conductivity are length / time. If the length units are defined as "m" (meters) and the time units are defined as "d" (days) in the *Units* dialog, then the units string next to the hydraulic conductivity input field would be "m/d".

Concentration units can be defined separately, and potentially inconsistently with, mass and length units. This allows for more flexibility but can also lead to confusion so you should use care when selecting concentration units.

Unit Conversion

Generally speaking, GMS does not convert quantities from one system of units to another. GMS only displays the chosen units to help the user make sure they are being consistent. However, in a few places, GMS will use the currently defined units in it's calculations. These include:

- in the *FEMWATER Fluid Properties* dialog
- in the *Curve Generator* dialog
- when calculating the stream stage constant when saving MODFLOW, and
- in the measure tool.

Preferences

The ***Edit | Preferences*** command brings up the *Preferences* dialog. There are four items in the preferences list that can be selected to modify which preferences can be edited.

General

The *General* tab has all the of the general options in GMS.

- Restore Factory Preferences

  This button switches all settings to the factory defaults.
- Show welcome dialog on startup
- Check for newer version on startup
- Confirm Deletions

  Whenever a set of selected objects is about to be deleted, the user can choose to be prompted to confirm the deletion. This is meant to ensure that objects are not deleted accidentally.
- Show new project wizard on startup
- XMDF File Compression

  When saving XMDF files, you can use compression. The compression factor can be specified.
- Project Explorer - The Project Explorer section allows the user modify preferences related to the Project Explorer.
  - Change Module when tree selection changes.

    This option changes the current module when a item is selected in the Project Explorer. This option is on by default.

- Scroll Project Explorer when changing module.

  This option will ensure the visibility of the tree item objects in a certain module when the module is changed.
- Synchronize active dataset and elevation dataset for TINs, 2D Meshes, 2D Grids, and 2D Scatter Sets.

- Contouring

  Allows the user to enter a filled contour tolerance.

- North arrow paths

  Specify the path to the folder containing the North Arrows.

Model Executables

The *Models* page allows the user to specify the location of model executables, as well as the option to use the Model Wrapper.

- *Model Wrapper*

  GMS is a pre- and post-processor for Numerical Models. Most of these numerical models are run externally in the DOS environment. The default option is to use the Model Wrapper. The Model Wrapper "wraps" itself around the same DOS executables and gives more model feedback including graphs and tables. The Model Wrapper includes a toggle box in the bottom left that, when checked and you exit the Model Wrapper, GMS will automatically read in the results of the solution. This toggle will only be checked by default if the model converged. The option to use the Model Wrapper or the old DOS view is included in the Preferences Dialog. Certain models, including Stochastic and inverse modeling, can only be run using the Model Wrapper.

Parallel versions of MODFLOW-2000, MODFLOW-2005, and SEAWAT are shipped with GMS to run simulations. Beginning with version 8, GMS ships parallel versions where the SAMG solver has been parallelized. Even when this option is on, when running Parallel PEST with MODFLOW the serial (non-parallel) version of MODFLOW will be used since the Parallel PEST will use all of the available cores on the computer with the serial version. Also see MODFLOW preferences for an option to turn on or off the parallel version.

MODFLOW

The *MODFLOW* page has options related to the MODFLOW interface in GMS.

- Compress MODFLOW H5 files

  This option will force the H5 files saved with MODFLOW to be compressed. Generally this option should be turned on.
- Create h5 copy of head solution

  Turning on this option can speed up reading the MODFLOW head solution especially when there are a large number of stress periods. When this option is on, GMS writes an HDF5 copy of the MODFLOW head solution upon reading it the first time. The following times GMS reads the head solution, it doesn't take as much time.
- Default version of MODFLOW to run
  - Double precision

    When this option is on, GMS will use the double precision version of MODFLOW to run simulations. By default this option is off and GMS uses the single precision version of MODFLOW.
  - Parallel (not used by Parallel PEST)

When this option is on, GMS will use the parallel version of MODFLOW to run simulations. Beginning with version 8, GMS ships parallel versions of MODFLOW where the SAMG solver has been parallelized. Even when this option is on, Parallel PEST will use the serial (non-parallel) version of MODFLOW since the Parallel PEST will use all of the available cores on the computer.

Images / CAD

- Bitmap Scale Factor

  When the copy command is selected, a bitmap image of the screen is placed on the clipboard. The scale factor can be used to increase :or decrease the resolution of the bitmap. This scale factor also applies when saving the image as a bitmap.

- CAD Symbol Size

  The size of the CAD symbol display can be specified.

Printing

Allows the user to specify scaling Points, Lines, and Fonts while printing.

Program Mode

GMS has the option of changing the "mode" or "skin" that GMS is running. The purpose of the mode is to simplify the items available in the interface.

Available Modes

- GMS

  This is the default mode where are the tools are available to the user.

- GMS 2D

  GMS 2D is for users that are only interested in using GMS to do seepage and slope stability. In GMS 2D mode the only tools that are available are the 2D mesh and the Map. There are several changes to the GMS interface when in this mode, including:

  1. Hiding of tools and modules not in use
  2. SEEP2D model automatically initialized from conceptual model
  3. SEEP2D boundary conditions automatically mapped from conceptual model (no need to run Map -> SEEP2D)
  4. Easily create all coverage types in one step
  5. Feature object types automatically assigned based on coverage attributes
  6. Allow only one conceptual model

- GMS Site

  GMS Site is for users that are only interested in using tools for site characterization. As such the tools available in this mode are TINs, Boreholes, Solids, 2D Scatter, and Map. These tools can be used to construct solid models of stratigraphy.

Caveats

When GMS is running in a particular mode then only data associated with the available tools will be read into GMS. For example, if I have a GMS project file that contains a 3D Grid, and I am running in "GMS 2D" mode then when I attempt to read in the project file I will get a message that GMS is not enabled to read in 3D Grid data. The only data that will read into GMS in "GMS 2D" mode is data associate with the 2D Mesh and the Map.

Map

Gives the user the option to automatically perform the **Map→SEEP2D** command.

TINs

- Vertex options
  - Retriangulate after deleting

    Default z-value
  - Confirm z-value
  - Interpolate for default z on interior
  - Extrapolate for default z on exterior
- Breakline options
  - Add supplementary points
  - Swap edges
- Minimum length ratio

Boreholes

- Auto-assign horizons

    Rows per lift

    Row overlapped between lifts

    Sets retained between lifts
  - Measure slope from borehole tips

    Average horizon slope

    Horizon slope weight

    Gap weight
- Auto-Fill cross sections
  - Prompt for vertex spacing
  - Do not redistribute vertices
  - Use default spacing

    Default spacing/length ratio

    Max. lense thickness ratio

Scatter sets

- 2D scatter

  Default dataset value

  - Confirm date set value
- 3D scatter

  Default dataset value

  Default Z value

  - Confirm date set value

2D Mesh

- Interpolate for default z on interior

  Default z (ft)
- Assign default z-value
- Prompt for z-value
- Insert nodes into triangulated mesh
- Check for coincident nodes
- Retriangulate voids when deleting

  Thin triangle aspect ratio

Graphics

- Ask which library to use if system does not support all functionality
- Autoselect the software library if system does support all functionality
- Always use system library (may not support all features)
- Always use software library (may be slower)

Materials

Many of the data types supported by GMS (ex., elements, solids, borehole regions) have a material ID associated with each object. This material ID is an index into a list of material types. These material types often represent different types of soil or rock. A global list of material attributes is maintained that can be edited using the ***Materials*** command in the *Edit* menu.

Insert, delete, move up, move down buttons

The Materials are listed in a table or spreadsheet. New materials can be created by typing in the last row, or by copying and pasting from a spreadsheet, or by selecting the **Insert** button.

Materials have an ID, name, color and pattern. If a numerical model interface is active (MODFLOW, FEMWATER etc.), the *Materials* dialog will show a tab control for each active model. Additional columns in the spreadsheet are added which correspond to the appropriate properties for the model.

A legend showing all the materials can be displayed in the Graphics Window. The legend can be turned on and off in the *Display Options* dialog.

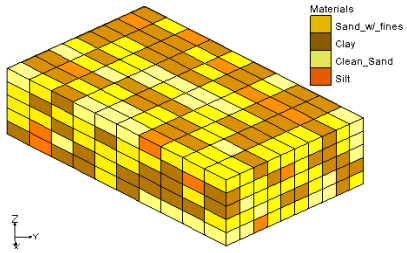
The material names, colors, patterns, and transparencies can be exported by selecting the **Export** button. This will save out a "material" file with a *.mat extension. This file can be imported into GMS using the *File* | **Open** command.

Related Topics

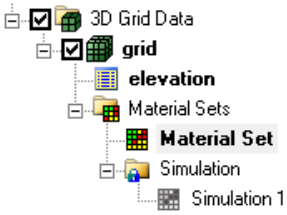
- Edit Menu

Material Set

A material set is similar to a dataset but represents materials instead of dataset values. Material sets can exist with 3D grids and meshes and can be created manually or by running a T-PROGS simulation. To create a material set manually you can right-click on the 3D grid or mesh in the Project Explorer and select the *New Material Set* command. Doing so will create a material set from the current cell or element materials. All Material sets are grouped under a Material Sets folder in the Project Explorer. Although grid cells and mesh elements (as well as other objects) have materials associated with them from the start and these material assignments can be changed on a cell-by-cell (or element-by-element) basis, a material set is not created until you specifically create it. Clicking on a material set causes the set to be applied to the grid cells (or mesh elements) and the material assignments of the cells updated.



3D grid showing materials from a material set.



A material set in the Project Explorer.

Properties

As with datasets, a material set has properties that can be viewed. Right-clicking on the material set in the Project Explorer and selecting the **Properties** command brings up the *Material Set Info* dialog. This dialog lists the materials in the material set, their frequency and percentage. A histogram is also drawn showing the frequency of the different materials in the set.

Viewing / editing

Clicking the **Edit Materials** button in the *Material Set Info* dialog brings up a table showing the materials in the material set, similar to viewing values in a dataset. The materials can be edited and saved.

Inactive values

As with datasets, materials sets can include inactive values (starting at GMS version 8.0). Inactive materials are rendered using a material with ID -9999999.

Material set solutions

TPROGS can generate multiple material sets as part of its solution. These are stored in a material set solution folder which is locked against editing. Material set solutions can be used to run a stochastic simulation in MODFLOW.

3.2.1. Coordinate Systems

Coordinate Systems

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

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

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

Horizontal System

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

- Local
- Geographic
- UTM
- State Plane
- CPP

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

Horizontal Units

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

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

Ellipsoid

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

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

Vertical System

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

- Local
- NGVD 29
- NAVD 88

Vertical Units

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

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

External Links

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

References

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

Coordinate Conversions

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

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

"Convert From" System

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

"Convert To" System

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

Restrictions

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

Projections

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

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

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

Project Projection

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

Local Projection

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

Reproject

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

There are three basic reprojection tasks that you can do:

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

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

Reproject everything

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

Reproject object

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

Single Point Reprojection

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

Restrictions

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

Supported Projections

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

References

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

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 = 6378206.4 m. (Clarke 1866 major spheroid radius)

Geographic Coordinate System

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

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

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

Local Coordinate System

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

Three scenarios are possible when dealing with local coordinate systems:

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

Coordinate Transformation

Coordinate Transformation means changing the position (xyz coordinates) of data. There are two ways to access this feature:

- ***Edit\Coordinate Transformation*** menu command. Accessed this way the command will transform all your data.
- **Right-click on an object in the Project Explorer** and select the *Transform* command. This transforms just the object clicked on.

Coordinate Transformation Wizard

Doing either of the above will bring up the *Coordinate Transformation Wizard*. There are two steps.

Step 1 - Choose the method of transformation

The two options are:

- *Geographic system transformation*. This option is used to transform from one coordinate system to another.
- *Translate, rotate, scale*. This performs a simple transformation to move, scale, or rotate your data.

Step 2 - Choose transformation options

Geographic system transformation option

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

"Convert From" System

The "Convert From" system defines the coordinate system the data is currently referenced to. When working from the Edit menu, this is the system GMS is working in and is dimmed by default because it is assumed you have already specified this system. When working from the Project Explorer, this is the coordinate system of the selected entity and must be selected. You may edit the current coordinate system by selecting the check box at the top of the dialog.

"Convert To" System

The "Convert To" system defines the system you will be working in after the conversion. When the dialog is invoked from the Edit menu, this is selected by the user and all data is converted from the current system to this new system. When the dialog is invoked from the Project Explorer, the "Convert To" system is dimmed because this is assumed to be the system GMS is working in and all other data is already in this system. In the convert to section you may select the horizontal and vertical system to be used after your conversion.

Restrictions

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

Translate, rotate, scale option

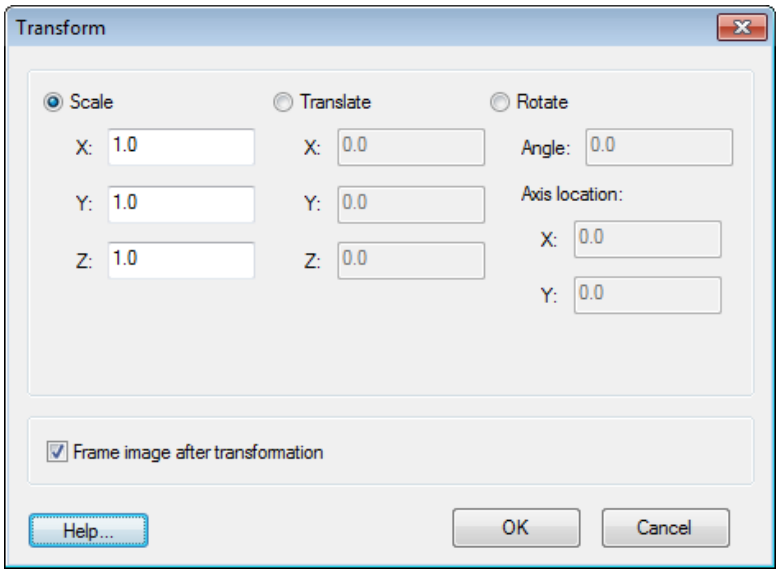
If the *Translate, rotate, scale* option was selected in step one, step two lets you chose to either translate, rotate or scale. You cannot do more than one transformation at a time - for example you cannot translate and scale at the same time. If you want to do both, you can perform the translation first, then perform the scale.

See Also

- Transform

Transform

The **Transform** command is used to move objects. In the *Transform* dialog, the transformation type can be chosen and then appropriate parameters can be entered. The following transformation types are available:



Transform dialog

- **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 around the Z axis. The rotation will occur counter-clockwise by the specified angle around the specified center of rotation.

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.

See Also

- Coordinate Transformation
- Edit Menu

3.3. The Display Menu

Display Menu

The *Display* menu is one of the standard menus and is available in all of the modules. The commands in the *Display* menu are as follows:

- **Display Options**

  Brings up the *Display Options* dialog
- **Contour Options**

  Opens the *Contour Options* dialog
- **Refresh**

  When editing the image in the Graphics Window it occasionally becomes necessary to update the display or refresh the screen by redrawing the image. Whenever possible, GMS automatically updates the display. However, in some cases small parts may be obscured by editing procedures. If so, the display can be refreshed by selecting the **Refresh** command from the *Display* menu. After selecting the **Refresh** command, the display process can be aborted by pressing the *ESC* key.
- **Refresh Display**

  When editing the image in the Graphics Window it occasionally becomes necessary to update the display or refresh the screen by redrawing the image. Whenever possible, GMS automatically updates the display. However, in some cases small parts may be obscured by editing procedures. If so, the display can be refreshed by selecting the ***Refresh Display*** command from the *Display* menu. After selecting the ***Refresh Display*** command, the display process can be aborted by pressing the ESC key.
- **Redraw Display**

  When zooming in and out in the Graphics Window some items such as arrows change size for increased drawing speed. This command refreshes the display and redraws these items at the size specified in the Display Options.
- **Frame Image**

  After altering the image display using the ***Zoom*** or ***Pan*** tools, the image can be centered by selecting the **Frame Image** command in the *View* menu. This command adjusts the window boundaries so that all currently visible objects just fit in the Graphics Window.

  Frames the window around whatever objects are currently selected.
- **Visibility**
  - **Hide –** A set of selected objects can be hidden by selecting the **Hide** command in the *Display* menu. Hidden objects cannot be selected. The **Frame Image** command only frames the image based on the visible objects.
  - **Show –** Hidden objects can be made visible by selecting the objects and choosing the **Show** command in the *Display* menu. This method applies to objects such as solids, TINs, and mesh layers which can be selected with an icon. Other items such as cells and elements cannot be selected when hidden but can be made visible by selecting the ***Show*** command with nothing selected. In this case, all hidden objects are made visible.
  - **Isolate –** In many cases it is useful to hide all of the current objects but one. One way to do this is to select all of the objects to hide (without selecting the desired object) and then select the **Hide** command. A quicker way to achieve the same result on one or more objects is to use the **Isolate** command in the *Display* menu. The

Isolate command makes the selected items visible and automatically makes all other items of the same type invisible.

- **View Angle**
  - The objects in the Graphics Window can be rotated and viewed in three dimensions. Two angles, bearing and dip, are used to rotate the view. 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. The bearing and dip angles can be explicitly defined in the *View Angle* dialog accessed by selecting the View Angle command from the *View* menu. The viewing angles can be manipulated interactively with the Rotate tool.
- **Window Bounds**
  - The region of the real world coordinate system that is mapped to the Graphics Window can be altered using the Pan and Zoom tools. It is also possible to precisely control the visible region by selecting the **Set Window Bounds** command from the *View* menu. This command brings up the *Set Window Boundaries* dialog. If the **X range to be specified (preserves aspect ratio)** option is selected, the x coordinate at the left and right and the y coordinate at the bottom of the **Graphics Window** are specified. The y coordinate at the top of the Graphics Window is not specified in order to maintain the aspect ratio. If the **Y range to be specified (preserves aspect ratio)** option is selected, the y coordinate at the top and bottom and the x coordinate at the left of the **Graphics Window** are specified. The x coordinate at the right of the **Graphics Window** is not specified in order to maintain the aspect ratio. If the **X and Y range to be specified (alters aspect ratio)** option is selected, the x coordinate at the right and left and y coordinate at the top and bottom of the **Graphics Window** are specified. Since all four coordinates are specified, the aspect ratio of the scene may be altered.
- **Views**
  - **Plan View –** Changes the viewing angle so that the user is looking down the z-axis with the x-axis horizontal and the y-axis vertical.
  - **Front View –** Changes the viewing angle so that the user is looking down the y-axis with the x-axis horizontal and the z-axis vertical.
  - **Side View –** Changes the viewing angle so that the user is looking down the x-axis with the y-axis horizontal and the z-axis vertical.
  - **Oblique View –** Changes the viewing angle so that the user is looking at the model at an angle of 45^{o} to each axis.
  - **Previous** &ndash Restores the Graphics Window viewing parameters as they were before the last viewing command was issued (rotate, zoom, pan, etc.).
- **General/Ortho Mode**

  A command is provided in the *Display* menu for switching between the orthogonal and general viewing modes. The orthogonal mode is only available with 3D grids.
- **Convert To Cad**

  Converts any visible geometric data into CAD format internally within GMS. This CAD data will be saved with the project as a DWG file, or can be exported as a separate DWG or DXF file. This command is found either in the the *Display* menu or by right-clicking in the empty space in the Project Explorer.
- **Toolbars**

  The *Toolbars* menu allows the user to hide and show the listed toolbars.
- **Plot Wizard**

  Brings up the *Plot Wizard* dialog.
- **Animate**

The **Animate** command launches the *Animation Wizard*.

- **Play Animation**

  The **Play Animation** command launches an *.avi player that allows the user to browse for an *.avi file and play the animation.

Contour Options

The options used to generate contours can be edited by selecting the ***Contour Options*** command in the **Data** menu. The items in the Contour Options dialog are as follows:

General vs. Grid Layers

There are two tabs associated with the *Contour Options* dialog. The options described below are for the General options. The Grid Layers options are used when contouring 3D grids.

Contour Method

There are 3 main types of contours: linear, color filled, and both linear and color filled.

Linear – The default method is Normal linear contours and causes the contours to be displayed as piece-wise linear strings.

Color Fill – If the Color fill between contours option is selected, the region between adjacent contour lines is filled with a solid color. There is also a smooth option available. When this option is turned on the color between contour lines varies according to the color ramp. The transparency of the filled contours can also be adjusted.

Linear and Color Fill – This option shows linear contours on top of color filled contours.

Block Fill - This option applies only to cell centered grids. Cells are filled with a single color. The color chosen corresponds to the color ramp value associated with the value at the cell center.

Line options

Bold contours – *The Bold every...* option can be used to display contours at selected intervals with a thicker line width.

Contour labels – *The Label every...* option can be used to plot labels on contours at selected intervals. The contour label options are edited using the Contour Labels dialog.

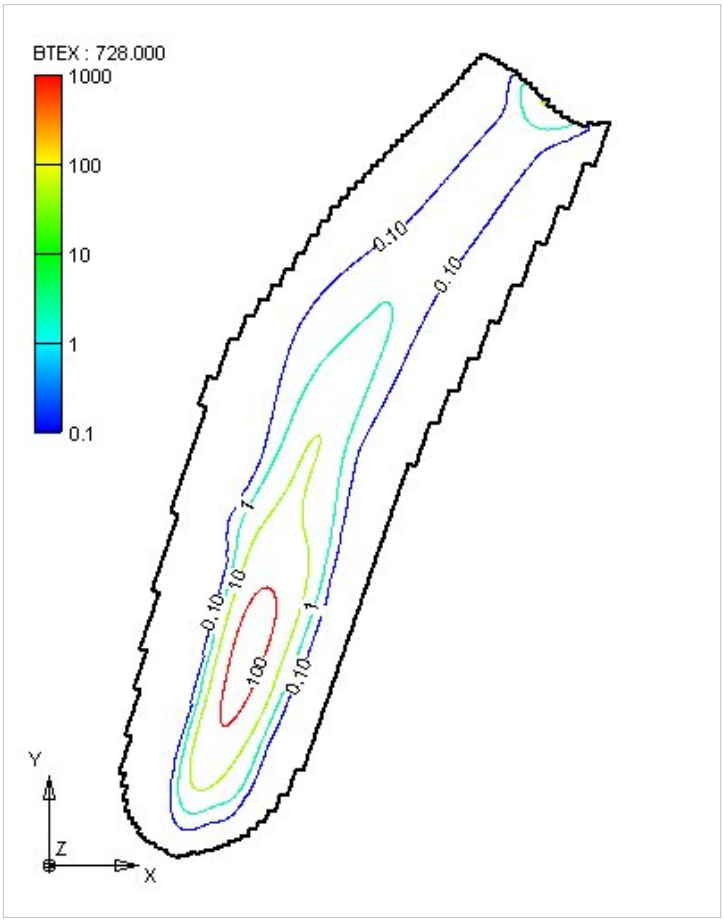
Data Range

The values shown in the Data range section correspond to the maximum and minimum values in the active data set. These values are sometimes useful when choosing an appropriate contour interval.

Use each timestep's max and min – This option is only applicable is your contours are colored according to a color ramp, and your data set has more than one time step. If this option is on, the maximum and minimum color ramp colors will be displayed for every timestep. Otherwise, the maximum and minimum colors apply to the entire data set and therefore will only appear when the current data set contains the data set maximum or minimum value.

Contour Specified Range – Regardless of which option is selected for the contour interval, a maximum and a minimum contour value can be specified and the contouring can be restricted to the specified range. If desired, the fill below or fill above options can be turned on to color fill above/below the specified range.

Contour Interval



The contour interval defines the values associated with the contours. Based on the option selected, the contour values are computed and displayed in the spreadsheet. The colors of the contours will also be displayed if a Color Ramp is being used.

Four options are provided for defining contour intervals in the Contour Options dialog. The options are as follows:

Number of Contours – With the Number of Contours option, an integer is entered representing the total number of contours. The contour interval is adjusted based on the current active data set so that the contours are evenly spaced and the number of contours correspond exactly to the specified value.

Specified Interval – With the Specified Interval option, the contour interval (5, 10, 20, etc.) is entered directly by the user.

Specified Values – The Specified Values option allows the user to enter a list specific contour values. Contours are only generated at these values.

Log Interval – When contouring chemical data, it is common to have a small "hot spot" somewhere in the interior of the grid where the concentrations are several orders of magnitude higher than the rest of the grid. In such cases, the contours are all biased to the high concentration zones and no contours are drawn in the low concentration zones. A common approach for dealing with such situations is to select a set of contour values corresponding to multiples of ten (0.01, 0.1, 1.0 10.0, etc.). This can be accomplished by explicitly assigning a set of contour intervals using the Specified Values option. The problem with this approach is that the colors assigned to the contours are still based on a linear interpolation of the data values, thus skewing the colors to the higher concentration zones.

If the Log Interval option is selected, GMS automatically assigns contour intervals as multiples of 10. Furthermore, if the color-fill or color ramp option is being used, the colors assigned to the contours are distributed in a logarithmic fashion, rather than linearly interpolated from the low to high values. A sample plot using logarithmic contour intervals is shown below:

If the Log interval option is selected, all contour intervals will be positive. No contours are drawn in areas where the data values are zero or negative. Furthermore, a "Starting value" must be entered. This value represents the minimum contour value.

Contour Labels

The ***Contour Label Options*** command in the **Data** menu is used to access the *Contour Label Options* dialog. This dialog is used to set the label color, the number of decimal places used to plot the label, and the spacing used when the labels are generated automatically. The default spacing value controls the placement of labels when labels are generated automatically.

Labels can be added to contours in one of two ways:

- 1. If the contour label option is selected in the Contour Options dialog, labels are automatically placed on the contours. The spacing of the labels is controlled with the Contour Labels dialog.
- 2. In some modules, contour labels can be added manually to contours by selecting the Contour Labels tool in the Tool Palette and clicking on the contours where labels are desired. By default, the data set value corresponding to the point that was clicked is computed and a label corresponding to the nearest contour value is drawn centered at the point that was clicked. An option can be set in the Contour Label Options dialog to use the exact value at the point that is clicked as opposed to using the nearest contour value. This option is useful to post data set value labels in regions where there are no contours.

Specified Dataset

The user specifies the dataset with the ***Select Dataset...*** button in the *Display Options|3D Grid* dialog. This brings up a new window where the specified dataset is selected. The contour options available for the specified dataset are the same as the active dataset options, yet each must be establish separately. This option is useful in comparing datasets.

Grid Layer Contour Options

When contouring 3D grid data, there are additional options available in the *Contour Options* dialog. These options let you chose the data that you want contoured. The default array used to fill cells is the active dataset. Another option for filling cells is the data colors option; this uses an input array from MODFLOW, MODPATH, MT3DMS, SEAM3D, or RT3D. Any of the input arrays from the models such as Kh, Kv, Recharge, etc. can be displayed, for any stress period. For MT3DMS/RT3D/SEAM3D, you can also specify the species.

Interpolation

GMS interpolates data from the cell centers (for cell-centered grids) to the cell corners and midsides so that the grid can be triangulated and contoured. By default, GMS uses a fast algorithm for interpolating the data that is adequate for the majority of situations. If the **Auto-interpolate contours** option is turned off, however, you can specify your own interpolation scheme. By default, GMS truncates the interpolated values to the max and min of the data, but you can turn that option off.

Animations

One of the most powerful visualization tools in GMS is animation. Animations can be created with transient or steady state data. The view angle and zoom factor can be changed during the animation as well. Animations are saved in the Windows (*.avi) format. AVI files can be played back externally to GMS using a variety of applications and can be inserted into multi-media documents and applications.

A new animation can be created by selecting the **Animate** command from the *Display* menu.

Animation Options

There are 4 options available when creating an animation:

Option	Type
Dataset	Transient
Particle Tracking	Transient
Cross-sections / Iso-surfaces	Steady State
Moving View	N/A

Transient vs. Steady State Animation

All of the options except *Moving view* are either considered Transient or Steady state. You cannot use a *Transient* option and a *Steady State* option at the same time. Moving view, however, can be used by itself or in conjunction with any combination of the other options.

The typical case for choosing to do a transient animation is when you have a transient solution generated by a model. The transient animation could illustrate how vectors, contours, fringes or iso-surfaces change at different solution times. A transient animation can also be used to show particle tracking over time, even with a steady state solution.

Steady state animation can only be performed using 3D grids or 3D meshes. A steady state animation can be used with a steady state dataset, or with one time step of a transient dataset. Two options are available for steady state animation: Cross-section / Iso-surface animation and Flow trace animation.

Dataset Animation

Transient dataset animation can be used with any object with a transient dataset. As each frame is generated, a set of values corresponding to the current time is loaded into memory and the image is redrawn using the current display options. Thus, if the *contour display* option is selected, the contours will vary from frame to frame according to the changes in the dataset.

The total number of frames generated in the animation 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 if necessary using linear interpolation of the data values at the specified time steps.

Particle Tracking Animation

Particle tracking can be animated to show how pathlines grow over time. Although it is considered a *Transient* option in the table above, this option can be used with a transient or a steady state solution. Only the forward tracking particle sets can be animated.

If Particle tracking is selected by itself, you can choose how many frames you want in the animation. GMS will then compute the time at which to generate each frame by dividing the total animation time evenly by the number of frames. The animation begins at the minimum start time of all forward tracking particle sets, and ends at the maximum travel time of all particles (in all forward tracking particle sets).

Cross-section / Iso-surface Animation

If the *Geometric surface animation* option is chosen, a cutting plane and/or an iso-surface can be animated. If the Animate cutting plane over specified XYZ range option is chosen, an x, y, and/or z cutting plane is incrementally moved through the mesh or grid from the specified beginning location to the ending location as each frame is generated. This generates an animation showing a moving cross section. If the Animate iso-surface option is chosen, a single iso-value is incrementally varied between the specified beginning and ending values and a different iso-surface is generated for each frame based on the value.

Moving View

The *Moving View* option can be used by itself or in conjunction with any combination of the other options. This option can be used to change how you view the data as the animation proceeds. You can define views and associate them with particular frames of the animation. When the animation is generated, GMS will interpolate between the defined views to determine what the view should be at every frame.

Flow Trace Animation

If the *Animate flow trace* option is chosen, a flow trace animation of the steady state vector data on 3D grid or 3D mesh cross sections is generated. Flow trace animation is a special type of animation that is similar to particle tracking. A series of particles is randomly generated on the cross sections and it is traced through time. Each particle has a limited life span. As a result, the particles appear as a series of streaks. Flow trace animation can result in highly intuitive images of a vector field.

If there is no vector dataset or cross section associated with the current 3D grid or 3D mesh, the *Animate flow trace* option and *Flow trace* options button are dimmed. The *Animate flow trace* option is also dimmed if the *Flow trace* option is not selected in the *Cross Section Options* dialog. If the active vector dataset is transient, only the current time step is used to generate the flow trace animation.

The *Avg. number of particles per triangle* edit field specifies the density of particles to be generated in the animation. The Decay ratio specifies the amount of time necessary for a particle path to decay as it passes points in the cross section. The Velocity magnitude limit and Velocity direction limit specify the distance that a particle will travel between consecutive frames.

CODEC

Starting in GMS 6.5 you can choose what codec you use to create your AVI movie. GMS will search your computer for all compatible codecs and they will be available in the pull down menu.

Animation Playback

Once a new animation has been generated, GMS 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.

Color Ramp

Several of the display options (contours, iso-surfaces, color-shaded vectors, etc.) use a color ramp to vary the display color based on a relative value. The color ramp can be edited by selecting the **Color Ramp Options** command in the *Data* menu. The *Color Ramp Options* dialog contains the following options:

Legend

If the *Legend* option is selected, a vertical strip of colors with a legend of corresponding dataset values is displayed in the upper left corner of the *Graphics Window* whenever the color ramp is used to display an object (shading, color contours, etc.). The length and width of the color legend are specified with the Legend width and Legend height edit fields. The values entered for legend width and height are in screen pixels.

Palette Method

The *Palette Method* section lists the three basic options for defining the color ramp:

Intensity Ramp

The *Intensity Ramp* option defines a ramp of colors corresponding to a varying intensities of a single color.

Hue Ramp

The *Hue Ramp* option defines a ramp of color hues (red-yellow-green-blue-magenta) at the maximum color intensity.

User-Defined

If the *User-Defined* option is selected, a user-defined color palette is used to define the color ramp.

User-Defined Palettes

The user-defined color palettes are listed in the upper right corner of the *Color Ramp Options* dialog. A new palette is created by selecting the **New** button. This button brings up the *New Palette* dialog listing a set of options for defining the initial color palette. These colors can be edited using the *Current Palette* section of the *Color Ramp Options* dialog. An existing palette can be deleted using the Delete button.

Once a set of user-defined color palettes are created, they are saved with the project to a palette file (*.pal). The **Import** and **Export** buttons can be used to share user-defined palettes between projects.

Current Palette

The current color palette is displayed in the *Current Palette* section. The min and max value of the color ramp can be set by clicking and dragging the two triangles just below the color palette. For user-defined color palettes, new colors can be added, colors can be deleted, and the color associated with a color entry can be edited using the tools just below the palette.

The data value associated with a selected color can be edited either by dragging the color or by entering a new value directly. The values can be displayed as either percentages or direct values (corresponding to the active dataset). The Edit Table button can be used to edit the colors and corresponding values directly in a tabular format.

Preview

The *Preview* section at the bottom of the *Color Ramp Options* dialog displays the color ramp defined by the current palette and max and min values. The **Reverse** button can be used to reverse the direction of the color ramp (for example, to switch from red-yellow-green-blue to blue-green-yellow-red).

3.3.1. Display Options

Display Options

Most of the data types in GMS have a set of display options that can be modified using the **Display Options** command in the *Display* menu. The **Display Options** command brings up the *Display Options* dialog. The dialog can also be brought up using the **Display Options** macro.

Data Type Tabs

The *Display Options* dialog contains a list of the data objects currently in memory. Corresponding data tabs are displayed in the dialog depending on what object type is selected in the list box. Each display feature associated with a data type is listed in the *Display Options* dialog. The check box next to the feature named can be toggled on or off to control whether or not the feature is to be displayed. In addition, the button next to the feature brings up a dialog that can be used to edit the display attributes of the feature (color, font, line thickness, etc.). The following table describes the display options tabs that exist for the various data types. Click on the links to learn more about the specific display options for each data type.

Image	Data Type	Tabs Also Available for Data Type
	TIN Data	
	Borehole Data	
	Solid Data	
	2D Mesh Data	SEEP2D
	2D Grid Data	
	2D Scatter Data	
	3D Mesh Data	FEMWATER, Particles, WASH123D, ADH
	3D Grid Data	MODFLOW, Particles, MT3D
	3D Scatter Data	
	Map Data	MODAEM, UTEXAS
	GIS Data	
	Cross Sections	
	Lighting Options	
	Axes	
	Drawing Grid	

General Display Options

The *Display Options* dialog also controls several general display options in GMS. These general display options are described in the table below.

Display Option	Description
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.
Background color	The *Color* dialog can be brought up to change the selected color. The selected color is displayed as the background color in the Graphics Window.
Triad options	To aide in visualization of 3D objects in oblique view, a XYZ triad can be displayed in the lower left corner of the Graphics Window. The display, size, and color of the triad can be specified in the *Display Options* dialog. The triad is useful in visualizing how the geometry currently defined in GMS aligns with the world coordinate system.

Drawing Grid Options

When entering new nodes or entering a polygon or polyline in plan view, it is often useful to have the coordinates snap to a uniform grid. This allows accurate placement of the objects when the desired coordinates are even multiples of some number. The properties of the drawing grid can be controlled through the *Drawing Grid* tab of the *Display Options* dialog. This dialog is accessed from the *Display* menu or the **Display Options** macro. The following table describes the display options available for the Drawing Grid.

Display Option	Description
Spacing	The *Spacing* edit field specifies the spacing of the grid nodes and grid lines in the drawing grid. The *Grid color* window specifies the color that is used to display the drawing grid in the *Graphics Window*.
Snap	If the *Snap* option is selected, all new vertices, nodes, points, etc., snap to the closest grid point as they are being created or when they are dragged interactively.
Display grid lines	If the *Display grid lines* option is selected, grid lines are displayed according to the Line spacing increment. For example, if the Grid spacing is set to 10 and the Line spacing increment is set to 5, a grid line will be drawn every 50 units. The grid line color can also be adjusted using the button next to the Display grid lines toggle.
Display grid points	If the *Display grid points* option is selected, grid points are displayed according to the Grid point spacing increment. The grid point color can also be adjusted using the button next to the Display grid points toggle.

Related Topics

- Display Options

Vectors

If the *Vectors* item in the *Display Options* dialog is selected for an object (TIN, Grid, or Mesh), vector plots can be generated using the active vector dataset for the object. One vector is placed at each node, cell, or vertex.

The display of vectors can be controlled using the *Vector Options* dialog accessed through the **Vector Options** command in the *Data* menu or from a button in the *Display Options* dialog. The dialog options are as follows:

Dimensions

The *Auto compute length* toggle is on then GMS will compute the length of the vectors. If this toggle is off then the Length can be edited. The edit fields labeled Length, Head width, Head length, and Stem width control the size and shape of the vectors. The Head width, Head length, and Stem width are expressed as a percentage of the Length of the vector.

Vary Length and/or Color

Often it is desirable to vary the display of the vectors according to the magnitude of the vector function at a current point. This can be done with the *Vary length according to magnitude* and *Vary color according to magnitude*. If the *Vary length according to magnitude* option is selected each vector is displayed with a length equal to the magnitude of the vector multiplied by the *Scaling* ratio. If the *Vary color according to magnitude* option is selected, the vectors with the smallest and largest magnitude are draw in the color of the lowest and highest colors on the current color ramp. All other vectors are drawn in a color which corresponds to its magnitude from the current color ramp.

Display Every Nth Vector

If the *Display every _nth vector* is selected, only every nth vector is drawn. This is useful when the model is so large that drawing every vector clutters the display.

Color Specified Range

It is possible to have the current color ramp vary over a specified range of the active vector dataset. This is done by selecting the *Color specified range* option and editing the *Min magnitude* and *Max magnitude* edit fields.

2D vs. 3D Vectors

If you are in one of the orthogonal views (xy plane, xz plane, yz plane) the vectors will be displayed as 2D vectors. If you are in oblique view then the vectors are displayed a 3D vectors.

Lighting Options

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

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

Toggles

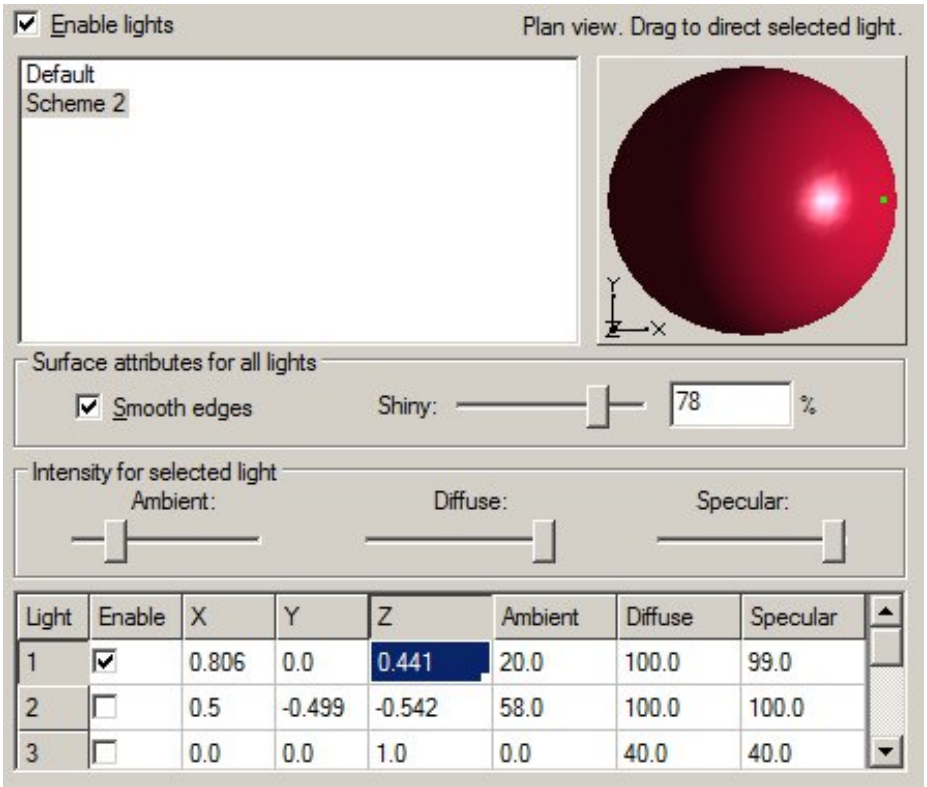
The first toggle allows the user to turn on the use of a light source. When this toggle is selected, the second toggle becomes available. The second toggle tells SMS to smooth corners between adjacent faces. This allows the faceted surface to appear as a smooth surface.

Slider

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

Light Position

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



The following table describes the lighting display options.

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

Plot Axes

The Plot Axes are a set of ruled lines oriented in either the world coordinate system or the grid coordinate system. The axes can be either 2D or 3D. The properties the plot axes can controlled through the Plot Axes tab of the *Display Options* dialog. The *Plot Axes* options are accessed by selecting the *Axes* item in the *Display Options* dialog. The *Plot Axes* tab is then displayed on the dialog, and is used to specify the attributes of the axes, axes ticks, axes labels, and axes numbers. The following table describes the display options available for the plot axes.

Display Option	Description
Display plot axes	If the Display plot axes toggle is selected, the plot axes will be displayed as specified and a sample of the plot axes will appear in the window in the upper left of the dialog. If the Display plot axes toggle is not selected, all plot axes will be hidden and the rest of the dialog will be dimmed.
Origin options	If the *Use world coordinate origin* option is selected, the numbers displayed on the plot axes are defined by the origin of the world coordinate system. If there is either a 2D or 3D grid currently defined in GMS, the origin of the grid can be used to define the numbering by selecting either the Use *3D grid origin* option or the *Use 2D grid origin* option. If there is not a 3D grid defined in GMS, the *Use 3D grid origin* option is dimmed. Likewise, if there is not a 2D grid defined in GMS, the Use *2D grid origin* option is dimmed. If the world coordinate origin is used, the world coordinate system will also be used to align the axes. However, if the origin of either the currently defined 2D or 3D grid is used, the plot axes can be aligned with either the world coordinate system or the local grid coordinate system. If the angle of rotation of the currently defined grid is 0.00 degrees, there is no difference between the world coordinate system and the local grid coordinate system. If there is an angle of rotation other than 0.00 degrees, the local grid coordinate system can be used by selecting the Local grid coordinates option. The world coordinate system can be used by selecting the World coordinates option.
Use exponential notation	This option controls whether or not the axes numbers are displayed using exponential notation.
Axis extents	The extents of the plot axes are specified with the controls in the spreadsheet. The *Auto scale X*, *Auto scale Y*, and *Auto scale Z* options are used to specify that the extents and spacing of the axes will be automatically calculated by GMS. If extents or spacing other than the calculated defaults are desired, any or all of the **Manual scale X**, **Manual scale Y**, and **Manual scale Z** options can be selected. If one of the Manual scale options is chosen, the min, max, tick interval, and how many ticks are to be labeled are specified for the corresponding direction. All of the directions that use the **Auto scale** option are further controlled by the *Fit to bounding volume* and the *Offset from bounding volume* options. If the *Fit to bounding volume* option is selected, the axes will be placed such that they fit tightly to the bounding volume of the currently defined objects in GMS. If the *Offset from bounding volume* option is chosen, the axes will be placed such that they are 15-20% larger than the bounding volume of the currently defined object in GMS.
Axis properties	The axis properties can be changed for each axis by right-clicking no the axis in the canvas window. The right-click menu options include *Hide axis* and *Edit Axis*. The *Edit Axis* options brings up a separate dialog with attributes including color and size that can be specified.

3.4. Other Tools

Annotations

Annotation Objects

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

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

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

Screen vs World Space Layers

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

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

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

Annotation Object Attributes

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

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

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

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

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

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

Screen Space Images

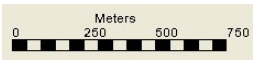
A screen space image is simply a graphics icon mapped to the screen. A typical application would be to display a company, department, or municipality logo next to the numeric model being displayed in the graphics window.

Attributes of the screen space images include:

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

Scale Bars

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



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

Attributes of the scale bar include:

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

North Arrows

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

Attributes of north arrows include:

- See screen space image attributes.

Text

Text can be created in world or screen space layers.

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

You can set the following attributes for text:

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

Lines/Arrows

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

The attributes available for Lines/Arrows include:

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

Ovals

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

The attributes that can be specified for ovals include:

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

Rectangles

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

Selection

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

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

Viewing Annotations At Specific Time Intervals

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

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

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

CAD Options

DWG and DXF files can be imported into GMS and displayed to assist in model placement or simply to enhance the display of a model.

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

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 → Feature Objects

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

CAD → TIN

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

Convert To CAD

This command converts any visible geometric data into CAD format internally within GMS. This CAD data will be saved with the project as a DWG file, or can be exported as a separate DWG or DXF file. This command is found either in the the *Display* menu or by right-clicking in the empty space in the Project Explorer.

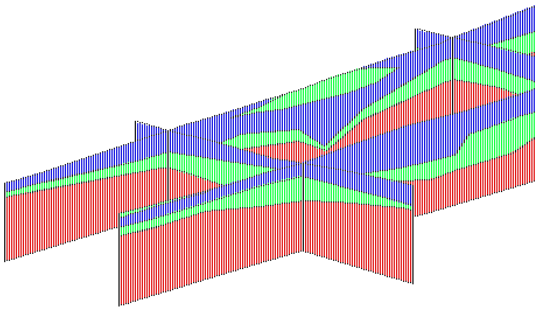
Cross Sections

Cross sections—also referred to as fence diagrams—are flat surfaces used to visualize the subsurface. GMS has different types of cross section objects:

- Solid Cross Sections
- Borehole Cross Sections
- 3D Grid Cross Sections
- 3D Mesh Cross Sections

Solid Cross Sections

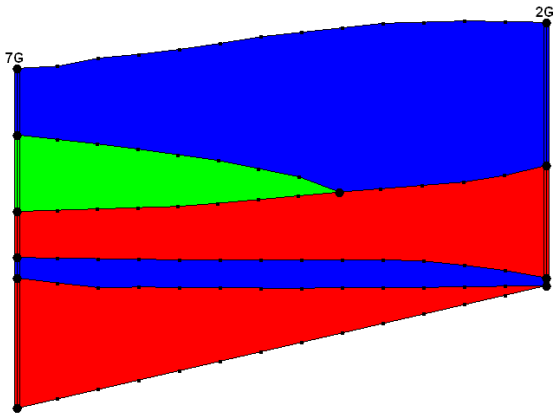
Solid cross sections can be created by "slicing" through a set of solids using the Create Cross Section tool. This can be done at any angle and the slicing can be done using a multi-segment polyline. Solid cross sections can be converted to a conceptual model. This is useful if you have a solid model of an embankment that you want to analyze using UTEXAS.



Example of fence diagrams created using solids

Borehole Cross Sections

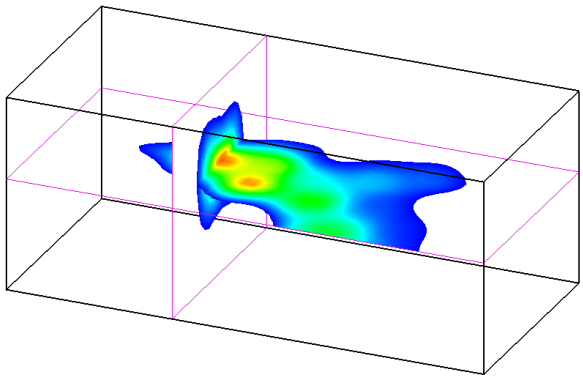
Borehole cross sections are quite different than solid, 3D grid, and 3D mesh cross sections. See the page on Borehole Cross Sections.



Borehole cross section

3D Grid Cross Sections

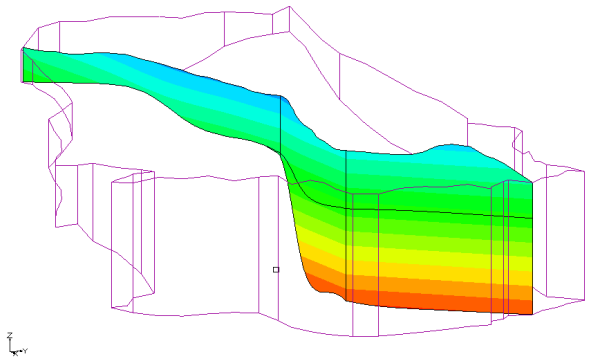
3D grid cross sections are created similar to solid cross sections. Data sets are automatically interpolated from the 3D grid to the cross sections and contours can be displayed on the cross sections.



Two cross sections cut through a 3D grid

3D Mesh Cross Sections

3D mesh cross sections are created similar to solid cross sections. Data sets are automatically interpolated from the 3D mesh to the cross sections and contours can be displayed on the cross sections.



Multi-panel cross section cut through a 3D mesh

Cross Section Options

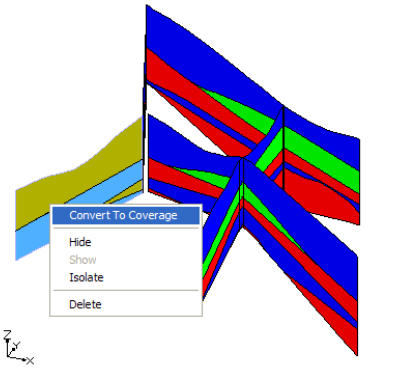
When cross sections are created from a mesh or a grid, values of the active scalar and vector data sets are interpolated to the cross sections. Whenever a new data set is chosen as the active data set for the mesh, the data values are re-interpolated to the cross sections.

The properties of all cross section data that GMS displays on the screen can be controlled through the Cross Sections tab of the Display Options dialog. This dialog is opened by right clicking on the Cross Sections 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 following table describes the display options available for the cross sections.

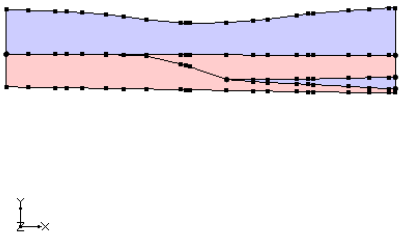
Display Option	Description
Interior edge removal	By default, the lines representing the intersection of the cross section with the faces of the cells or elements are displayed on the cross section. These lines can be hidden by selecting the Interior edge removal option.
Cross section edges	If this option is on, the lines that make up the cross section are displayed.
Cross section faces	If this option is on, then the cross section will be displayed as a set of filled polygons.
Contours	If the Contours item is selected, contours are displayed on the cross sections using the active scalar dataset.
Vectors	If the Vectors item is selected, vectors are displayed on the cross sections using the active vector dataset when the cross sections are displayed.
Flow trace	If the Flow trace item is selected, a flow trace image is texture mapped to each cross section using the active vector dataset when flow trace option is used with the Animation wizard.

Convert Cross Section to Coverage

Cross sections can be converted to coverages by right clicking on the coverage and selecting the ***Convert To Coverage*** command. The outer boundary of each material zone is converted into arcs and polygons are automatically built.



Converting a cross section to a coverage.



The resulting coverage from the cross section.

When this command is executed the user is prompted to select where the new coverage(s) should be created in the project explorer. If the user wishes to have the materials from the cross section assigned to the appropriate polygons in the coverage then the user should select an appropriate conceptual model. The conceptual model must support the assignment of materials to polygons (MODFLOW, SEEP2D/UTEXAS, FEMWATER).

Coordinate Transformation

The coordinates of points from the cross section are transformed into the XY plane. The x coordinates are calculated relative to zero where zero is defined as one end of the cross section. Currently the end of the cross section that is designated as zero is the end with the minimum Y coordinate. If the cross section is horizontal then the end of the cross section with the minimum X is used.

The x coordinates are calculated by finding the distance of the point from the end of the cross section that was set as zero. The Y coordinates are calculated by subtracting the minimum z value of the cross section from the z value of the point (y = z - z_min).

Limitations

- The cross section must be vertical.
- The cross section must be a single panel (when viewing the cross section in plan view it appears as a single line segment).

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) or transient (one value per item per time step). The values in the dataset can be scalar values or vector values. Certain types of objects in GMS have an associated list of scalar datasets and a list of vector datasets. Each of the following objects in GMS can have both scalar a vector datasets:

- TINs
- 2D Meshes
- 2D Grids
- 3D Meshes
- 3D Grids

The following objects can only have scalar datasets:

- Borehole Sample Data
- 2D Scatter Point Sets
- 3D Scatter Point Set

The commands for manipulating datasets are located in the Data menu.

Datasets are used for both pre- and post-processing of models. For example, a scalar dataset associated with a 3D grid can represent starting values of head or values of hydraulic conductivity for a groundwater modeling problem. Another dataset associated with the same grid may represent computed head values. Datasets can be used to generate contours, iso-surfaces, vector plots, and animation sequences.

Generating Datasets

Datasets can be generated in a variety of ways. They can represent output from a groundwater model (head, drawdown, etc.). They can represent tabular values in a text file entered by the user or exported from another application such as a GIS. They can be created by interpolating from a scatter point set to a TIN, grid, or mesh. Datasets can also be generated by performing mathematical operations on other datasets with the Data Calculator.

One advantage of the dataset list approach for managing information is that it facilitates transfer of information between different types of models or models with differing resolution. This is accomplished through scatter point sets and interpolation. TINs, borehole contacts, borehole sample data, grids, and meshes can all be converted to a 2D or 3D scatter point set. When an object is converted to a scatter point set, all scalar datasets associated with the object are copied to the new scatter point set. The datasets can then be transferred from the scatter point set group to other objects of any type using one of the supported interpolation schemes.

Importing/Exporting Datasets A dataset can be exported by right-clicking on the dataset in the Project Explorer and select the Export command. The Save As dialog has three filter types that can be exported:

- Binary Dataset Files (*.dat)
- ASCII Dataset Files (*.dat)
- HDF5 Dataset Files (*.h5)

A dataset can be imported by right-clicking on any of the objects explained above in the Project Explorer and select the Import Dataset command. This command brings up the File Open dialog.

Project Explorer

Datasets are displayed and managed in the Project Explorer. Click here to learn about datasets in the Project Explorer.

Active Dataset

Each object (TIN, Grid, Mesh, or Scatter Point Set) in GMS has a set of values which is designated as the "active dataset." The active dataset is an important part of model visualization in GMS. Each time the display is refreshed, the contours and other display features are generated using the active dataset.

The active dataset and time step are displayed in the Project Explorer. Left-clicking on a solution or dataset in the Project Explorer makes that item "active" and the display is automatically updated in the GMS window. The table below shows the icons displayed next to the active datasets.

	active scalar dataset
	active vector dataset
	active ccf dataset

Dataset Info

Selecting the ***Properties*** command in right click menu on a dataset in the Project Explorer will bring up the Dataset Info Dialog. The components of the dialog are as follows:

Statistics

A set of statistics related to the active dataset is displayed on the left side of the Dataset Info dialog. the statixtics for all time steps can be shown by selecting the *Show all time step stats*. toggle. A histogram of the data values is displayed on the right side. Right-clicking on the plot and selecting the ***Plot Data*** command brings up the X-Axis Precision dialog. This dialog enables users to set the precision of the x-axis of the Histogram.

View/Edit Values

For datasets associated with model solutions, the data values can be displayed in a spreadsheet using the View Values button. For all other datasets, the button label changes to Edit Values and the spreadsheet can be used to change the dataset values.

Active/Inactive Flags

If the currently selected module is the 3D Grid module, the active/inactive status of cells can be changed by selecting the Edit button at the bottom left of the dialog. This brings up the *Active/Inactive Flags* dialog.

Date/Time Display

For a transient dataset, the time values can be displayed in the Time Step Window using either a relative time format (e.g., 100.0) or using a date/time format (e.g., 1/12/1998 3:23:48). The relative times are computed using a reference time that is defined for the model (MODFLOW, FEMWATER etc.). The reference time represents the date/time corresponding to t=0.

Datasets in the Project Explorer

The solutions and datasets associated with a particular object (grid, mesh, scatter point set, etc.) are managed using the Project Explorer.

Datasets

	scalar dataset		**active scalar dataset**
	vector dataset		**active vector dataset**
	ccf dataset		**active ccf dataset**

3 different types of datasets are shown in the Data Tree: scalar, vector, and cff. The icons used to identify the datasets are shown below.

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 GMS supports. Solutions are shown in the Project Explorer as a folder with an lock on it . If a solution is transient then the time steps are displayed in the Times Step Window. Datasets can not be moved into or out of a solution folder. Also, the solution may contain text files such as the *.out and *.glo files produced by MODFLOW. These files can viewed by right clicking on the item and selecting **View File** from the pop up menu, or you can double click on the item.

Clicking on a solution or dataset makes that item "active" and the display is automatically updated in the GMS window.

Folder

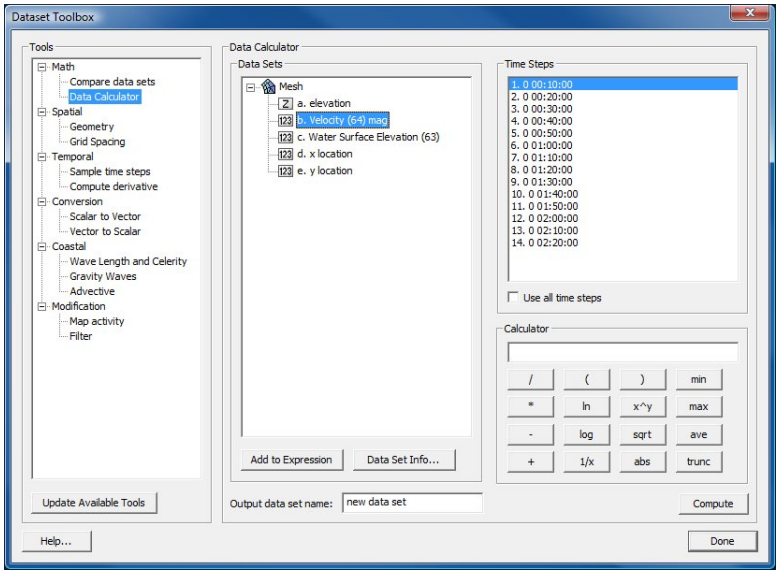
The datasets and solutions are organized by folders. The user can create new folders and move datasets, solutions, and folders to other folders anywhere on the Project Explorer. Folders can be created by right clicking on the certain items in the Project Explorer and selecting **New Folder** in the menu.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

Data Calculator

At a glance

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

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



SMS Data Calculator

Expression Field

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

List of Datasets

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

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

Result Name

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

Operators

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

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

Operating With Transient Datasets

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

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

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

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

Compute Button

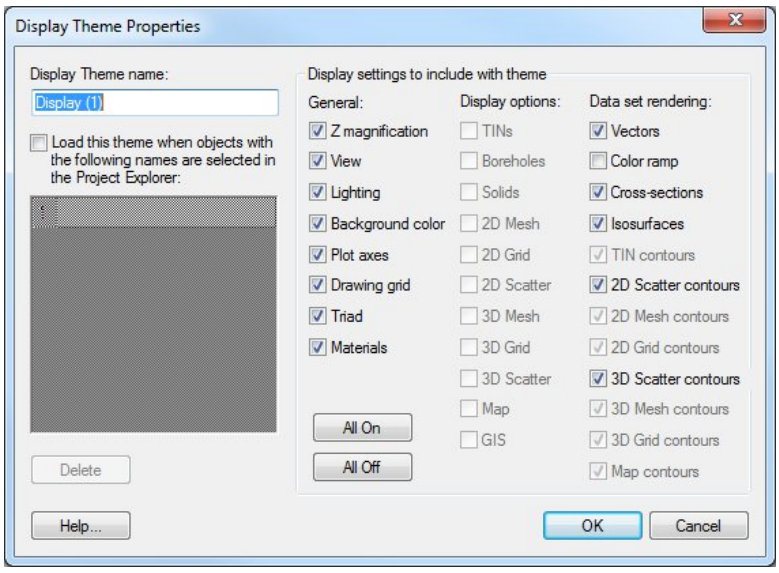
Once an expression is formulated and a name for the resulting dataset has been specified, the expression can be evaluated by selecting the **Compute** button. At this point, the dataset is created and the name of the new dataset should appear in the list of datasets.

Related Links

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

Display Theme

A *Display Theme* is a collection of display options. The user can create a display theme by right-clicking in the blank space of the Project Explorer and selecting the *New* | **Display Theme** menu command. This will bring up the *Display Theme Properties* dialog. In this dialog the user can name the display theme, select which display options to include in the display theme, and enter the names of items that will automatically use the display theme. For more information on display themes see the Display Themes tutorial here.



The *Display Theme Properties* dialog

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. You can also paste your 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 your curve to be entered in date/time format.

Import/Export Buttons The Import and Export buttons allow the user to read in or save 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

Article Sources and Contributors

What is GMS? *Source*: http://www.xmswiki.com/xms/index.php?oldid=41425 *Contributors*: Tdye, Unjedai

The GMS Screen *Source*: http://www.xmswiki.com/xms/index.php?oldid=55354 *Contributors*: Jcreer, Ksmith

Tool Palettes *Source*: http://www.xmswiki.com/xms/index.php?oldid=55326 *Contributors*: Jcreer, Ksmith, Unjedai

Project Explorer *Source*: http://www.xmswiki.com/xms/index.php?oldid=55357 *Contributors*: Jcreer, Ksmith

Tutorials *Source*: http://www.xmswiki.com/xms/index.php?oldid=47824 *Contributors*: Alanlemon, Ksmith, Unjedai

64 bit *Source*: http://www.xmswiki.com/xms/index.php?oldid=55363 *Contributors*: Jcreer, Unjedai

License Agreement *Source*: http://www.xmswiki.com/xms/index.php?oldid=47410 *Contributors*: Jcreer, Ksmith

Registering GMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=37301 *Contributors*: Ksmith, Unjedai

Community Edition *Source*: http://www.xmswiki.com/xms/index.php?oldid=36781 *Contributors*: Ksmith

Graphics Card Troubleshooting *Source*: http://www.xmswiki.com/xms/index.php?oldid=47401 *Contributors*: Jcreer, Ksmith

Reporting Bugs *Source*: http://www.xmswiki.com/xms/index.php?oldid=36306 *Contributors*: Unjedai

File Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=60661 *Contributors*: Jcreer, Ksmith, Unjedai

Edit Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=60682 *Contributors*: Jcreer, Ksmith, Unjedai

Units *Source*: http://www.xmswiki.com/xms/index.php?oldid=60674 *Contributors*: Jcreer, Ksmith, Unjedai

Preferences *Source*: http://www.xmswiki.com/xms/index.php?oldid=55491 *Contributors*: Bdolinar, Jcreer, Ksmith

Materials *Source*: http://www.xmswiki.com/xms/index.php?oldid=60673 *Contributors*: Alanlemon, Jcreer, Ksmith

Material Set *Source*: http://www.xmswiki.com/xms/index.php?oldid=55314 *Contributors*: Jcreer, Ksmith

Coordinate Systems *Source*: http://www.xmswiki.com/xms/index.php?oldid=41675 *Contributors*: Jcreer, Ksmith

Coordinate Conversions *Source*: http://www.xmswiki.com/xms/index.php?oldid=55766 *Contributors*: Jcreer, Ksmith

Projections *Source*: http://www.xmswiki.com/xms/index.php?oldid=53175 *Contributors*: Jcreer, Ksmith, Unjedai

CPP Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=37717 *Contributors*: Ksmith

Geographic Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=36074 *Contributors*: Ksmith, Unjedai

Local Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=38290 *Contributors*: Jcreer, Ksmith

Coordinate Transformation *Source*: http://www.xmswiki.com/xms/index.php?oldid=55493 *Contributors*: Jcreer, Ksmith, Wood

Transform *Source*: http://www.xmswiki.com/xms/index.php?oldid=60684 *Contributors*: Jcreer, Ksmith, Unjedai

Display Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=60681 *Contributors*: Jcreer, Ksmith, Unjedai

Contour Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39633 *Contributors*: Jcreer, Ksmith, Unjedai

Animations *Source*: http://www.xmswiki.com/xms/index.php?oldid=55596 *Contributors*: Jcreer, Ksmith, Unjedai

Color Ramp *Source*: http://www.xmswiki.com/xms/index.php?oldid=59492 *Contributors*: Jcreer, Ksmith

Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=55652 *Contributors*: Jcreer, Ksmith, Unjedai

Drawing Grid Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=55616 *Contributors*: Jcreer, Ksmith

Vectors *Source*: http://www.xmswiki.com/xms/index.php?oldid=55630 *Contributors*: Alanlemon, Jcreer, Ksmith

Lighting Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=46709 *Contributors*: Cdickson, Jcreer, Ksmith

Plot Axes *Source*: http://www.xmswiki.com/xms/index.php?oldid=55632 *Contributors*: Jcreer, Ksmith

Annotations *Source*: http://www.xmswiki.com/xms/index.php?oldid=56285 *Contributors*: Jcreer, Ksmith

CAD Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=55639 *Contributors*: Jcreer, Ksmith

Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=55661 *Contributors*: Jcreer, Ksmith

Datasets *Source*: http://www.xmswiki.com/xms/index.php?oldid=55299 *Contributors*: Jcreer, Ksmith

Data Calculator *Source*: http://www.xmswiki.com/xms/index.php?oldid=55305 *Contributors*: Jcreer, Jdhowlett, Ksmith

Display Theme *Source*: http://www.xmswiki.com/xms/index.php?oldid=55743 *Contributors*: Jcreer, Ksmith

XY Series Editor *Source*: http://www.xmswiki.com/xms/index.php?oldid=30324 *Contributors*: Jdhowlett

Image Sources, Licenses and Contributors

Image:Vertical_Displacement_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Vertical_Displacement_b.png *License*: unknown *Contributors*: Bdolinar
Image:fort_buchanan_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Fort_buchanan_b.png *License*: unknown *Contributors*: Unjedai
Image:geos3d_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Geos3d_b.png *License*: unknown *Contributors*: Unjedai
Image:Icon_TIN.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_TIN.gif *License*: unknown *Contributors*: Cballard
Image:Icon_BH.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_BH.gif *License*: unknown *Contributors*: Cballard
Image:Icon_Solid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_Solid.gif *License*: unknown *Contributors*: Cballard
Image:Icon_2Dmesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_2Dmesh.gif *License*: unknown *Contributors*: Cballard
Image:Icon_2Dgrid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_2Dgrid.gif *License*: unknown *Contributors*: Cballard
Image:Icon_2Dscat.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_2Dscat.gif *License*: unknown *Contributors*: Cballard
Image:Icon_3Dmesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_3Dmesh.gif *License*: unknown *Contributors*: Cballard
Image:Icon_3Dgrid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_3Dgrid.gif *License*: unknown *Contributors*: Cballard
Image:Icon_3Dscat.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_3Dscat.gif *License*: unknown *Contributors*: Cballard
Image:Icon_map.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_map.gif *License*: unknown *Contributors*: Cballard
Image:Icon_GIS.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_GIS.gif *License*: unknown *Contributors*: Cballard
Image:GMS_screen.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GMS_screen.jpg *License*: unknown *Contributors*: Ksmith
Image:Right_Click_GW.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Right_Click_GW.jpg *License*: unknown *Contributors*: Ksmith
Image:macros.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Macros.gif *License*: unknown *Contributors*: Cballard
Image:Tool_Pan.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Pan.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Zoom.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Zoom.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Rotate.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Rotate.GIF *License*: unknown *Contributors*: Jhandy
Image:tool_measure.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_measure.png *License*: unknown *Contributors*: Unjedai
Image:MiniGridPlot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:MiniGridPlot.gif *License*: unknown *Contributors*: Cballard
Image:MiniGridToolbar.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:MiniGridToolbar.png *License*: unknown *Contributors*: Unjedai
Image:Modules_ToolBar.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modules_ToolBar.gif *License*: unknown *Contributors*: Cballard
Image:Tree_General.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_General.jpg *License*: unknown *Contributors*: Ksmith
Image:dlg_register_gms.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dlg_register_gms.png *License*: unknown *Contributors*: Unjedai
Image:dlg_gms_ce_warning.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dlg_gms_ce_warning.png *License*: unknown *Contributors*: Unjedai
Image:gms_ce_watermark.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gms_ce_watermark.png *License*: unknown *Contributors*: Unjedai
Image:Material_Tools.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Material_Tools.png *License*: unknown *Contributors*: Unjedai
Image:material_set.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Material_set.png *License*: unknown *Contributors*: Unjedai
Image:material_set_in_project_explorer.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Material_set_in_project_explorer.png *License*: unknown *Contributors*: Unjedai
Image:ellipsiodcoordsys.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ellipsiodcoordsys.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys1.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys2.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys3.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys4.jpg *License*: unknown *Contributors*: Ksmith
Image:TransformDialogGMS.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:TransformDialogGMS.png *License*: unknown *Contributors*: Unjedai
Image:Hide.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hide.PNG *License*: unknown *Contributors*: Jhandy
Image:Show.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Show.PNG *License*: unknown *Contributors*: Jhandy
Image:Isolate.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Isolate.PNG *License*: unknown *Contributors*: Jhandy
Image:PlanView.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:PlanView.PNG *License*: unknown *Contributors*: Jhandy
Image:FrontView.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:FrontView.PNG *License*: unknown *Contributors*: Jhandy
Image:SideView.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:SideView.PNG *License*: unknown *Contributors*: Jhandy
Image:OrthoView.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:OrthoView.PNG *License*: unknown *Contributors*: Jhandy
Image:logcont.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Logcont.jpg *License*: unknown *Contributors*: Ksmith
Image:dispopts.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dispopts.gif *License*: unknown *Contributors*: Jdhowlett
Image:Display_TIN.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_TIN.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Borehole.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Borehole.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Solid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Solid.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_2DMesh.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_2DMesh.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_2DGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_2DGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_2DScatterSet.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_2DScatterSet.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_3DMesh.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_3DMesh.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_3DGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_3DGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_3DScatterSet.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_3DScatterSet.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Map.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Map.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_GIS.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_GIS.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_CrossSection.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_CrossSection.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Lighting.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Lighting.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Axes.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Axes.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_DrawingGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_DrawingGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:DrawingGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:DrawingGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:LightingDialog.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:LightingDialog.jpg *License*: unknown *Contributors*: Ksmith
Image:Axes.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Axes.GIF *License*: unknown *Contributors*: Jhandy
Image:AnnotationTree.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:AnnotationTree.jpg *License*: unknown *Contributors*: Djordan
Image:Scale.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scale.jpg *License*: unknown *Contributors*: Sramiah
Image:North-arrow.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:North-arrow.jpg *License*: unknown *Contributors*: Ksmith
Image:xsect.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect.png *License*: unknown *Contributors*: Unjedai

Image:Xsect_Sample.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect_Sample.png *License*: unknown *Contributors*: Unjedai
Image:Xsects_3dg.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsects_3dg.png *License*: unknown *Contributors*: Unjedai
Image:Xsect_3dm.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect_3dm.png *License*: unknown *Contributors*: Unjedai
Image:CrossSectionFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:CrossSectionFolder.GIF *License*: unknown *Contributors*: Jhandy
image:x2cov_1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:X2cov_1.png *License*: unknown *Contributors*: Unjedai
image:x2cov_2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:X2cov_2.png *License*: unknown *Contributors*: Unjedai
Image:Tree_DataSetActive.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_DataSetActive.gif *License*: unknown *Contributors*: Cballard
Image:Tree_VectorDataSetActive.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_VectorDataSetActive.gif *License*: unknown *Contributors*: Cballard
Image:Tree_CCFDataSetActive.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_CCFDataSetActive.gif *License*: unknown *Contributors*: Cballard
Image:Tree_DataSet.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_DataSet.gif *License*: unknown *Contributors*: Cballard
Image:Tree_VectorDataSet.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_VectorDataSet.gif *License*: unknown *Contributors*: Cballard
Image:Tree_CCFDataSet.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_CCFDataSet.gif *License*: unknown *Contributors*: Cballard
Image:Tree_Folder_Lock.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_Folder_Lock.gif *License*: unknown *Contributors*: Cballard
Image:SMS_Data_Calculator.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Data_Calculator.jpg *License*: unknown *Contributors*: Jcreer, Jdhowlett
File:GMS Display Theme.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GMS_Display_Theme.jpg *License*: unknown *Contributors*: Jcreer